SYNTEC TECHNOLOGY CO.,LTD.

CNC Alarm Manual.

匯出日期：2023-10-02
修改日期：2023-08-30

- Operation Alarm -OP
- System Alarm -SYS
- Axial Motion Alarm -MOT
- Spindle Alarm - SPD
- Syntax Compiler Alarm - COM
- Program Execute Error Alarm - COR
- Background Execute Alarm - BGND
- ROT Turret Alarm - ROT
- ATC Alarm - ATC
- Serial PLC Axis Alarm – SERIALPLCAXIS
- SRI Alarm -SRI
- MACRO Alarm - MACRO
- Device Alarm - STATION
- Mechanical Logic Control Alarm - MLC
- Laser Marking Alarm - LASERCTRL
- CIP Alarm - CIP
- ENIP Alarm - ENIP
- Extended Software Stroke Limit Alarm - EXTSTROKELIMIT
- Setpoint Axis Alarm - SETPOINTAXIS
- Routine Script-RS
- Alarm Class Table

SYNTEC

1 Driver Alarm Link

Brand	Communication	Alarm Type	Alarm Doc Link	Remark
SYNTEC	M2	Drv_M2_SYNTEC	Driver Alarm Manual	
	M3	Drv_M3_SYNTEC		
YASKAWA	M2	Drv_M2_YASKAWA		
	M3	Drv_M3_YASKAWA		
DELTA	ECAT	Drv_ECAT_DELTA	Manual Download Link	
PANASONIC	ECAT	Drv_ECAT_PSONIC_ALM	Manual Download Link	
	RTEX	Drv_RTEX_PSONIC_ALM	Manual Download Link	
SERVOTRONIX	ECAT	Drv_ECAT_SVTRONIX_ALM	Manual Download Link	

SYNTEC

2 Operation Alarm -OP

Alarm ID	OP-001	Alarm Title	Axis board parameter setting error or abnormal hardware
Description	Controller does not detect the hardware interrupt signal (old axis board: one interpolation interval sends one signal. new axis board: 0.5ms sends one signal.)		
Possible Cause	1. Controller parameters setting error. 2. CPU BIOS setting error. 3. Axis board error or DIP switch setting error. 4. Axis board and ISA SLOT have poor contact or axis board oppressed ISA SLOT. 5. Memory too low (less than 1MB). 6. Controller failed to establish communication with the drive (other than Yaskawa, Syntec). (Other driver brand's manufacturer code must be able to read via the communication command, the model format must be start with "S3P_")		
Solution	1. Check whether controller Pr1 *Motion board base address setting is set according to hardware specifications (refer to application manual parameter explanation). 2. Check whether BIOS' IRQ11 setting is Legacy ISA. 3. Check DIP switch of axis board. 4. Check if the 1st axis board's disconnection signal ISR has short circuit (other axis boards must open). 5. Replace the Axis board to avoid the poor contact due to dirt and foreign foreign debris between axis board and ISA slot. 6. If fail to set up communication, please ask driver vendor to contact with Syntec OEM if needed, and will cooperate to fix driver software.		

Alarm ID	OP-002	Alarm Title	Parameter storage access failure, system halt
Description	System fails to set up PARAM.DAT and PARAM.LKN. System parameter file name is PARAM.DAT. And will save in \DiskC\OpenCNC\Data. System parameter backup file name is PARAM.LKN, and will save in \DiskC\OpenCNC\LKN. When CNC can't find or both normally use and backup system parameter file are damaged while booting, system will reset these two files according to parameter default. Note: If system is x86, \DiskC is in first piece of CF card; if system is ARM, \DiskC is in first piece of Flash.		

Possible Cause	1. x86 System: 1st peice of CF card is out of action. 2. ARM System: 1st piece of Flash is out of action.		
Solution	x86 System: 1. Unplug controller first CF card, and plug CF card's card reader into personal computer. Execute driver repairmen to CF card on personal computer and recopy backup parameter PARAM.DAT into controller dictionary in \DiskC\OpenCNC\Data. 2. Change first CF card. (Please notice first piece of CF card need to install boot system and CNC core software.) ARM System: 1. Please contact Syntec OEM.		
Alarm ID	**OP-003**	**Alarm Title**	**Parameter storage access failure, system halt**
Description	System fail to set up REGISTRY.DAT、REGISTRY.LKN and REGISTRY.MIR. Process data file name when use normally is REGISTRY.DAT, exist in dictionary on \DiskA\OpenCNC\USER. First backup process data file name when is REGISTRY.LKN, exist in dictionary on \DiskA\OpenCNC\LKN. Second backup process data file name when is REGISTRY.MIR, exist in dictionary on \DiskA\OpenCNC\MIR. When it can't find or both normally use and backup system parameter file are damaged at boot time, system will reset these three files according to parameter default. Note: If system is x86, \DiskA is in first piece of CF card. Is system is ARM, \DiskA is in first piece of Flash.		
Possible Cause	1. x86 system: second CF card malfunction. 2. ARM system: second Flash card mulfunction.		
Solution	1. x86 system: If alarm kept happening, please backup user data into new CF card and change the second piece of CF card. 2. ARM system: If alarm kept happening, please backup user data and format DiskA then restore user data.		
Alarm ID	**OP-004**	**Alarm Title**	**Machining data loss, re-calibrate before machining**

Description	Once booted, controller will re-load the system registry file backed up before last power off into memory. After loading, if controller discovers the last shutdown state is busy or if the register file and the first backup file are damaged, one alarm will be sent to the user. The name of registry file is REGISTRY.DAT, saved with file path D:\CNC\USER. The first backup registry file is REGISTRY. MIR, saved with file path D:\CNC\ MIR. The second backup file of registry is REGISTRY. LKN, saved with file path D:\CNC\ LKN Note: If system is x86, DiskA is in first piece of CF card. If system is ARM, DiskA is in first piece of Flash.
Possible Cause	1. Shut down or power off controller in busy state. 2. File REGISTRY.DAT is damaged. 3. x86 system : second piece of CF card is broken. ARM system : second piece of Flash is broken.
Solution	1. If all machining data is confirmed to be normal, cut off power and reboot controller and driver to clear alarm. 2. x86 system: If alarm kept happening, please backup user data into new CF card and change the second CF card. 3. ARM system: If alarm kept happening, please backup user data and format DiskA then restore user data.

Alarm ID	**OP-005**	**Alarm Title**	**I/O transmission error**
Description	When PIO5 of Watch Dog fails, the system will display warning		
Possible Cause	1. Pr5 setting doesn't match hardware structure, for example: a. 22A doesn't dispose HK adapter plate. b. 10B/10F doesn't configure two RIO when Pr5=8. 2. IO relative module (PIO5/RIO) malfunction. 3. Not supplied external 24V Voltage to RIO or RIO cable malfunction. 4. Motherboard overheats (Check System Data No. 39) 5. Controller ground cable is influenced by noise. 6. Motherboard malfunction.		
Solution	1. Set correct Pr5 parameter value according to hardware configuration. 2. Change IO relative model (PIO5/RIO). 3. Supply power to RIO correctly and check RIO cable. 4. Check if electic cabinet fan works normally. 5. Check machine has grounding correctly. 6. Change Motherboard 7. If HK adapter board is not used, please refer to custom I/O comparison table manual. Set Pr5 to 100 and set IO point (HK, MPG, PANEL-EX) on adapter board to disable (-1). 8. According to 7. If RIO is not used, please sets RIO point to disable. (-1)		

Alarm ID	**OP-006**	**Alarm Title**	**Permanent storage data CRC check failure, re-calibrate before machining**
Description	IO card hardware error.		
Possible Cause	1. IO card malfunction. 2. Fram CRC error. 3. Fram has not initialized. 4. For 10F, 200 Series controller restore backup data when CRC error. 5. Wrong setting of DIP switch when using two pieces of PIO5.		
solution	1. Change IO card. 2. Please upgrade to 10.112.95, 10.114.29 or later version. 3. Reboot controller and driver. 4. Correctly set PIO5 dip switch.		

Alarm ID	**OP-007**	**Alarm title**	**Low memory, memory space is less than 1M**
Description	System memory less than 1MB.		
Possible Cause	Developer designed controller HMI with too much components, graphs or pictures, and leading to memory less than 1MB.		
Solution	Designing controller HMI need to reduce excessive components. Use effective variable and component can avoid consuming too much memory.		

Alarm ID	**OP-008**	**Alarm title**	**Fatal low memory, memory space is less than 100K**
Description	System memory is less than 100k.		
Possible Cause	Developer designed controller HMI with too much components, graphs or pictures, and leads to memory less than 100 KB.		
Solution	Reduce excessive components when designing controller HMI. Use effective variable and component can avoid consuming too much memory.		

Alarm ID	**OP-009**	**Alarm title**	**The number of interpolation loss times is bigger than 100**

Description	System Data No.54 (Interpolation delay times) is more than 100. Interpolation delay: Axis card doesn't send the command calculated by last interpolation time, the system will ignore the interpolation value to avoid unsent command value be covered and cause order missing.
Possible Cause	1. Axis card malfunction. 2. System resource runs out. 3. The setting of interpolation time interval (Pr3203) is too small.
Solution	1. Replace the hardware of axis card. 2. Replace motherboard. 3. Increase Pr3202 interpolation time interval set value.

Alarm ID	**OP-010**	**Alarm title**	**Axis Group setting error**
Description	When no axis belongs to CNC main system path.		
Possible Cause	Parameter setting error Ex: Assume that the machine exists XYZ axis, each associated axis group is set as follows: Pr701=1, Pr702=4, Pr703=4, and Pr731=3. CNC main system path has first path, second path and third path. But without axis belongs to second path, controller will issue this alarm.		
Solution	Please check Pr701~Pr720 axis affiliated path and Pr731 CNC main system path number setting value.		

Alarm ID	**OP-011**	**Alarm title**	**【Pause point offset error】**
Description	When use multi-path in machining, with axis belongs to Plural path in different workpiece coordinate system. If user operators inappropriately, when modifying workpiece coordinate during block stop, axis offset will be different in each path. This condition will cause controller system can't send out correct moving command.		
Possible Cause	1. When changing axis workpiece coordinate system, user doesn't revise that axis all affiliated path. 2. When changing all axis affiliated paths' workpiece coordinate, each affiliated changing value is different.		
Solution	1. Please make sure affiliated path, that axis's workpiece coordinate system all changed. (Refer to Pr701~Pr702 axis affiliated path) 2. Please make sure that axis workpiece coordinate system revision value will all be the same in different affiliated path.		

Alarm ID	OP-012	Alarm title	CF card set by Pr3219 error，check CF card or system setting value
Description	When CNC uses Dos, if we set Pr3219 equal to 3 and second CF card (user data) is not inserted, system mis-used REGISTRY.DAT data in order to avoid CF card malfunction.		
Possible Cause	1. The system doesn't have the 2nd CF card or cannot read the 2nd CF card. 2. The system has two CF cards, but both CF card has only one partition disk.		
Solution	1. Check whether the 2nd CF card exists or there is CF card reading error. 2. Set the first card to the default state, which is having wo partition disks condition. 3.Set parameter 3219 to 0 and all use C:		

Alarm ID	OP-013	Alarm title	Software version and model can't driver hardware, call your vender!
Description	Controller installed software version that hardware doesn't support. Before eliminating alarm, if user move the machine, controller will issue following error alarm.		
Possible Cause	1. Controller install software version that hardware doesn't support. a. EZ milling machine doesn't support version after 10.114. b. 10A/B doesn't support version after 10.116.10. c. Software doesn't support the FPGA device. d. Pr9 *Servo board type doesn't support the FPGA device. 2. Controller has changed both motherboard or CF card and IO card at the same time, so it causes hardware information incompatible. Machine code dropped to EZ so it can't support original software version.		
Solution	1. Install software version that hardware supports. a. EZ milling machine standard configures 10.112 version. If user want to use higher version, user need to open software option function Option1 [EZ2/3/4 upgrade 10A software function]. b. 10A/B because of memory issue, if user wants to use higher version, recommend to change into 10F. c. Please contact and provide backup file to Vendor-Syntec. d. Please contact and provide backup file to Vendor-Syntec. 2. Please contact Vendor-Syntec to solve hardware information incompatible problem and reset controller serial code.		

Alarm ID	OP-014	Alarm title	WinCE option not enabled, call your vender!

Description	Controller installs WinCE version software, and WinCE function option is not activated, so axis card can't send command which cause the controller issue following error alarm after machining start.
Possible Cause	Option software function Option2. "Controller upgrades WinCE system" is not activated.
Solution	1. Re-install Dos version. 2. Please contact Syntec OEM, to open option software function Option2 "Controller upgrades WinCE system" 3. Super controller doesn't have this error.

Alarm ID	**OP-015**	**Alarm title**	**【The output contacts are overloaded; please check for any abnormal wiring】**
Description	Controller detects PLC O point overloading. Sixteen O point as a group. When one O point in that group overloads 350 mA will send out alarm of O point in that range is overloading.		
Possible Cause	1. Loads more than 350mA. 2. Output connection point is short circuit. 3. Machine has electricity leakage 4. machine loading device is aged.		
Solution	1. Verify load power consumption. 2. Verify that the output is not short-circuited. 3. Machine grounding. 4. Replace load equipment for testing.		

Alarm ID	**OP-016**	**Alarm Title**	**Straightness compensation function configuration error**
Description	Straightness accuracy offset function current supplies at most five groups of moving axis and offset axis setting. In each group, the compensation position of the "at least two and at most twenty" moving axis and the corresponding offset axis' compensation amount must be entered. Therefore, when the system detects that the straightness compensation function parameter setting error, the controller issues this alarm.		
Possible Cause	1. Multi groups of moving axis are corresponding to same offset axis simultaneously. 2. Moving axis is also offset axis. 3. Activate straightness accuracy offset's moving axis and offset position descends. 4. Activate straightness accuracy offset's moving axis and offset position is less than two positions.		

Alarm ID	OP-016	Alarm Title	Straightness compensation function configuration error
Solution	1. Check Pr1481~Pr1490 Straightness offset moving axis number and compensating axis number a. map multiple sets of moving axes to different offset axis. b. configure moving axis not as an offset axis. 2. Check Pr7001~Pr7025 Straightness offset moving axis position and Pr7051~Pr7125* Position of each group's compensation point in moving axis pitch compensation form a. fill in increasing compensation position order. b. fill in at least two and at most twenty compensation positions. 3. Expand 2D linear compensation function (20%), supported versions: 10.118.12E, 10.118.15.		

Alarm ID	OP-017	Alarm Title	Port number or I point mapped to handwheel error
Description	Pr2021~Pr2027 setting error.		
Possible Cause	1. Pr2021~Pr2030 *Port number or register number for MPG port number setting is not supported. 2. Pr2024~Pr2027 *Continued I-Bit no. for MPG two continues I points setting is not supported. 3. Pr13 *Number of servo card setting error.		
Solution	1. Refer to the description in the manual for correct configurations according to the controller code. 2. Check whether Pr13 *Number of servo card setting is correct or not.		

Alarm ID	OP-018	Alarm Title	Port configuration error
Description	Pr21~Pr40 * Axis corresponding axis card port number or Pr1621~Pr1630 Axis ID or Port Number for Spindle setting error. Serial PLC Axis, ROT port number setting error. Pr3261~Pr3263 *M3-IO Station No setting error.		

SYNTEC

Alarm ID	OP-018	Alarm Title	Port configuration error
Possible Cause	1. Pr21~Pr40 axis port number setting is not supported. 2. Pr21~Pr40 axis port number setting is repeated, only number 17, 18 (virtual axis) can be repeated. 3. When using variable-frequency spindle (Pr1791~Pr1800 sets as 0), Pr1621~Pr1630 spindle port number setting is not supported. 4. When using SRI DA spindle (Pr1791~Pr1800 sets as 7), Pr1621~Pr1630 register number setting is not at User-defined 5. Serial Bus controller uses variable-frequency spindle (Pr1791~Pr1800 sets as 0). Variable-frequency spindle port number (Pr1621~Pr1630) can't have conflict with other serial bus controller port number. 6. Pr3261~Pr3263 port number setting has conflict with axis (Pr21~Pr40, Pr1621~Pr1630). 7. Serial PLC axis and ROT port number is not supported. 8. Pulse controller which is 10.116.54L, 10.118.0G, 10.118.7 and above version, if port number hardware setting is not supported, this alarm will be triggered. 9. Pr13 *Number of servo card setting error Note: Port number setting can refer to Controller Model Axial Port Number Setting List.		
Solution	1. Refer to the Description in the parameter manual to configure the correct port number. 2. Configure the port number in Pr1621~Pr1630 which is not conflict with other serial servo ports, set to 19 is recommended. 3. When using SRI DA spindle (Pr1791~Pr1800 sets as 7), configure the port number in Pr1621~Pr1630 in User-defined region. 4. Check whether Pr3261~Pr3263 and Pr21~Pr40, Pr1621~Pr1630 conflict with each other, and correct the conflict. 5. For pulse controller version 10.116.54L, 10.118.0G, 10.118.7 and above, if user wants to use virtual axis, please set as 17/18, or use G10 L800 to specify the virtual axis. 6. Check if Pr13 *Number of servo card is set correctly.		

Alarm ID	OP-019	Alarm Title	Prohibit to switch the metric and imperial systems in machining/not ready state
Description	When system is Busy or Not Ready, can't switch into Metric and Imperial units via HMI operation.		
Possible Cause	When system is Busy or Not Ready, switch into Metric and Imperial units via HMI operation.		
Solution	Avoid to switch the metric and imperial systems in Busy or Not Ready state via HMI operations.		

Alarm ID	OP-020	Alarm title	M code setting conflict

Description	1. When the following M code related parameter setting has repeated M code and cause setting conflict. a. Pr3599 *Register M code for Advanced Look-Ahead mode b. Pr3600 *Register custom Macro interrupt M code c. Pr3601~Pr3610 *M code Macro call registry d. Pr3804 Part count M code e. Pr3741, Pr3743 Rotary axis auxiliary brake M code 2. When Pr3599 *Register M code for Advanced Look-Ahead mode set as 1, 2, 30, 98, 99, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 3. When Pr3600 *Register custom Macro interrupt M code sets as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 4. When Pr3741, Pr3743 Rotary axis auxiliary brake M code set as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 5. When Pr3763 Braking M code of Indexing Axis set as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198.
Possible Cause	As stated above.
Solution	After modifying the above parameter settings so that no conflict exist, reboot the controller and driver.

Alarm ID	**OP-021**	**Alarm title**	**Use last known user data, re-calibrate user data before machining**
Description	The controller's machine data, tool table and scan table information are stored in Registry.dat, ToolTable.dat and ScanTable.dat respectively, and each has three backups of .mir, .lkn and .cdb. If either .dat and .mir are abnormal and cannot be used, the system will use .lkn or .cdb. If either .dat and .mir are abnormal and cannot be used, the system will use .lkn or .cdb to restore the data. If the Registry is abnormal, the machining file name will be cleared.		
Possible Cause	1. X86 system (with CF card): It is possible that the CF card is damaged and the machine data, tool table or scan table file is damaged. 2. ARM system (without CF card): DISKA damage results in damage to machine data, tool table or scan table data. 3. The following is a list of possible file damage situations and their restoration sources. a. Machine data and other data are damaged, restored from DISKA backup. b. Tool table data is damaged, and it is restored from DISKA backup. c. Scan table data damaged, restored from DISKA backup. d. Machine data and other data are damaged, restored from DISKC backup. e. Tool table data is damaged, and it is restored from DISKC backup. f. Scan table data damaged, restored from DISKC backup.		
Solution	1. Reset or check whether the machine data, tool table and scan table data are correct. 2. Scan CF card disk. 3. Reboot controller and driver.		

Alarm ID	OP-022	Alarm title	User data rebuild, re-calibrate user data before machining
Description	None of Registry, ToolTable and ScanTable in the controller has .dat, backup files .mir, .lkn and .cdb. User data is rebuilt.		
Possible Cause	1. X86 system (with CF card): CF2 card may has been replaced or formatted, causing user's original data and backups being cleared out. 2. ARM system (without CF card): DISKA and DISKC are damaged or formatted, causing user's original data and backups being cleared out. 3. The possible file corruption situations are listed below. a. Machine data, other data, and their relative backups are all destroyed. b. Tool table data and its backups are all destroyed. c. Scan table data and its backups are all destroyed.		
Solution	1. Reset the machine data, tool table and scan table data, and then reboot controller and driver. 2. If this alarm still cannot be eliminated after reboot, format CF2/DISKA and CF2/DISKC. 3. If alarm still cannot be eliminated after format, CF2/DISKA and CF2/DISKC are malfunction and need repair.		

Alarm ID	OP-023	Alarm title	Power break in machining, re-calibrate before machining
Description	When new machining data need to be stored during Cycle Start, the controller will store a state flag into Fram or REGISTRY.DAT, and it will be removed when the specified data have already been stored. While controller booting, if this flag is not cleared, this alarm will be issued.		
Possible Cause	The power is turned off when new machining data not yet been stored during machining process.		

SYNTEC

Solution	1. Please re-calibrate your machining data. For example, a. Check the Imperial/Metric (G70/G71) unit selection. b. Check the home point, offset, and rotational angle of the coordinate system corresponding to the G92/G92.1/G68.2/G68.3 command. c. Check the tool number and the compensation amount of the tool length compensation command (G43). d. Check the settings of the 2nd Software Stroke Limit command (G22). e. Check the values stored in the the persistent variables and R bit, i.g. Macro @656~@1999、R100~R255、R10000~R10999. f. For more details, please refer to Registry Data/L Variable List. 2. After completing step 1, following the following step by version : a. For software version 10.118.40B, 10.118.41 or above, please trigger "**reset**" to clear alarm. b. For other versions, please restart the controller to clear alarm. 3. Way to avoid: Avoid switching the power off during machining. Or, trigger C47 to store the machining data before switching the power off.

Alarm ID	**OP-024**	**Alarm title**	**Machine data file write fail, re-calibrate before machining**
Description	Before shutting down controller, the last writing of REGISTRY.DAT or REGISTRY.MIR file is unsuccessful.		
Possible Cause	1. CF card is damaged. 2. REGISTRY.DAT or REGISTRY.MIR files are set as read-only. 3. System file processing device's resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

Alarm ID	**OP-025**	**Alarm title**	**Machine data file write fail many times, re-calibrate before machining**
Description	Before shutting down controller, the accumulated unsuccessful of writing REGISTRY.DAT or REGISTRY.MIR file exceed 100 times.		
Possible Cause	1. CF card damaged. 2. REGISTRY.DAT or REGISTRY.MIR file is set to read-only. 3. System's file processing device resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

Alarm ID	OP-026	Alarm title	Machine data file fault error, re-calibrate before machining and do scan disk
Description	Before shutting down controller, unsuccessful of writing REGISTRY.DAT or REGISTRY.MIR file exceed 100 times continuously happened.		
Possible Cause	1. CF card damaged. 2. REGISTRY.DAT or REGISTRY.MIR file is set to read-only. 3. System's file processing device resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

Alarm ID	OP-027	Alarm Title	CNC main system axis group to be executed unselected
Description	No CNC main system path is chosen and cycle start triggered.		
Possible Cause	Please check R21 value: 1. Chosen path not exists or not belongs to CNC main system path. 2. No CNC main system path is chosen (Only happens when R21.11 status is ON).		
Solution	1. Check if any logic error when PLC triggers R21 option, which results in the correct path is not selected successfully. 2. Select an existing CNC main system path via R21.		

Alarm ID	OP-028	Alarm Title	Prohibit CNC main system axis group switch in machining
Description	When CNC is under Not Ready state, change CNC main system path that was going to be executed and triggers cycle start.		
Possible Cause	Please check R21 value, whether it is different from original process condition. (Change the chosen CNC main system path that was going to be executed)		
Solution	Modify R21 to the value in the original machining status, then can continue machining.		

Alarm ID	OP-029	Alarm Title	CNC axis over system support
Description	Assign too much axis that belongs to a normal path (Not Loader or auto labeling path).		

Alarm ID	OP-029	Alarm Title	CNC axis over system support
Possible Cause	Refer to following parameter setting, when the axis number of a normal path exist the limit of system axis number, this alarm will be issued. 1. Pr701~Pr720 *Axial associate axis groups 2. Pr733~Pr734 *Loader axis group 3. Pr737 * Woodworking labeling axis group For example: Use 21MA, standard configuration is 6 axis. If 8 axis is activated (activate software option function Option20 [Loader axis number expansion]/ Option37 [auto labeling axis number expansion]) and configures following parameter settings 1. Pr701~Pr704 sets as 1, number 1~4 axis are first path. 2. Pr705~Pr708 sets as 2, number5~8 axis are second path. 3. Pr733, Pr734 and Pr737 all set as 0, no path is Loader/ auto labeling path. Therefore system has 8 axis, which over 21MA system's axis number limit.		
Solution	Check and correct the following parameter settings: 1. Pr701~Pr720 *Axial associate axis groups 2. Pr733~Pr734 *Loader axis group 3. Pr737 * Woodworking labeling axis group Based on the above example, in case of the 2nd path operation requirements can be fulfilled with the Loader path, Pr733 set as 2 to specify the 2nd path is the Loader path. Only 4 axis belongs to a normal path as system axis, when total axis number not exceed the standard 21MA axis number configuration, alarm can be removed.		

Alarm ID	OP-030	Alarm Title	Debug mode, disable debug mode to resume normal operation.
Description	Active debug mode, Pr3221 set as 1 or 2.		
Possible Cause	As stated above.		
Solution	Disable the debug mode, Pr3221 set as 0.		

Alarm ID	OP-031	Alarm Title	GANTRY robot option not enabled
Description	Pr3201 activate robot function, but software option function Option24 is not activate [Cartesian Coordinate Robot Type]. (Currently only 10.117 version has this alarm.)		
Possible Cause	Pr3201 is different from correspond controller’s machine mode.		

Alarm ID	OP-031	Alarm Title	GANTRY robot option not enabled
Solution	1. Enable Option24 [Cartesian Coordinate Robot Type] for GANTRY type robot. 2. In case of not robot, configure Pr3201 correctly.		

Alarm ID	OP-032	Alarm Title	Mechanism type configuration conflicted
Description	Pr3201 Machine Type setting conflicted.		
Possible Cause	Currently only the following mechanism type support five axis function; 1. Pr3201 sets as 0, close lathe feature. (Use general milling interface) 2. Pr3201 sets as 1, Lathe Habit Type C. 3. Pr3201 sets as 2, Lathe Habit Type A. 4. Pr3201 sets as 3, Lathe Habit Type B. Therefore, open both five axis (Pr3001, Pr3101, Pr5501, Pr5601) and other non-lathe/mill machine tool feature's mechanism type (Pr3201) at the same time, and alarm will be issued to inform user. For example: 1. Activate first group of five axis function (Pr3001 isn't 0. First path default to use first five axis kinematic chain) and first path is not lath/mill machine tool attribute. (Pr3201 in first path is not 0~3) 2. Activate second group of five axis function (Pr3101 isn't 0. Second path default use second five axis kinematic chain) and second path is not lath/mill machine tool attribute. (Pr3201 in second path is not 0~3) Besides, when activate option software function Option 29 (Four axis dedicated Rotate Tool Center Point function (4AXRTCP)), however, and option software function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function) are not activated, but sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 1~3, and this alarm will be issued.		
Solution	Only the milling machine supports the 5-axis models. Configure Pr3201 to 0~3, or configure the mapped 5-axis function parameters (Pr3001, Pr3101, Pr5501 and Pr5601) to 0. [Note] In the versions after the 10.116.54G and 10.118.0D, the lathe can enable RTCP and is limited to 200TB-5. 1. Only lathe/mill machine tool supports five axis machine type. Please sets Pr3201 as 0~3, or sets corresponding five axis function parameter as 0. (Pr3001, Pr3101, Pr5501, Pr5601) 2. Active RTCP for Lathe 200TB-5. Supported version: 10.116.54G, 10.118.0D or above versions. 3. Please sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 4 or 5, or please open software option function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function).		

Alarm ID	OP-033	Alarm Title	Do not set machine position during machining

Description	Using C025 ~ and C230 ~ to set machine position when machining
Possible Cause	Using C025 ~ and C230 ~ to set machine position when machining
Solution	Do not use C025 ~ and C230 ~ to set machine position when machining.

Alarm ID	**OP-041**	**Alarm Title**	**Machining file inexistent in the specific path**
Description	System can't find ECAM.xml in designated path, so can't activate cam function.		
Possible Cause	1. File name is wrong. 2. File location is wrong.		
Solution	Check the ECAM.xml.		

Alarm ID	**OP-042**	**Alarm Title**	**Machining program data error, please check NC File**
Description	ECAM.xml file format is wrong.		
Possible Cause	Self edit ECAM.xml, and format is wrong.		
Solution	Check the ECAM.xml.		

Alarm ID	**OP-043**	**Alarm Title**	**ECAM group more than 8 groups**
Description	Too much ECAM group.		
Possible Cause	Self edit ECAM.xml, sets up too many ECAM groups.		
Solution	Check the graphic dialog input to ensure the configured number of CAM groups is appropriate.		

Alarm ID	**OP-044**	**Alarm Title**	**CAM curves more than 32 lines**
Description	Too much cam curve line.		
Possible Cause	Self edit ECAM.xml, sets up too many cam curve line.		

Alarm ID	OP-044	Alarm Title	CAM curves more than 32 lines
Solution	Check the graphic dialog input to ensure the configured number of CAM curves is appropriate.		

Alarm ID	OP-045	Alarm Title	CAM curve speed planning failed
Description	Fail to plan cam curve speed.		
Possible Cause	1. Cam curve design doesn't match movement parameter. 2. Movement parameter setting is not reasonable. 3. Doesn't execute re-read after fixing insertion point through R value.		
Solution	1. Check if the planned CAM curve line is reasonable under the limit of machine motion configurations. 2. Correct the machine motion parameter within the reasonable range. 3. Re-read R values and set C300 to On.		

Alarm ID	OP-046	Alarm Title	ECAM – no driven axis name
Description	Electronic cam-Can't find axis name.		
Possible Cause	Pr321~Pr340 axis name is not set.		
Solution	Check and correct Pr321~340 setting.		

Alarm ID	OP-047	Alarm Title	Use non self-defined R value or too many R values.
Description	R value used is out of self-defined range or insertion point sets over 100 R value.		
Possible Cause	1. Use non self-defined R value. 2. Self-edit ECAM.xml, insertion point sets over 100 R value.		
Solution	Check ECAM.xml, if the R value of workpiece counts and the insert point are out of user self-defined range, and whether insertion point used over 100 R value.		

Alarm ID	OP-048	Alarm Title	Curve insert point re-read R value failed
Description	When re-read R value, electronic cam is under coupling mode.		
Possible Cause	Electronic cam group is under coupling mode.		
Solution	Release all CAM coupling and set S301~S308 to Off.		

Alarm ID	OP-050	Alarm Title	SRI status error
Description	SRI condition abnormal.		
Possible Cause	Please check the followed SRI alarm, and refer to SRI Alarm -SRI.		
Solution	Refer to the solution of followed SRI alarm. After resolving the issue, reboot controller and driver.		

Alarm ID	OP-051	Alarm Title	M3-IO communication error
Description	M3-IO communication abnormal and cause watchdog has accumulated over 100 errors.		
Possible Cause	1. M3 communication cable is loose. 2. Noise interference. 3. Using the laser cruise function without installing the ADD-on card.		
Solution	1. Check if the communication cable is normal. 2. Replace communication cables. 3. Reboot controller and driver. 4. Check whether the laser cruise function is required. If necessary, confirm that the ADD-on card is installed correctly.		

Alarm ID	OP-052	Alarm Title	M3-IO device status error
Description	Controller detects M3-IO device is abnormal through M3 communication.		
Possible Cause	M3-IO card abnormal.		
Solution	1. Check if the M3-IO card is inserted securely. 2. Check the setting of Pr3261~Pr3263. 3. Reboot controller and driver.		

Alarm ID	OP-053	Alarm Title	Serial communication initialization failed
Description	When controller fail to communicate with driver.		
Possible Cause	1. Driver is abnormal. 2. Driver or controller hardware malfunction. 3. When Pr9 sets as EtherCat communication, and driver is not connected. 4. When Pr9 sets as RTEX communication, and driver is not connected. 5. M3 multi-axis driver, the first station is not set.		
Solution	1. Reboot the controller and driver. 2. Replace the driver, if it is not effective, the controller hardware may be malfunctioned. 3. Connect EtherCAT driver. 4. M2 / M3 initialization error code display on D53, please contact the Syntec OEM and provide D53 value. 5. Set the first station, or enlarge the set value of Pr3203 (interpolation time). Note: For EtherCAT communication error, refer to EtherCAT Drive Application Manual..		

Alarm ID	OP-054	Alarm Title	Multi-axis driver initialization failed
Description	Multi-axis driver initialization failed.		
Possible Cause	first servo axis function is not activated when using multi-axis driver and cause initialization failure. Note: This alarm has be removed in 10.116.36P and above version. Multi-axis driver don’t have to activate first servo axis.		
Solution	Activate first servo axis in multi-axis driver, reboot the controller and driver.		

Alarm ID	OP-055	Alarm Title	Drive information reading error
Description	Failed to read all drives' information.		
Possible Cause	1. Communication between controller and driver is not stable, causing the M3 drive message or EtherCAT drive mailbox communication takes too much time. 2. M3 Drive doesn’t support M3 message communication.		
Solution	1. Check the wiring of the communication cable between the controller and the drive and resolve the causes for unstable communication. 2. If it is M3 network, check if the connected drives support the M3 message communication.		

Alarm ID	**OP-056**	**Alarm Title**	**Some axial directions cannot Servo ON. System cannot operates.**
Description	When there is axis servo not servo on, cycle start or other manual function is triggered.		
Possible Cause	1. When power on, axis doesn't complete magnetic pole angle detection and cause servo disabled. 2. Driver power supplication is abnormal or there is driver alarm, and cause servo change from servo on condition into disabled condition. 3. Driver is malfunction.		
Solution	1. After power on, wait for a while then operate the machine. 2. Check if the driver power supply status is normal and if there is driver alarm. 3. Check if the driver is normal.		

Alarm ID	**OP-057**	**Alarm title**	**【Tool retraction function, hand wheel shift only supports R518, R519 set to 0】**
Description	When using G10.6 toll retraction function, R518, R519 must be set as 0, then user can use MPG to move.		
Possible Cause	R518, R519 is not 0.		
Solution	Set R518, R519 to 0.		

Alarm ID	**OP-058**	**Alarm Title**	**Burn mode. All I/O connections to servo is disconnected**
Description	When controller and driver is burning firmware, user can't operate driver's servo axis or IO device.		
Possible Cause	Executed controller remote update driver firmware.		
Solution	Reboot the controller and driver.		

Alarm ID	**OP-059**	**Alarm Title**	**Motion control module loading failed. Reinstall software**
Description	Fail to load because of controller internal file is lost or destroyed.		

Alarm ID	OP-059	Alarm Title	Motion control module loading failed. Reinstall software
Possible Cause	Controller install package is destroyed.		
Solution	Download and re-install the controller installation package.		

Alarm ID	OP-060	Alarm Title	M3-IO device number configuration error
Description	Controller M3-IO device number setting error.		
Possible Cause	When using M3-IO device number repeated.		
Solution	Check if Pr3261~3263 value is repeated.		

Alarm ID	OP-061	Alarm Title	M3 external device status error
Description	Controller M3 external device’s communication condition is abnormal.		
Possible Cause	1. M3 communication cable is loose. 2. Noise interference. 3. M3 external device is abnormal.		
Solution	1. Check if communication cable is normal. 2. Change communication cable. 3. Check if M3 external device is firmly connected. 4. Reboot controller and driver.		

Alarm ID	OP-062	Alarm Title	Customized IO mapping table destroyed
Description	When using custom IO mapping table, there is error in reading files.		

Alarm ID	OP-062	Alarm Title	Customized IO mapping table destroyed
Possible Cause	1. Custom IO mapping table is incomplete, destroyed or missing. 2. Custom IO mapping table version is incompatible with software version. 3. Software 10.118.9 or above version's custom IO supports SRI. If downgrade after setting or backup files into older version, there will be version incompatible issue. 4. Software 10.118.11 or above version's custom IO supports M3-IO expansion card with two DA input (Scan in Interpolation time), if downgrade after setting or backup files into old version, there will be version incompatible issue. 5. Software 10.118.24A, 10.118.25A, 10.118.28N, 10.118.38 or above version's custom IO supports single IO setting, if downgrade after setting or backup files into old version, there will be version incompatible issue.		
Solution	1. Enter the IO configuration page, and restore default value, then re-set IO mapping table. 2. Use Restore function, confirm the backup file is normal and restore.		
Alarm ID	OP-063	Alarm title	【Custom IO mapping table setting conflicts】
Description	There is conflict in IO point setting when using custom IO mapping table.		
Possible Cause	Custom IO mapping table I point or O point setting repeated.		
Solution	1. Enter IO setting screen, modify the conflicting IO points. 2. Reboot controller and driver.		
Alarm ID	OP-064	Alarm Title	Configured as custom IO. Review the IO mapping table.
Description	When Pr5 is set to be using custom IO mapping table.		
Possible Cause	Pr5 set as 100. And change from standard IO into custom IO.		
Solution	The alarm is used for reminder. Reboot controller and driver to remove alarm.		
Alarm ID	OP-065	Alarm Title	Lasermarking status error
Description	Laser marking condition is abnormal.		
Possible Cause	Please check the followed LaserCtrl alarm. And refer to Serial PLC Axis Alarm – SERIALPLCAXIS		

Alarm ID	OP-065	Alarm Title	Lasermarking status error
Solution	Refer to the solution of LaserCtrl alarm. After resolving the issue, reboot controller and driver.		

Alarm ID	OP-067	Alarm title	【The wireless handheld box device is abnormal】
Description	Wireless handheld box external device's communication is abnormal.		
Possible Cause	1. Wireless transmitting device is unplugged. 2. Wireless transmitting device can't be read normally by USB.		
Solution	1. Check whether the wireless transmitter is loose. 2. Press the Reset button and try to reconnect. If communication is normal, this alarm can be removed.		

Alarm ID	OP-068	Alarm title	【The software version does not support this M3-IO device】
Description	1. Driver's firmware version doesn't support hardware information on M3-IO for controller to identify. 2. Controller's software version cannot identify M3-IO device on multi-axis driver.		
Possible Cause	1. Only driver with 2.7.0 or above version support M3-IO device hardware information; Previous version can't get hardware information so doesn't support all M3-IO device, in order to protect machine operation safety. 2. Controller with 10.118.10 or above version has M3-IO device identification protection. Besides following hardware, all will issue this alarm to prevent machine from wrong action. a. 16I 8O b. 32I 32O MPG c. 32I 32O MPG 2DA 3. Pr3261~Pr3263 M3-IO station number setting error.		
Solution	1. Please update driver firmware to 2.7.0 and above version. 2. Please contact Syntec OEM, request for the software version that supports the M3-IO devices and upgrade the software. 3. Change back to the following three M3-IO devices a. 16I 8O b. 32I 32O MPG c. 32I 32O MPG 2DA 4. Refer to Pr3261~Pr3263 *M3-IO Station No and set the M3-IO station number correctly.		

Alarm ID	OP-069	Alarm title	【Over axis limit that interpolation time can support】
Description	Total axis number is over the limit that interpolation time can support.		
Possible Cause	1. Interpolation time Pr3203 is too small. 2. Axis number setting error.		
Solution	1. Interpolation time Pr3203 recommends to set 2000 or lager. Detailed support axis number please refer to Pr3203. 2. Multi-axis axis number please start setting with x000 (x means station number).		

Alarm ID	OP-070	Alarm title	【Product doesn’t support Pr9 setting】
Description	This product doesn’t support Pr9 setting.		
Possible Cause	1. Set Pr9 as 120/121 (RTEX), but hardware doesn’t support it. 2. Set Pr9 as 120/121 (RTEX), but software(versions 10.118.30E, 10.118.32E, 10.118.40I, 10.118.41I, 10.118.45 or above) doesn’t support it. 3. Pr9 setting error.		
Solution	1. Please contact Syntec OEM to know product information. 2. Revert the controller to the version under 10.118.30E, 10.118.32E, 10.118.40I, 10.118.41I or 10.118.45. 3. Refer to Pr9 *Servo board type, set the usable parameter value for the product.		

Alarm ID	OP-071	Alarm title	【Driver power off, and perform power off retract】
Description	Pr1041~Pr1060 Syntec M3 axial power-off Tool auto retract distance (BLU) is set, when driver's power is cut off and perform retract action.		
Possible Cause	1. power blackout. 2. Main power supply is shut down. 3. RST power supply is cut off.		
Solution	Confirm RST power supply of driver.		

Alarm ID	OP-072	Alarm title	【System has performed power-off retract】
Description	System has performed power-off retract.		

Alarm ID	**OP-072**	**Alarm title**	**【System has performed power-off retract】**
Possible Cause	Detect Syntec M3 driver power-off abnormally. 1. Parameter Pr1041~Pr1060 has setting retract value. 2. Syntec M3 driver supports power-off retract function.		
Solution	After confirm power supply, reboot controller and driver.		

Alarm ID	**OP-073**	**Alarm title**	**【Pr3816 is changed. Tool compensation value will be cleared after reboot.】**
Description	Controller has detected that Pr3816 is changed. Prompt user after reboot controller and driver, tool compensation value will be cleared.		
Possible Cause	1. User has modified Pr3816. 2. Modified Pr3816 when import/restore parameter.		
Solution	1. Set Pr3816 back to initial setting value before reboot controller and driver, and tool compensation won't be cleared. 2. Reset to remove alarm.		

Alarm ID	**OP-074**	**Alarm title**	**【Tool compensation value is cleared. Please reset it.】**
Description	Reboot controller and driver after modified Pr3816, tool compensation value is cleared. Please reset it.		
Possible Cause	1. User has modified Pr3816. 2. Modified Pr3816 when import/restore parameter.		
Solution	1. Reset tool compensation value. 2. Reboot controller and driver to remove alarm.		

Alarm ID	**OP-075**	**Alarm title**	**【Wireless handheld box packet is lost】**
Description	Wireless handheld box packet has lost over 100 times in 10 minutes.		
Possible Cause	Wireless receiving device doesn't receive correct handheld box packet.		

Alarm ID	OP-075	Alarm title	【Wireless handheld box packet is lost】
Solution	1. Confirm wireless receiver antenna has been installed and faces up. 2. Handheld box and receiver distance must be shorter than 10 meter and no obstacle in between. 3. Increase wireless receiver noise resistant ability. i.e. add magnetic ring. 4. Separate wireless receiver's USB cable and power cable.		

Alarm ID	OP-076	Alarm title	【Wireless handheld box packet CRC is abnormal】
Description	Wireless handheld box packet CRC has abnormal check over 100 times in 10 minutes.		
Possible Cause	Wireless receiver has noise interference.		
Solution	1. Confirm wireless receiver antenna has been installed and it faces up. 2. Handheld box and receiver distance must be shorter than 10 meter and no obstacle is in between. 3. Increase wireless receiver noise resistance ability. i.e. add magnetic ring. 4. Separate wireless receiver's USB cable and power cable.		

Alarm ID	OP-077	Alarm title	【Driver parameter read and write permission has changed.】
Description	Driver parameter read and write permission has changed. Please reboot controller and driver to read driver parameter.		
Possible Cause	1. Connected driver parameter read and write permission isn’t fixed. It depends on driver parameter. 2. System detects driver parameter read and write permission isn’t complete open, and will automatically activate read and write permission.		
Solution	Wait for a while after power on, reboot controller and driver to remove alarm. Note: Some brands’ driver need to wait for driver to save parameter. And each brand's saving time needed is different. If user cut off power too early, the alarm may still exist. It is recommended to wait for one minute.		

Alarm ID	OP-078	Alarm title	【Unexpected Error】
Description	Controller detects unexpected error.		
Possible Cause	An unexpected condition appears.		
Solution	Please contact Syntec OEM.		

Alarm ID	OP-079	Alarm title	【Assigned connect device doesn't exist.】
Description	Can't find assigned connect device.		
Possible Cause	1. Controller network card doesn't exist. 2. EnIP Macro program assigned 'Network card number' or 'Slave ID' is incorrect.		
Solution	1. Please contact Syntec OEM. 2. Check EnIP Macro program assigned 'Network card number' or 'Slave ID' whether is correct.		

Alarm ID	OP-080	Alarm title	【Driver initialize. Take too much time to pre-read.】
Description	Controller can't communicate with driver normally, or driver parameter takes too much time to pre-read.		
Possible Cause	1. Controller can't communicate with driver normally. 2. A critical alarm on the drive caused the initialization to fail. 3. Take too much time to read driver parameter.		
Solution	1. Reboot controller and driver. 2. Solve all drive critical alarms. 3. Decrease Pr3202 interpolation time, then reboot controller and driver. 4. Please contact Syntec OEM.		

Alarm ID	OP-081	Alarm title	【ATC has an important alarm】
Description	ATC (auto tool change) has an critical alarm, CNC and ROT will stop to avoid machine crashing.		
Possible Cause	ATC process cannot be performed successfully, ATC has an critical alarm.		
Solution	After troubleshoot ATC alarm, Reset to remove alarm.		

Alarm ID	OP-082	Alarm Title	【ESI file read failed】
Description	Abnormal to read the ESI file imported by the user.		
Possible Cause	1. User imports more than ESI files		

Solution	1. Delete ESI file user imported with SA. 2. After deleting ESI file, must re-import ESI file to communicate with corresponding EtherCAT driver.

Alarm ID	**OP-083**	**Alarm Title**	**【Minimum effective interpolation time is 2ms on dual core systems】**
Description	The minimum effective interpolation time is 2ms on dual core systems. If the value of Pr3203 is set below 2,000, the controller will run as if it were 2,000.		
Possible Cause	The value of Pr3203 is less than 2,000.		
Solution	Modify the value of Pr3203 so that it is NO LESS THAN 2,000 and reboot the controller and the driver.		

Alarm ID	**OP-084**	**Alarm Title**	**【PLC.lad abnormal】**
Description	When the system loads the PLC ladder diagram, the file is found abnormal.		
Possible Cause	1. PLC.lad not exist. 2. PLC ladder diagram's execution line number more than 4000 lines (not include notation line numbers).		
Solution	1. Re-import PLC.lad and reboot controller. 2. PLC ladder diagram's execution line number should not exceed 4000 lines (not include notation line numbers).		

Alarm ID	**OP-085**	**Alarm Title**	**【PLC ladder diagram syntax error】**
Description	When the system loads the PLC ladder diagram, the file is found syntax error.		
Possible Cause	PLC ladder diagram syntax error.		
Solution	1. Re-import correct PLC.lad and reboot controller. 2. Correct PLC ladder diagram in edit mode, and reboot controller.		

Alarm ID	**OP-089**	**Alarm Title**	**Register of subscribed channel located in system reserved area**

Explanation	The initial register of subscribed channel is set in the system reserved area.
Possible Cause	1. The initial target register R(n) of subscribed channel is set to CNC system interface area.
Solution	1. Assign the correct register

Alarm ID	**OP-090**	**Alarm Title**	**Information setting error of subscribing channel**
Description	Information setting error of subscribing channel when applying specific functions.		
Possible Cause	1. When applying the subscribing channel as dual feedback inspection function, the corresponding channel data is not position feedback (includes position feedback, controller monitoring position feedback, controller monitoring 2nd position feedback)		
Solution	1. Please check if the subscribing channel data set by the axis card port number corresponding to Pr241~Pr260 axis dual feedback is position feedback.		

Alarm ID	**OP-091**	**Alarm title**	**【System has serious error before the previous power off, re-calibrate before machining】**
Description	The controller detected a serious error in the system before the previous power off.		
Possible Cause	The system has an unexpected error before the previous power off.		
Solution	1. Check whether machining setting data is correct. 2. Reboot controller and driver. 3. Please contact Syntec OEM, to confirm the cause of the serious error before the previous power off.		

Alarm ID	**OP-094**	**Alarm title**	**【Setpoint Axis major alarm occurred】**
Description	Setpoint Axis has major alarm, CNC will stop to avoid machine crashing.		
Possible Cause	Setpoint Axis has alarm or drive alarm.		

Alarm ID	OP-094	Alarm title	【Setpoint Axis major alarm occurred】
Solution	After clearing Setpoint Axis alarm and drive alarm, this alarm will be cleared.		

Alarm ID	OP-095	Alarm title	【System has serious exception, please reboot the system】
Description	The controller detected a serious error in the system, please reboot system immediately.		
Possible Cause	The system has an unexpected error, that will cause system instability.		
Solution	Reboot controller and driver.		

Alarm ID	OP-096	Alarm title	【Loader axis group use non Syntec M3 drive】
Description	The axis of loader axis group (Pr733, Pr734) can use only Syntec M3 drive or virtual axis.		
Possible Cause	The axis of loader axis group is not Syntec M3 drive and virtual axis.		
Solution	The axis of loader axis group change to use Syntec M3 drive or modify parameter Pr21~ to be virtual axis.		

Note: OP-096 support version: 10.118.48Y, 10.118.52S, 10.118.56M, 10.118.60G, 10.118.66A and earlier.

Alarm ID	OP-097	Alarm title	【Independent axis group (Pr741=1) check error】
Description	When Pr741 is 1, the system will make axis group, which specified by Pr733~ and Pr737, into independent axis group. Furthermore, the system will check the axis, which is belonged to independent axis group, to inhibit "one axis for multiple axis groups", "axis exchange" and "axis coupling" setting. The independent axis group is also inhibited to use setpoint axis.		
Possible Cause	1. Pr741 is set accidentally as 1. 2. When Pr741 is 1, the axis of independent axis group is also belonged to another axis groups. 3. When Pr741 is 1, the "axis exchange" and "axis coupling" parameter is set to axis of independent axis group. 4. When Pr741 is 1, there is at least one setpoint axis that is belonged to independent axis group.		

Solution	1. Pr741 is set to 0. 2. Check Pr701~ setting and confirm the axis of independent axis group is only used by an unique independent axis group. 3. Check Pr3721~, Pr3821~ setting and make sure the axis ID can not specify axis of independent axis group. 4. Check the setting of 【*Belonging to axis group ID】 of setpoint axis parameter, which can not use independent axis group ID.

Alarm ID	**OP-098**	**Alarm title**	**【The version of the serial parameter file is incompatible】**
Description	The serial parameter definition file placed during the driver update is incompatible.		
Possible Cause	The driver firmware version does not match the controller version.		
Solution	1. Update the driver firmware version to the controller compatible version. 2. Update the controller software version to the driver compatible version.		

Alarm ID	**OP-099**	**Alarm Title**	**【Illegal Setting for Independent Axis Groups (Pr732=2)】**
Description	If Pr732 is set to 2, the following restrictions apply： 1. Except that Pr742 is set to 1, one axis can only be associated to ONE axis group. 2. Inter-Axis Group use of the following functions are NOT allowed: a. Axis Exchange. b. Axis Coupling. c. ECAM.		
Possible Cause	1. Pr732 is unexpectedly set to 2. 2. When Pr732 is set to 2, Pr742 is not set to 1, and at least one axis is assigned to multiple axis groups (Pr701~). 3. When Pr732 is set to 2, there exists inter-axis group use of the following functions: a. Axis Exchange. b. Axis Coupling. c. ECAM.		

Alarm ID	OP-099	Alarm Title	【Illegal Setting for Independent Axis Groups (Pr732=2)】
Solution	1. Modify Pr732 so that it is not equal to 2. 2. Modify Pr701~ so that each axis belongs uniquely to ONE axis group. If the axis needs to be used among multiple axis groups, change Pr742 to 1 and use it as a roaming axis. 3. Modify Pr3721~, Pr3821~ so that there is no inter-axis setting. 4. Check and modify the setting of ECAM so that there is no inter-axis setting.		

Alarm ID	OP-099	Alarm Title	【Illegal Trigger of Halted Point Return】
Description	When the axis group independent function is turned on (Pr732 is set to 2), if multiple axis groups execute custom pause point return at the same time, Then only the axis group that first triggered the return of the custom pause point can be executed correctly. The rest of the axis groups will not be executed (will not be cyclically started).		
Possible Cause	When the axis group independent function (Pr732 is set to 2) and the custom pause point return function (Pr3852 is set to 1) are both enabled, Simultaneously trigger multiple axis groups to execute custom pause point return.		
Solution	After the execution of the axis group that first triggered the return of the custom pause point is completed, the cycle start of other axis groups is triggered again.		

Alarm ID	OP-101	Alarm Title	【EtherCAT-IO communication error】
Description	EtherCAT-IO communication is abnormal continuously for over 10 ms. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Cause	1. EtherCAT communication cable is loose. 2. Noise interference.		
Solution	1. Check if the communication cable is normal. 2. Replace communication cables. 3. Reboot controller and IO device.		

Alarm ID	OP-102	Alarm Title	【EtherCAT devices exceeds the number of supported stations】
Description	EtherCAT devices connected in series exceeds the number of stations supported by controller.		
Possible Cause	More than 16 EtherCAT devices are connected in series.		
Solution	Please reduce the number of EtherCAT device stations connected in series.		

Alarm ID	OP-103	Alarm Title	【Drivers connected don't support SPA3.0】
Description	The current configuration of controller and drivers don't support SPA3.0.		
Possible Cause	Controller doesn't connent any Syntec driver or software version of Syntec driver is lower than 4.4.0, 5.2.2.		
Solution	1. If there is no Syntec driver connected: change Pr3850 to be 0, use SPA2.0 instead. 2. If there is a connected Syntec driver: update software of Syntec driver to 4.4.0, 5.2.2 or higher.		

Alarm ID	OP-104	Alarm Title	【Inclined Axes control configure failed】
Description	Configuring inclined axes control fails.		
Possible Cause	1. Inclined axis or orthe axis's axis type is rotary axis. 2. Inclined axis or orthe axis is assigned to different coordinate or multiple coordinates. 3. Inclined axis or orthe axis is actived axis exchange function. 4. Inclined axis or orthe axis is actived roaming axis function. 5. Inclined axis or orthe axis is actived axis coupling function.		
Solution	1. Please modify Pr221~, and assign inclined axis or orthe axis's axis type is linear axis. 2. Please modify Pr701~， and assign inclined axis and orthe axis are same single coordinate. 3. Please modify Pr3721~ so that inclined axis or orthe axis isn't actived axis exchange. 4. Same as no.2 suggestion， when inclined axis and orthe axis are assigned same single coordinate, it don't deal with as roaming axis. 5. Please modify Pr3821~ so that inclined axis or orthe axis isn't assigned axis		

2.1 Controller Model Axial Port Number Setting List

Note: Below chart version is based on 10.116.38N, 10.116.54L, 10.118.0G, 10.118.7 and above, olde version may be slightly different.

Model	Pulse	M2	M3		ECAT	RTEX	Virtual Axis
			Single Axis	Multi-Axis			
7MA/7TA 60PA/60PC/60PE 81R-SCARA/81R-Delta/81R-Gantry 6TD-E 70SB FC-B/FC-B-5 60DB(FC Type) 60WA(FC Type) 71SA 60CB 60WB 600WA 72BA/720BA	-	-	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	1~9 (Note 3*)	-	17~18
HC-7C HC-8C/HC-10C/HC-15C HC-8C-5/HC-10C-5/HC-15C-5 HC-8E/HC-10E/HC-15E HC-80MI 22MA/22TA/22TB 220MA/220MB/220TB 220MA-5/220MB-5/220TB-5	-	-	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	1~9 (Note 3*)	1~16 (Note 7*)	17~18

Model	Pulse	M2	M3		ECAT	RTEX	Virtual Axis
			Single Axis	Multi-Axis			
6TD-H 70CB/70CB-5	19 (Note 4*)	1~16	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	1~9 (Note 3*)	-	17~18
6MD-H 21MA-H/21TA-H/ 21TB-H 210MA-H/210MB-H/210TB-H 210MA-H5/210MB-H5/210TB-H5/610MA-H5 610WA-H/610WB-H	19 (Note 4*)	1~16	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	1~9 (Note 3*)	1~16 (Note 7*)	17~18
6MA-E/6MB-E/6TA-E 21MA-E/21TA-E/ 21TB-E 6GA-E/ 21GA-E 210MA-E/ 210MB-E/ 210TB-E 210MA-E5/ 210MB-E5/ 210TB-E5/ 610MA-E5	19 (Note 4*)	-	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	1~9 (Note 3*)	1~16 (Note 7*)	17~18
HC-80GM 70CA 80S 60DB(SCD Type) 60WA(SCD Type)	-	-	1~16, 21~22 (Note 2*)	1000~1010 2000~2010 .... 16000~16010	-	-	17~18

Model	Pulse	M2	M3		ECAT	RTEX	Virtual Axis
			Single Axis	Multi-Axis			
6D 21MA/21TA/21TB HC-8B/HC-10B/ HC-12B/HC-15B 60PB 21MA-X/21TA-X/ 21TB-X 200MA/200MB/ 200TA/200TB 200MA-X/200MB-X/ 200TA-X/200TB-X 200MA-X5/200MB-X5	19 (Note 4*)	1~16	-	-	-	-	17~18
20MA-R / 20TA-R	19 (Note 4*)	-	-	-	-	1~16	17~18
FC-A 60CA	1~4	-	-	-	-	-	17~18
HC-8S 3TA/3TS 6MA/6TA/6CA/ 6MB/6TB/6CB/6B-GG 11MA//11TA HC-8A/HC-10A/ HC-12A/HC-15A	1~7	-	-	-	-	-	17~18
11MB/11TB	1~11	-	-	-	-	-	17~18

Model	Pulse	M2	M3		ECAT	RTEX	Virtual Axis
			Single Axis	Multi-Axis			
10MA/10TA/10MC/10TC/10MD/10TD/10ME/10TE/10MF/10TF 11MA-X/11TA-X EZ	1~5	-	-	-	-	-	17~18
10MB/10TB 11MB-X/11TB-X	1~10 (Note 5*)	-	-	-	-	-	17~18
Super/Super-X/Super-X5	5 * Axis Board Number (Note 6*)	-	-	-	-	-	17~18

Note 2: Only version 10.118.3 and above support 21~22 setting.

Note 3: List the current **maximum number of axis** that can be connected by **one single brand**. For the number of axis that can be connected by each brand, refer to EtherCAT Communication Development Progress..

Note 4: this pulse port is SP port.

Note 5: please check 10MB/10TB Pr13 needs to set as 2.

Note 6: Refer to Pr13 Number of servo card for axis board number detail setting.

Note 7: List the current **maximum number of axis** that can be connected by **one single brand**. For the number of axis that can be connected by each brand, refer to RTEX Driver Application Manual..

2.2 OP-001 Axis board parameter setting error or abnormal hardware

Alarm ID	OP-001	Alarm Title	Axis board parameter setting error or abnormal hardware
Description	Controller does not detect the hardware interrupt signal (old axis board: one interpolation interval sends one signal. new axis board: 0.5ms sends one signal.)		

Possible Cause	1. Controller parameters setting error. 2. CPU BIOS setting error. 3. Axis board error or DIP switch setting error. 4. Axis board and ISA SLOT have poor contact or axis board oppressed ISA SLOT. 5. Memory too low (less than 1MB). 6. Controller failed to establish communication with the drive (other than Yaskawa, Syntec). (Other driver brand's manufacturer code must be able to read via the communication command, the model format must be start with "S3P_")
Solution	1. Check whether controller Pr1 *Motion board base address setting is set according to hardware specifications (refer to application manual parameter explanation). 2. Check whether BIOS' IRQ11 setting is Legacy ISA. 3. Check DIP switch of axis board. 4. Check if the 1st axis board's disconnection signal ISR has short circuit (other axis boards must open). 5. Replace the Axis board to avoid the poor contact due to dirt and foreign foreign debris between axis board and ISA slot. 6. If fail to set up communication, please ask driver vendor to contact with Syntec OEM if needed, and will cooperate to fix driver software.

2.3 OP-002 Parameter storage access failure, system halt

Alarm ID	**OP-002**	**Alarm Title**	**Parameter storage access failure, system halt**
Description	System fails to set up PARAM.DAT and PARAM.LKN. System parameter file name is PARAM.DAT. And will save in \DiskC\OpenCNC\Data. System parameter backup file name is PARAM.LKN, and will save in \DiskC\OpenCNC\LKN. When CNC can't find or both normally use and backup system parameter file are damaged while booting, system will reset these two files according to parameter default. Note: If system is x86, \DiskC is in first piece of CF card; if system is ARM, \DiskC is in first piece of Flash.		
Possible Cause	1. x86 System: 1st peice of CF card is out of action. 2. ARM System: 1st piece of Flash is out of action.		

Solution	x86 System: 1. Unplug controller first CF card, and plug CF card's card reader into personal computer. Execute driver repairmen to CF card on personal computer and recopy backup parameter PARAM.DAT into controller dictionary in \DiskC\OpenCNC\Data. 2. Change first CF card. (Please notice first piece of CF card need to install boot system and CNC core software.) ARM System: 1. Please contact Syntec OEM.

2.4 OP-003 Parameter storage access failure, system halt

Alarm ID	**OP-003**	**Alarm Title**	**Parameter storage access failure, system halt**
Description	System fail to set up REGISTRY.DAT、 REGISTRY.LKN and REGISTRY.MIR. Process data file name when use normally is REGISTRY.DAT, exist in dictionary on \DiskA\OpenCNC\USER. First backup process data file name when is REGISTRY.LKN, exist in dictionary on \DiskA\OpenCNC\LKN. Second backup process data file name when is REGISTRY.MIR, exist in dictionary on \DiskA\OpenCNC\MIR. When it can't find or both normally use and backup system parameter file are damaged at boot time, system will reset these three files according to parameter default. Note: If system is x86, \DiskA is in first piece of CF card. Is system is ARM, \DiskA is in first piece of Flash.		
Possible Cause	1. x86 system: second CF card malfunction. 2. ARM system: second Flash card mulfunction.		
Solution	1. x86 system: If alarm kept happening, please backup user data into new CF card and change the second piece of CF card. 2. ARM system: If alarm kept happening, please backup user data and format DiskA then restore user data.		

2.5 OP-004 Machining data loss, re-calibrate before machining

Alarm ID	**OP-004**	**Alarm Title**	**Machining data loss, re-calibrate before machining**

Description	Once booted, controller will re-load the system registry file backed up before last power off into memory. After loading, if controller discovers the last shutdown state is busy or if the register file and the first backup file are damaged, one alarm will be sent to the user. The name of registry file is REGISTRY.DAT, saved with file path D:\CNC\USER. The first backup registry file is REGISTRY. MIR, saved with file path D:\CNC\ MIR. The second backup file of registry is REGISTRY. LKN, saved with file path D:\CNC\ LKN Note: If system is x86, DiskA is in first piece of CF card. If system is ARM, DiskA is in first piece of Flash.
Possible Cause	1. Shut down or power off controller in busy state. 2. File REGISTRY.DAT is damaged. 3. x86 system : second piece of CF card is broken. ARM system : second piece of Flash is broken.
Solution	1. If all machining data is confirmed to be normal, cut off power and reboot controller and driver to clear alarm. 2. x86 system: If alarm kept happening, please backup user data into new CF card and change the second CF card. 3. ARM system: If alarm kept happening, please backup user data and format DiskA then restore user data.

2.6 OP-005 I/O transmission error

Alarm ID	**OP-005**	**Alarm Title**	**I/O transmission error**
Description	When PIO5 of Watch Dog fails, the system will display warning		
Possible Cause	1. Pr5 setting doesn't match hardware structure, for example: a. 22A doesn't dispose HK adapter plate. b. 10B/10F doesn't configure two RIO when Pr5=8. 2. IO relative module (PIO5/RIO) malfunction. 3. Not supplied external 24V Voltage to RIO or RIO cable malfunction. 4. Motherboard overheats (Check System Data No. 39) 5. Controller ground cable is influenced by noise. 6. Motherboard malfunction.		

Solution	1. Set correct Pr5 parameter value according to hardware configuration. 2. Change IO relative model (PIO5/RIO). 3. Supply power to RIO correctly and check RIO cable. 4. Check if electic cabinet fan works normally. 5. Check machine has grounding correctly. 6. Change Motherboard 7. If HK adapter board is not used, please refer to custom I/O comparison table manual. Set Pr5 to 100 and set IO point (HK, MPG, PANEL-EX) on adapter board to disable (-1). 8. According to 7. If RIO is not used, please sets RIO point to disable. (-1)

2.7 OP-006 Permanent storage data CRC check failure, re-calibrate before machining

Alarm ID	**OP-006**	**Alarm Title**	**Permanent storage data CRC check failure, re-calibrate before machining**
Description	IO card hardware error.		
Possible Cause	1. IO card malfunction. 2. Fram CRC error. 3. Fram has not initialized. 4. For 10F, 200 Series controller restore backup data when CRC error. 5. Wrong setting of DIP switch when using two pieces of PIO5.		
solution	1. Change IO card. 2. Please upgrade to 10.112.95, 10.114.29 or later version. 3. Reboot controller and driver. 4. Correctly set PIO5 dip switch.		

2.8 OP-007 Low memory, memory space is less than 1M

Alarm ID	**OP-007**	**Alarm title**	**Low memory, memory space is less than 1M**
Description	System memory less than 1MB.		
Possible Cause	Developer designed controller HMI with too much components, graphs or pictures, and leading to memory less than 1MB.		
Solution	Designing controller HMI need to reduce excessive components. Use effective variable and component can avoid consuming too much memory.		

2.9 OP-008 Fatal low memory, memory space is less than 100K

Alarm ID	OP-008	Alarm title	Fatal low memory, memory space is less than 100K
Description	System memory is less than 100k.		
Possible Cause	Developer designed controller HMI with too much components, graphs or pictures, and leads to memory less than 100 KB.		
Solution	Reduce excessive components when designing controller HMI. Use effective variable and component can avoid consuming too much memory.		

2.10 OP-009 The number of interpolation loss times is bigger than 100

Alarm ID	OP-009	Alarm title	The number of interpolation loss times is bigger than 100
Description	System Data No.54 (Interpolation delay times) is more than 100. Interpolation delay: Axis card doesn't send the command calculated by last interpolation time, the system will ignore the interpolation value to avoid unsent command value be covered and cause order missing.		
Possible Cause	1. Axis card malfunction. 2. System resource runs out. 3. The setting of interpolation time interval (Pr3203) is too small.		
Solution	1. Replace the hardware of axis card. 2. Replace motherboard. 3. Increase Pr3202 interpolation time interval set value.		

2.11 OP-010 Axis Group setting error

Alarm ID	OP-010	Alarm title	Axis Group setting error
Description	When no axis belongs to CNC main system path.		

Possible Cause	Parameter setting error Ex: Assume that the machine exists XYZ axis, each associated axis group is set as follows: Pr701=1, Pr702=4, Pr703=4, and Pr731=3. CNC main system path has first path, second path and third path. But without axis belongs to second path, controller will issue this alarm.
Solution	Please check Pr701~Pr720 axis affiliated path and Pr731 CNC main system path number setting value.

2.12 OP-011 Pause point offset error

Alarm ID	**OP-011**	**Alarm title**	**【Pause point offset error】**
Description	When use multi-path in machining, with axis belongs to Plural path in different workpiece coordinate system. If user operators inappropriately, when modifying workpiece coordinate during block stop, axis offset will be different in each path. This condition will cause controller system can't send out correct moving command.		
Possible Cause	1. When changing axis workpiece coordinate system, user doesn't revise that axis all affiliated path. 2. When changing all axis affiliated paths' workpiece coordinate, each affiliated changing value is different.		
Solution	1. Please make sure affiliated path, that axis's workpiece coordinate system all changed. (Refer to Pr701~Pr702 axis affiliated path) 2. Please make sure that axis workpiece coordinate system revision value will all be the same in different affiliated path.		

2.13 OP-012 CF card set by Pr3219 error， check CF card or system setting value

Alarm ID	**OP-012**	**Alarm title**	**CF card set by Pr3219 error，check CF card or system setting value**
Description	When CNC uses Dos, if we set Pr3219 equal to 3 and second CF card (user data) is not inserted, system mis-used REGISTRY.DAT data in order to avoid CF card malfunction.		
Possible Cause	1. The system doesn't have the 2nd CF card or cannot read the 2nd CF card. 2. The system has two CF cards, but both CF card has only one partition disk.		

Solution	1. Check whether the 2nd CF card exists or there is CF card reading error. 2. Set the first card to the default state, which is having wo partition disks condition. 3.Set parameter 3219 to 0 and all use C:

2.14 OP-013 Software version and model can't driver hardware, call your vender!

Alarm ID	**OP-013**	**Alarm title**	**Software version and model can't driver hardware, call your vender!**
Description	Controller installed software version that hardware doesn't support. Before eliminating alarm, if user move the machine, controller will issue following error alarm.		
Possible Cause	1. Controller install software version that hardware doesn't support. a. EZ milling machine doesn't support version after 10.114. b. 10A/B doesn't support version after 10.116.10. c. Software doesn't support the FPGA device. d. Pr9 *Servo board type doesn't support the FPGA device. 2. Controller has changed both motherboard or CF card and IO card at the same time, so it causes hardware information incompatible. Machine code dropped to EZ so it can't support original software version.		
Solution	1. Install software version that hardware supports. a. EZ milling machine standard configures 10.112 version. If user want to use higher version, user need to open software option function Option1 [EZ2/3/4 upgrade 10A software function]. b. 10A/B because of memory issue, if user wants to use higher version, recommend to change into 10F. c. Please contact and provide backup file to Vendor-Syntec. d. Please contact and provide backup file to Vendor-Syntec. 2. Please contact Vendor-Syntec to solve hardware information incompatible problem and reset controller serial code.		

2.15 OP-014 WinCE option not enabled, call your vender!

Alarm ID	**OP-014**	**Alarm title**	**WinCE option not enabled, call your vender!**
Description	Controller installs WinCE version software, and WinCE function option is not activated, so axis card can't send command which cause the controller issue following error alarm after machining start.		

Possible Cause	Option software function Option2. "Controller upgrades WinCE system" is not activated.
Solution	1. Re-install Dos version. 2. Please contact Syntec OEM, to open option software function Option2 "Controller upgrades WinCE system" 3. Super controller doesn't have this error.

2.16 OP-015 The output contacts are overloaded; please check for any abnormal wiring

Alarm ID	**OP-015**	**Alarm title**	**【The output contacts are overloaded; please check for any abnormal wiring】**
Description	Controller detects PLC O point overloading. Sixteen O point as a group. When one O point in that group overloads 350 mA will send out alarm of O point in that range is overloading.		
Possible Cause	1. Loads more than 350mA. 2. Output connection point is short circuit. 3. Machine has electricity leakage 4. machine loading device is aged.		
Solution	1. Verify load power consumption. 2. Verify that the output is not short-circuited. 3. Machine grounding. 4. Replace load equipment for testing.		

2.17 OP-016 Straightness compensation function configuration error

Alarm ID	**OP-016**	**Alarm Title**	**Straightness compensation function configuration error**
Description	Straightness accuracy offset function current supplies at most five groups of moving axis and offset axis setting. In each group, the compensation position of the "at least two and at most twenty" moving axis and the corresponding offset axis' compensation amount must be entered. Therefore, when the system detects that the straightness compensation function parameter setting error, the controller issues this alarm.		

Alarm ID	OP-016	Alarm Title	Straightness compensation function configuration error
Possible Cause	1. Multi groups of moving axis are corresponding to same offset axis simultaneously. 2. Moving axis is also offset axis. 3. Activate straightness accuracy offset's moving axis and offset position descends. 4. Activate straightness accuracy offset's moving axis and offset position is less than two positions.		
Solution	1. Check Pr1481~Pr1490 Straightness offset moving axis number and compensating axis number a. map multiple sets of moving axes to different offset axis. b. configure moving axis not as an offset axis. 2. Check Pr7001~Pr7025 Straightness offset moving axis position and Pr7051~Pr7125* Position of each group's compensation point in moving axis pitch compensation form a. fill in increasing compensation position order. b. fill in at least two and at most twenty compensation positions. 3. Expand 2D linear compensation function (20%), supported versions: 10.118.12E, 10.118.15.		

2.18 OP-017 Port number or I point mapped to handwheel error

Alarm ID	OP-017	Alarm Title	Port number or I point mapped to handwheel error
Description	Pr2021~Pr2027 setting error.		
Possible Cause	1. Pr2021~Pr2030 *Port number or register number for MPG port number setting is not supported. 2. Pr2024~Pr2027 *Continued I-Bit no. for MPG two continues I points setting is not supported. 3. Pr13 *Number of servo card setting error.		
Solution	1. Refer to the description in the manual for correct configurations according to the controller code. 2. Check whether Pr13 *Number of servo card setting is correct or not.		

2.19 OP-018 Port configuration error

Alarm ID	OP-018	Alarm Title	Port configuration error
Description	Pr21~Pr40 * Axis corresponding axis card port number or Pr1621~Pr1630 Axis ID or Port Number for Spindle setting error. Serial PLC Axis, ROT port number setting error. Pr3261~Pr3263 *M3-IO Station No setting error.		

Alarm ID	OP-018	Alarm Title	Port configuration error
Possible Cause	1. Pr21~Pr40 axis port number setting is not supported. 2. Pr21~Pr40 axis port number setting is repeated, only number 17, 18 (virtual axis) can be repeated. 3. When using variable-frequency spindle (Pr1791~Pr1800 sets as 0), Pr1621~Pr1630 spindle port number setting is not supported. 4. When using SRI DA spindle (Pr1791~Pr1800 sets as 7), Pr1621~Pr1630 register number setting is not at User-defined 5. Serial Bus controller uses variable-frequency spindle (Pr1791~Pr1800 sets as 0). Variable-frequency spindle port number (Pr1621~Pr1630) can't have conflict with other serial bus controller port number. 6. Pr3261~Pr3263 port number setting has conflict with axis (Pr21~Pr40, Pr1621~Pr1630). 7. Serial PLC axis and ROT port number is not supported. 8. Pulse controller which is 10.116.54L, 10.118.0G, 10.118.7 and above version, if port number hardware setting is not supported, this alarm will be triggered. 9. Pr13 *Number of servo card setting error Note: Port number setting can refer to Controller Model Axial Port Number Setting List.		
Solution	1. Refer to the Description in the parameter manual to configure the correct port number. 2. Configure the port number in Pr1621~Pr1630 which is not conflict with other serial servo ports, set to 19 is recommended. 3. When using SRI DA spindle (Pr1791~Pr1800 sets as 7), configure the port number in Pr1621~Pr1630 in User-defined region. 4. Check whether Pr3261~Pr3263 and Pr21~Pr40, Pr1621~Pr1630 conflict with each other, and correct the conflict. 5. For pulse controller version 10.116.54L, 10.118.0G, 10.118.7 and above, if user wants to use virtual axis, please set as 17/18, or use G10 L800 to specify the virtual axis. 6. Check if Pr13 *Number of servo card is set correctly.		

2.20 OP-019 Prohibit to switch the metric and imperial systems in machining/not ready state

Alarm ID	OP-019	Alarm Title	Prohibit to switch the metric and imperial systems in machining/not ready state
Description	When system is Busy or Not Ready, can't switch into Metric and Imperial units via HMI operation.		
Possible Cause	When system is Busy or Not Ready, switch into Metric and Imperial units via HMI operation.		

Solution	Avoid to switch the metric and imperial systems in Busy or Not Ready state via HMI operations.

2.21 OP-020 M code setting conflict

Alarm ID	**OP-020**	**Alarm title**	**M code setting conflict**
Description	1. When the following M code related parameter setting has repeated M code and cause setting conflict. a. Pr3599 *Register M code for Advanced Look-Ahead mode b. Pr3600 *Register custom Macro interrupt M code c. Pr3601~Pr3610 *M code Macro call registry d. Pr3804 Part count M code e. Pr3741, Pr3743 Rotary axis auxiliary brake M code 2. When Pr3599 *Register M code for Advanced Look-Ahead mode set as 1, 2, 30, 98, 99, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 3. When Pr3600 *Register custom Macro interrupt M code sets as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 4. When Pr3741, Pr3743 Rotary axis auxiliary brake M code set as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198. 5. When Pr3763 Braking M code of Indexing Axis set as 1, 2, 29, 30, 97, 98, 99, 197, 198, it is conflicted with M1/M2/M30/M98/M99/M198.		
Possible Cause	As stated above.		
Solution	After modifying the above parameter settings so that no conflict exist, reboot the controller and driver.		

2.22 OP-021 Use last known user data, re-calibrate user data before machining

Alarm ID	**OP-021**	**Alarm title**	**Use last known user data, re-calibrate user data before machining**
Description	The controller's machine data, tool table and scan table information are stored in Registry.dat, ToolTable.dat and ScanTable.dat respectively, and each has three backups of .mir, .lkn and .cdb. If either .dat and .mir are abnormal and cannot be used, the system will use .lkn or .cdb. If either .dat and .mir are abnormal and cannot be used, the system will use .lkn or .cdb to restore the data. If the Registry is abnormal, the machining file name will be cleared.		

Possible Cause	1. X86 system (with CF card): It is possible that the CF card is damaged and the machine data, tool table or scan table file is damaged. 2. ARM system (without CF card): DISKA damage results in damage to machine data, tool table or scan table data. 3. The following is a list of possible file damage situations and their restoration sources. a. Machine data and other data are damaged, restored from DISKA backup. b. Tool table data is damaged, and it is restored from DISKA backup. c. Scan table data damaged, restored from DISKA backup. d. Machine data and other data are damaged, restored from DISKC backup. e. Tool table data is damaged, and it is restored from DISKC backup. f. Scan table data damaged, restored from DISKC backup.
Solution	1. Reset or check whether the machine data, tool table and scan table data are correct. 2. Scan CF card disk. 3. Reboot controller and driver.

2.23 OP-022 User data rebuild, re-calibrate user data before machining

Alarm ID	**OP-022**	**Alarm title**	**User data rebuild, re-calibrate user data before machining**
Description	None of Registry, ToolTable and ScanTable in the controller has .dat, backup files .mir, .lkn and .cdb. User data is rebuilt.		
Possible Cause	1. X86 system (with CF card): CF2 card may has been replaced or formatted, causing user's original data and backups being cleared out. 2. ARM system (without CF card): DISKA and DISKC are damaged or formatted, causing user's original data and backups being cleared out. 3. The possible file corruption situations are listed below. a. Machine data, other data, and their relative backups are all destroyed. b. Tool table data and its backups are all destroyed. c. Scan table data and its backups are all destroyed.		
Solution	1. Reset the machine data, tool table and scan table data, and then reboot controller and driver. 2. If this alarm still cannot be eliminated after reboot, format CF2/DISKA and CF2/DISKC. 3. If alarm still cannot be eliminated after format, CF2/DISKA and CF2/DISKC are malfunction and need repair.		

2.24 OP-023 Power break in machining, re-calibrate before machining

Alarm ID	**OP-023**	**Alarm title**	**Power break in machining, re-calibrate before machining**

Description	When new machining data need to be stored during Cycle Start, the controller will store a state flag into Fram or REGISTRY.DAT, and it will be removed when the specified data have already been stored. While controller booting, if this flag is not cleared, this alarm will be issued.
Possible Cause	The power is turned off when new machining data not yet been stored during machining process.
Solution	1. Please re-calibrate your machining data. For example, a. Check the Imperial/Metric (G70/G71) unit selection. b. Check the home point, offset, and rotational angle of the coordinate system corresponding to the G92/G92.1/G68.2/G68.3 command. c. Check the tool number and the compensation amount of the tool length compensation command (G43). d. Check the settings of the 2nd Software Stroke Limit command (G22). e. Check the values stored in the the persistent variables and R bit, i.g. Macro @656~@1999、R100~R255、R10000~R10999. f. For more details, please refer to Registry Data/L Variable List. 2. After completing step 1, following the following step by version : a. For software version 10.118.40B, 10.118.41 or above, please trigger "**reset**" to clear alarm. b. For other versions, please restart the controller to clear alarm. 3. Way to avoid: Avoid switching the power off during machining. Or, trigger C47 to store the machining data before switching the power off.

2.25 OP-024 Machine data file write fail, re-calibrate before machining

Alarm ID	**OP-024**	**Alarm title**	**Machine data file write fail, re-calibrate before machining**
Description	Before shutting down controller, the last writing of REGISTRY.DAT or REGISTRY.MIR file is unsuccessful.		
Possible Cause	1. CF card is damaged. 2. REGISTRY.DAT or REGISTRY.MIR files are set as read-only. 3. System file processing device's resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

2.26 OP-025 Machine data file write fail many times, re-calibrate before machining

Alarm ID	OP-025	Alarm title	Machine data file write fail many times, re-calibrate before machining
Description	Before shutting down controller, the accumulated unsuccessful of writing REGISTRY.DAT or REGISTRY.MIR file exceed 100 times.		
Possible Cause	1. CF card damaged. 2. REGISTRY.DAT or REGISTRY.MIR file is set to read-only. 3. System's file processing device resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

2.27 OP-026 Machine data file fault error, re-calibrate before machining and do scan disk

Alarm ID	OP-026	Alarm title	Machine data file fault error, re-calibrate before machining and do scan disk
Description	Before shutting down controller, unsuccessful of writing REGISTRY.DAT or REGISTRY.MIR file exceed 100 times continuously happened.		
Possible Cause	1. CF card damaged. 2. REGISTRY.DAT or REGISTRY.MIR file is set to read-only. 3. System's file processing device resource is not sufficient, lead to writing file unsuccessful.		
Solution	1. Scan CF card disk. 2. Check whether machining setting is correct. 3. Reboot controller and driver.		

2.28 OP-027 CNC main system axis group to be executed unselected

Alarm ID	OP-027	Alarm Title	CNC main system axis group to be executed unselected
Description	No CNC main system path is chosen and cycle start triggered.		

Alarm ID	OP-027	Alarm Title	CNC main system axis group to be executed unselected
Possible Cause	Please check R21 value: 1. Chosen path not exists or not belongs to CNC main system path. 2. No CNC main system path is chosen (Only happens when R21.11 status is ON).		
Solution	1. Check if any logic error when PLC triggers R21 option, which results in the correct path is not selected successfully. 2. Select an existing CNC main system path via R21.		

2.29 OP-028 Prohibit CNC main system axis group switch in machining

Alarm ID	OP-028	Alarm Title	Prohibit CNC main system axis group switch in machining
Description	When CNC is under Not Ready state, change CNC main system path that was going to be executed and triggers cycle start.		
Possible Cause	Please check R21 value, whether it is different from original process condition. (Change the chosen CNC main system path that was going to be executed)		
Solution	Modify R21 to the value in the original machining status, then can continue machining.		

2.30 OP-029 CNC axis over system support

Alarm ID	OP-029	Alarm Title	CNC axis over system support
Description	Assign too much axis that belongs to a normal path (Not Loader or auto labeling path).		

SYNTEC

Alarm ID	OP-029	Alarm Title	CNC axis over system support
Possible Cause	Refer to following parameter setting, when the axis number of a normal path exist the limit of system axis number, this alarm will be issued. 1. Pr701~Pr720 *Axial associate axis groups 2. Pr733~Pr734 *Loader axis group 3. Pr737 * Woodworking labeling axis group For example: Use 21MA, standard configuration is 6 axis. If 8 axis is activated (activate software option function Option20 [Loader axis number expansion]/ Option37 [auto labeling axis number expansion]) and configures following parameter settings 1. Pr701~Pr704 sets as 1, number 1~4 axis are first path. 2. Pr705~Pr708 sets as 2, number5~8 axis are second path. 3. Pr733, Pr734 and Pr737 all set as 0, no path is Loader/ auto labeling path. Therefore system has 8 axis, which over 21MA system's axis number limit.		
Solution	Check and correct the following parameter settings: 1. Pr701~Pr720 *Axial associate axis groups 2. Pr733~Pr734 *Loader axis group 3. Pr737 * Woodworking labeling axis group Based on the above example, in case of the 2nd path operation requirements can be fulfilled with the Loader path, Pr733 set as 2 to specify the 2nd path is the Loader path. Only 4 axis belongs to a normal path as system axis, when total axis number not exceed the standard 21MA axis number configuration, alarm can be removed.		

2.31 OP-030 Debug mode is activated, disable debug mode to resume normal operation.

Alarm ID	OP-030	Alarm Title	Debug mode, disable debug mode to resume normal operation.
Description	Active debug mode, Pr3221 set as 1 or 2.		
Possible Cause	As stated above.		
Solution	Disable the debug mode, Pr3221 set as 0.		

2.32 OP-031 GANTRY robot option not enabled

Alarm ID	OP-031	Alarm Title	GANTRY robot option not enabled
Description	Pr3201 activate robot function, but software option function Option24 is not activate [Cartesian Coordinate Robot Type]. (Currently only 10.117 version has this alarm.)		
Possible Cause	Pr3201 is different from correspond controller's machine mode.		
Solution	1. Enable Option24 [Cartesian Coordinate Robot Type] for GANTRY type robot. 2. In case of not robot, configure Pr3201 correctly.		

2.33 OP-032 Mechanism type configuration conflicted

Alarm ID	OP-032	Alarm Title	Mechanism type configuration conflicted
Description	Pr3201 Machine Type setting conflicted.		
Possible Cause	Currently only the following mechanism type support five axis function; 1. Pr3201 sets as 0, close lathe feature. (Use general milling interface) 2. Pr3201 sets as 1, Lathe Habit Type C. 3. Pr3201 sets as 2, Lathe Habit Type A. 4. Pr3201 sets as 3, Lathe Habit Type B. Therefore, open both five axis (Pr3001, Pr3101, Pr5501, Pr5601) and other non-lathe/mill machine tool feature's mechanism type (Pr3201) at the same time, and alarm will be issued to inform user. For example: 1. Activate first group of five axis function (Pr3001 isn't 0. First path default to use first five axis kinematic chain) and first path is not lath/mill machine tool attribute. (Pr3201 in first path is not 0~3) 2. Activate second group of five axis function (Pr3101 isn't 0. Second path default use second five axis kinematic chain) and second path is not lath/mill machine tool attribute. (Pr3201 in second path is not 0~3) Besides, when activate option software function Option 29 (Four axis dedicated Rotate Tool Center Point function (4AXRTCP)), however, and option software function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function) are not activated, but sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 1~3, and this alarm will be issued.		

Alarm ID	OP-032	Alarm Title	Mechanism type configuration conflicted
Solution	Only the milling machine supports the 5-axis models. Configure Pr3201 to 0~3, or configure the mapped 5-axis function parameters (Pr3001, Pr3101, Pr5501 and Pr5601) to 0. [Note] In the versions after the 10.116.54G and 10.118.0D, the lathe can enable RTCP and is limited to 200TB-5. 1. Only lathe/mill machine tool supports five axis machine type. Please sets Pr3201 as 0~3, or sets corresponding five axis function parameter as 0. (Pr3001, Pr3101, Pr5501, Pr5601) 2. Active RTCP for Lathe 200TB-5. Supported version: 10.116.54G, 10.118.0D or above versions. 3. Please sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 4 or 5, or please open software option function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function).		

2.34 OP-033 Do not set machine position during machining

Alarm ID	OP-033	Alarm Title	Do not set machine position during machining
Description	Using C025 ~ and C230 ~ to set machine position when machining		
Possible Cause	Using C025 ~ and C230 ~ to set machine position when machining		
Solution	Do not use C025 ~ and C230 ~ to set machine position when machining.		

2.35 OP-041 Machining file inexistent in the specific path

Alarm ID	OP-041	Alarm Title	Machining file inexistent in the specific path
Description	System can’t find ECAM.xml in designated path, so can’t activate cam function.		
Possible Cause	1. File name is wrong. 2. File location is wrong.		
Solution	Check the ECAM.xml.		

2.36 OP-042 Machining program data error, please check NC File

Alarm ID	OP-042	Alarm Title	Machining program data error, please check NC File
Description	ECAM.xml file format is wrong.		
Possible Cause	Self edit ECAM.xml, and format is wrong.		
Solution	Check the ECAM.xml.		

2.37 OP-043 ECAM group more than 8 groups

Alarm ID	OP-043	Alarm Title	ECAM group more than 8 groups
Description	Too much ECAM group.		
Possible Cause	Self edit ECAM.xml, sets up too many ECAM groups.		
Solution	Check the graphic dialog input to ensure the configured number of CAM groups is appropriate.		

2.38 OP-044 CAM curves more than 32 lines

Alarm ID	OP-044	Alarm Title	CAM curves more than 32 lines
Description	Too much cam curve line.		
Possible Cause	Self edit ECAM.xml, sets up too many cam curve line.		
Solution	Check the graphic dialog input to ensure the configured number of CAM curves is appropriate.		

2.39 OP-045 CAM curve speed planning failed

Alarm ID	OP-045	Alarm Title	CAM curve speed planning failed
Description	Fail to plan cam curve speed.		
Possible Cause	1. Cam curve design doesn't match movement parameter. 2. Movement parameter setting is not reasonable. 3. Doesn't execute re-read after fixing insertion point through R value.		
Solution	1. Check if the planned CAM curve line is reasonable under the limit of machine motion configurations. 2. Correct the machine motion parameter within the reasonable range. 3. Re-read R values and set C300 to On.		

2.40 OP-046 ECAM – no driven axis name

Alarm ID	OP-046	Alarm Title	ECAM – no driven axis name
Description	Electronic cam-Can't find axis name.		
Possible Cause	Pr321~Pr340 axis name is not set.		
Solution	Check and correct Pr321~340 setting.		

2.41 OP-047 Use Non self-defined R value or too many R values

Alarm ID	OP-047	Alarm Title	Use non self-defined R value or too many R values.
Description	R value used is out of self-defined range or insertion point sets over 100 R value.		
Possible Cause	1. Use non self-defined R value. 2. Self-edit ECAM.xml, insertion point sets over 100 R value.		
Solution	Check ECAM.xml, if the R value of workpiece counts and the insert point are out of user self-defined range, and whether insertion point used over 100 R value.		

2.42 OP-048 Curve insert point re-read R value failed

Alarm ID	OP-048	Alarm Title	Curve insert point re-read R value failed
Description	When re-read R value, electronic cam is under coupling mode.		
Possible Cause	Electronic cam group is under coupling mode.		
Solution	Release all CAM coupling and set S301~S308 to Off.		

2.43 OP-050 SRI status error

Alarm ID	OP-050	Alarm Title	SRI status error
Description	SRI condition abnormal.		
Possible Cause	Please check the followed SRI alarm, and refer to SRI Alarm -SRI.		
Solution	Refer to the solution of followed SRI alarm. After resolving the issue, reboot controller and driver.		

2.44 OP-051 M3-IO communication error

Alarm ID	OP-051	Alarm Title	M3-IO communication error
Description	M3-IO communication abnormal and cause watchdog has accumulated over 100 errors.		
Possible Cause	1. M3 communication cable is loose. 2. Noise interference. 3. Using the laser cruise function without installing the ADD-on card.		
Solution	1. Check if the communication cable is normal. 2. Replace communication cables. 3. Reboot controller and driver. 4. Check whether the laser cruise function is required. If necessary, confirm that the ADD-on card is installed correctly.		

2.45 OP-052 M3-IO device status error

Alarm ID	OP-052	Alarm Title	M3-IO device status error
Description	Controller detects M3-IO device is abnormal through M3 communication.		
Possible Cause	M3-IO card abnormal.		
Solution	1. Check if the M3-IO card is inserted securely. 2. Check the setting of Pr3261~Pr3263. 3. Reboot controller and driver.		

2.46 OP-053 Serial communication initialization failed

Alarm ID	OP-053	Alarm Title	Serial communication initialization failed
Description	When controller fail to communicate with driver.		
Possible Cause	1. Driver is abnormal. 2. Driver or controller hardware malfunction. 3. When Pr9 sets as EtherCat communication, and driver is not connected. 4. When Pr9 sets as RTEX communication, and driver is not connected. 5. M3 multi-axis driver, the first station is not set.		
Solution	1. Reboot the controller and driver. 2. Replace the driver, if it is not effective, the controller hardware may be malfunctioned. 3. Connect EtherCAT driver. 4. M2 / M3 initialization error code display on D53, please contact the Syntec OEM and provide D53 value. 5. Set the first station, or enlarge the set value of Pr3203 (interpolation time). Note: For EtherCAT communication error, refer to EtherCAT Drive Application Manual..		

2.47 OP-054 Multi-axis driver initialization failed

Alarm ID	OP-054	Alarm Title	Multi-axis driver initialization failed
Description	Multi-axis driver initialization failed.		

Alarm ID	OP-054	Alarm Title	Multi-axis driver initialization failed
Possible Cause	first servo axis function is not activated when using multi-axis driver and cause initialization failure. Note: This alarm has be removed in 10.116.36P and above version. Multi-axis driver don't have to activate first servo axis.		
Solution	Activate first servo axis in multi-axis driver, reboot the controller and driver.		

2.48 OP-055 M3 communication error

Alarm ID	OP-055	Alarm Title	Drive information reading error
Description	Failed to read all drives' information.		
Possible Cause	1. Communication between controller and driver is not stable, causing the M3 drive message or EtherCAT drive mailbox communication takes too much time. 2. M3 Drive doesn't support M3 message communication.		
Solution	1. Check the wiring of the communication cable between the controller and the drive and resolve the causes for unstable communication. 2. If it is M3 network, check if the connected drives support the M3 message communication.		

2.49 OP-056 Some axial directions cannot Servo ON. System cannot operates.

Alarm ID	OP-056	Alarm Title	Some axial directions cannot Servo ON. System cannot operates.
Description	When there is axis servo not servo on, cycle start or other manual function is triggered.		
Possible Cause	1. When power on, axis doesn't complete magnetic pole angle detection and cause servo disabled. 2. Driver power supplication is abnormal or there is driver alarm, and cause servo change from servo on condition into disabled condition. 3. Driver is malfunction.		
Solution	1. After power on, wait for a while then operate the machine. 2. Check if the driver power supply status is normal and if there is driver alarm. 3. Check if the driver is normal.		

2.50 OP-057 Tool retraction function, hand wheel shift only supports R518, R519 set to 0

Alarm ID	**OP-057**	**Alarm title**	**【Tool retraction function, hand wheel shift only supports R518, R519 set to 0】**
Description	When using G10.6 toll retraction function, R518, R519 must be set as 0, then user can use MPG to move.		
Possible Cause	R518, R519 is not 0.		
Solution	Set R518, R519 to 0.		

2.51 OP-058 Burn mode. All I/O connections to servo is disconnected

Alarm ID	**OP-058**	**Alarm Title**	**Burn mode. All I/O connections to servo is disconnected**
Description	When controller and driver is burning firmware, user can't operate driver's servo axis or IO device.		
Possible Cause	Executed controller remote update driver firmware.		
Solution	Reboot the controller and driver.		

2.52 OP-059 Motion control module loading failed. Reinstall software

Alarm ID	**OP-059**	**Alarm Title**	**Motion control module loading failed. Reinstall software**
Description	Fail to load because of controller internal file is lost or destroyed.		
Possible Cause	Controller install package is destroyed.		
Solution	Download and re-install the controller installation package.		

2.53 OP-060 M3-IO device number configuration error

Alarm ID	OP-060	Alarm Title	M3-IO device number configuration error
Description	Controller M3-IO device number setting error.		
Possible Cause	When using M3-IO device number repeated.		
Solution	Check if Pr3261~3263 value is repeated.		

2.54 OP-061 M3 external device status error

Alarm ID	OP-061	Alarm Title	M3 external device status error
Description	Controller M3 external device's communication condition is abnormal.		
Possible Cause	1. M3 communication cable is loose. 2. Noise interference. 3. M3 external device is abnormal.		
Solution	1. Check if communication cable is normal. 2. Change communication cable. 3. Check if M3 external device is firmly connected. 4. Reboot controller and driver.		

2.55 OP-062 Customized IO mapping table destroyed

Alarm ID	OP-062	Alarm Title	Customized IO mapping table destroyed
Description	When using custom IO mapping table, there is error in reading files.		

Alarm ID	OP-062	Alarm Title	Customized IO mapping table destroyed
Possible Cause	1. Custom IO mapping table is incomplete, destroyed or missing. 2. Custom IO mapping table version is incompatible with software version. 3. Software 10.118.9 or above version's custom IO supports SRI. If downgrade after setting or backup files into older version, there will be version incompatible issue. 4. Software 10.118.11 or above version's custom IO supports M3-IO expansion card with two DA input (Scan in Interpolation time), if downgrade after setting or backup files into old version, there will be version incompatible issue. 5. Software 10.118.24A, 10.118.25A, 10.118.28N, 10.118.38 or above version's custom IO supports single IO setting, if downgrade after setting or backup files into old version, there will be version incompatible issue.		
Solution	1. Enter the IO configuration page, and restore default value, then re-set IO mapping table. 2. Use Restore function, confirm the backup file is normal and restore.		

2.56 OP-063 Custom IO mapping table setting conflicts

Alarm ID	OP-063	Alarm title	【Custom IO mapping table setting conflicts】
Description	There is conflict in IO point setting when using custom IO mapping table.		
Possible Cause	Custom IO mapping table I point or O point setting repeated.		
Solution	1. Enter IO setting screen, modify the conflicting IO points. 2. Reboot controller and driver.		

2.57 OP-064 Configured as custom IO. Review the IO mapping table.

Alarm ID	OP-064	Alarm Title	Configured as custom IO. Review the IO mapping table.
Description	When Pr5 is set to be using custom IO mapping table.		
Possible Cause	Pr5 set as 100. And change from standard IO into custom IO.		
Solution	The alarm is used for reminder. Reboot controller and driver to remove alarm.		

2.58 OP-065 Laser marking status error

Alarm ID	OP-065	Alarm Title	Lasermarking status error
Description	Laser marking condition is abnormal.		
Possible Cause	Please check the followed LaserCtrl alarm. And refer to Serial PLC Axis Alarm – SERIALPLCAXIS		
Solution	Refer to the solution of LaserCtrl alarm. After resolving the issue, reboot controller and driver.		

2.59 OP-067 The wireless handheld box device is abnormal

Alarm ID	OP-067	Alarm title	【The wireless handheld box device is abnormal】
Description	Wireless handheld box external device's communication is abnormal.		
Possible Cause	1. Wireless transmitting device is unplugged. 2. Wireless transmitting device can't be read normally by USB.		
Solution	1. Check whether the wireless transmitter is loose. 2. Press the Reset button and try to reconnect. If communication is normal, this alarm can be removed.		

2.60 OP-068 The software version does not support this M3-IO device

Alarm ID	OP-068	Alarm title	【The software version does not support this M3-IO device】
Description	1. Driver's firmware version doesn't support hardware information on M3-IO for controller to identify. 2. Controller's software version cannot identify M3-IO device on multi-axis driver.		

Alarm ID	OP-068	Alarm title	【The software version does not support this M3-IO device】
Possible Cause	1. Only driver with 2.7.0 or above version support M3-IO device hardware information; Previous version can't get hardware information so doesn't support all M3-IO device, in order to protect machine operation safety. 2. Controller with 10.118.10 or above version has M3-IO device identification protection. Besides following hardware, all will issue this alarm to prevent machine from wrong action. a. 16I 8O b. 32I 32O MPG c. 32I 32O MPG 2DA 3. Pr3261~Pr3263 M3-IO station number setting error.		
Solution	1. Please update driver firmware to 2.7.0 and above version. 2. Please contact Syntec OEM, request for the software version that supports the M3-IO devices and upgrade the software. 3. Change back to the following three M3-IO devices a. 16I 8O b. 32I 32O MPG c. 32I 32O MPG 2DA 4. Refer to Pr3261~Pr3263 *M3-IO Station No and set the M3-IO station number correctly.		

2.61 OP-069 Over axis limit that interpolation time can support

Alarm ID	OP-069	Alarm title	【Over axis limit that interpolation time can support】
Description	Total axis number is over the limit that interpolation time can support.		
Possible Cause	1. Interpolation time Pr3203 is too small. 2. Axis number setting error.		
Solution	1. Interpolation time Pr3203 recommends to set 2000 or lager. Detailed support axis number please refer to Pr3203. 2. Multi-axis axis number please start setting with x000 (x means station number).		

2.62 OP-070 Product doesn't support Pr9 setting

Alarm ID	OP-070	Alarm title	【Product doesn't support Pr9 setting】
Description	This product doesn't support Pr9 setting.		

Alarm ID	OP-070	Alarm title	【Product doesn’t support Pr9 setting】
Possible Cause	1. Set Pr9 as 120/121 (RTEX), but hardware doesn’t support it. 2. Set Pr9 as 120/121 (RTEX), but software(versions 10.118.30E, 10.118.32E, 10.118.40I, 10.118.41I, 10.118.45 or above) doesn’t support it. 3. Pr9 setting error.		
Solution	1. Please contact Syntec OEM to know product information. 2. Revert the controller to the version under 10.118.30E, 10.118.32E, 10.118.40I, 10.118.41I or 10.118.45. 3. Refer to Pr9 *Servo board type, set the usable parameter value for the product.		

2.63 OP-071 Driver power off, and perform power off retract

Alarm ID	OP-071	Alarm title	【Driver power off, and perform power off retract】
Description	Pr1041~Pr1060 Syntec M3 axial power-off Tool auto retract distance (BLU) is set, when driver's power is cut off and perform retract action.		
Possible Cause	1. power blackout. 2. Main power supply is shut down. 3. RST power supply is cut off.		
Solution	Confirm RST power supply of driver.		

2.64 OP-072 System has performed power-off retract

Alarm ID	OP-072	Alarm title	【System has performed power-off retract】
Description	System has performed power-off retract.		
Possible Cause	Detect Syntec M3 driver power-off abnormally. 1. Parameter Pr1041~Pr1060 has setting retract value. 2. Syntec M3 driver supports power-off retract function.		
Solution	After confirm power supply, reboot controller and driver.		

2.65 OP-073 Pr3816 is changed. Tool compensation value will be cleared after reboot.

Alarm ID	**OP-073**	**Alarm title**	**【Pr3816 is changed. Tool compensation value will be cleared after reboot.】**
Description	Controller has detected that Pr3816 is changed. Prompt user after reboot controller and driver, tool compensation value will be cleared.		
Possible Cause	1. User has modified Pr3816. 2. Modified Pr3816 when import/restore parameter.		
Solution	1. Set Pr3816 back to initial setting value before reboot controller and driver, and tool compensation won't be cleared. 2. Reset to remove alarm.		

2.66 OP-074 Tool compensation value is cleared. Please reset it.

Alarm ID	**OP-074**	**Alarm title**	**【Tool compensation value is cleared. Please reset it.】**
Description	Reboot controller and driver after modified Pr3816, tool compensation value is cleared. Please reset it.		
Possible Cause	1. User has modified Pr3816. 2. Modified Pr3816 when import/restore parameter.		
Solution	1. Reset tool compensation value. 2. Reboot controller and driver to remove alarm.		

2.67 OP-075 Wireless handheld box packet is lost

Alarm ID	**OP-075**	**Alarm title**	**【Wireless handheld box packet is lost】**
Description	Wireless handheld box packet has lost over 100 times in 10 minutes.		
Possible Cause	Wireless receiving device doesn't receive correct handheld box packet.		

Alarm ID	OP-075	Alarm title	【Wireless handheld box packet is lost】
Solution	1. Confirm wireless receiver antenna has been installed and faces up. 2. Handheld box and receiver distance must be shorter than 10 meter and no obstacle in between. 3. Increase wireless receiver noise resistant ability. i.e. add magnetic ring. 4. Separate wireless receiver's USB cable and power cable.		

2.68 OP-076 Wireless handheld box packet CRC is abnormal

Alarm ID	OP-076	Alarm title	【Wireless handheld box packet CRC is abnormal】
Description	Wireless handheld box packet CRC has abnormal check over 100 times in 10 minutes.		
Possible Cause	Wireless receiver has noise interference.		
Solution	1. Confirm wireless receiver antenna has been installed and it faces up. 2. Handheld box and receiver distance must be shorter than 10 meter and no obstacle is in between. 3. Increase wireless receiver noise resistance ability. i.e. add magnetic ring. 4. Separate wireless receiver's USB cable and power cable.		

2.69 OP-077 Driver parameter read and write permission has changed.

Alarm ID	OP-077	Alarm title	【Driver parameter read and write permission has changed.】
Description	Driver parameter read and write permission has changed. Please reboot controller and driver to read driver parameter.		
Possible Cause	1. Connected driver parameter read and write permission isn’t fixed. It depends on driver parameter. 2. System detects driver parameter read and write permission isn’t complete open, and will automatically activate read and write permission.		
Solution	Wait for a while after power on, reboot controller and driver to remove alarm. Note: Some brands’ driver need to wait for driver to save parameter. And each brand's saving time needed is different. If user cut off power too early, the alarm may still exist. It is recommended to wait for one minute.		

2.70 OP-078 Unexpected Error

Alarm ID	OP-078	Alarm title	【Unexpected Error】
Description	Controller detects unexpected error.		
Possible Cause	An unexpected condition appears.		
Solution	Please contact Syntec OEM.		

2.71 OP-079 Assigned connect device doesn’t exist.

Alarm ID	OP-079	Alarm title	【Assigned connect device doesn’t exist.】
Description	Can’t find assigned connect device.		
Possible Cause	1. Controller network card doesn’t exist. 2. EnIP Macro program assigned ‘Network card number’ or ‘Slave ID’ is incorrect.		
Solution	1. Please contact Syntec OEM. 2. Check EnIP Macro program assigned ‘Network card number’ or ‘Slave ID’ whether is correct.		

2.72 OP-080 Driver initialize. Take too much time to pre-read.

Alarm ID	OP-080	Alarm title	【Driver initialize. Take too much time to pre-read.】
Description	Controller can’t communicate with driver normally, or driver parameter takes too much time to pre-read.		
Possible Cause	1. Controller can’t communicate with driver normally. 2. A critical alarm on the drive caused the initialization to fail. 3. Take too much time to read driver parameter.		
Solution	1. Reboot controller and driver. 2. Solve all drive critical alarms. 3. Decrease Pr3202 interpolation time, then reboot controller and driver. 4. Please contact Syntec OEM.		

2.73 OP-081 ATC has an critical alarm

Alarm ID	OP-081	Alarm title	【ATC has an important alarm】
Description	ATC (auto tool change) has an critical alarm, CNC and ROT will stop to avoid machine crashing.		
Possible Cause	ATC process cannot be performed successfully, ATC has an critical alarm.		
Solution	After troubleshoot ATC alarm, Reset to remove alarm.		

2.74 OP-082 ESI file read failed

Alarm ID	OP-082	Alarm Title	【ESI file read failed】
Description	Abnormal to read the ESI file imported by the user.		
Possible Cause	1. User imports more than ESI files		
Solution	1. Delete ESI file user imported with SA. 2. After deleting ESI file, must re-import ESI file to communicate with corresponding EtherCAT driver.		

2.75 OP-083 Minimum effective interpolation time is 2ms on dual core systems

Alarm ID	OP-083	Alarm Title	【Minimum effective interpolation time is 2ms on dual core systems】
Description	The minimum effective interpolation time is 2ms on dual core systems. If the value of Pr3203 is set below 2,000, the controller will run as if it were 2,000.		
Possible Cause	The value of Pr3203 is less than 2,000.		
Solution	Modify the value of Pr3203 so that it is NO LESS THAN 2,000 and reboot the controller and the driver.		

2.76 OP-084 PLC.lad abnormal

Alarm ID	OP-084	Alarm Title	【PLC.lad abnormal】
Description	When the system loads the PLC ladder diagram, the file is found abnormal.		
Possible Cause	1. PLC.lad not exist. 2. PLC ladder diagram's execution line number more than 4000 lines (not include notation line numbers).		
Solution	1. Re-import PLC.lad and reboot controller. 2. PLC ladder diagram's execution line number should not exceed 4000 lines (not include notation line numbers).		

2.77 OP-085 PLC ladder diagram syntax error

Alarm ID	OP-085	Alarm Title	【PLC ladder diagram syntax error】
Description	When the system loads the PLC ladder diagram, the file is found syntax error.		
Possible Cause	PLC ladder diagram syntax error.		
Solution	1. Re-import correct PLC.lad and reboot controller. 2. Correct PLC ladder diagram in edit mode, and reboot controller.		

2.78 OP-087 Not Support Dynamic Equilibrium Function

Alarm ID	OP-087	Alarm Topic	【Not support dynamic equilibrium function】
Description	The grinder balancing function is not supported if Option41 "Grinder balancing function" is not enabled as a software option.		
Probable Reason	Turn on dynamic balance inspection or dynamic balance debugging without turning on the software option function Option41 "Grinder Dynamic Balance Function".		
Exclusion Method	If you want to use grinder dynamic equilibrium, please open the choose function Option41 "Grinder Dynamic Balance Function".		

2.79 OP-088 Dynamic Equilibrium Function Error

Alarm ID	OP-088	Alarm Topic	【Dynamic Equilibrium Function Error】
Description	Error occurred when enable spindle dynamic equilibrium function.		
Probable Reason	1. Spindle number setting error. 2. Open the dynamic equilibrium adjustment and dynamic equilibrium inspection simultaneously. 3. The controller could not open dynamic equilibrium in the mode. 4. The fill-in time Pr3203 is set too large. 5. Dynamic equilibrium function could only support Syntec driver 6. The driver does not support dynamic equilibrium function version.		
Exclusion Method	1. Following the step check controller parameter setting.: a. Check the setting of Pr1996 to confirm whether the spindle is specified to use the dynamic balancing function. b. Based on the spindle number specified in Pr1996, check Pr1621~ to make sure that the corresponding axis number is specified for that spindle; if so, check Pr21~ again to make sure that the correct axis card port number is specified for that axis number. 2. Please close the dynamic inspection or close. Please turn off the dynamic balance commissioning, or turn off the dynamic balance detection. 3. Please switch the controller to following mode: a. Automatic execution. b. MDI c. Continue inching. d. Incremental inching. 4. Please set Pr3203 bellow 2000. 5. Please use Syntec driver. 6. Please update the driver to version 2.5.1 or later.		

2.80 OP-089 Register of subscribed channel located in system reserved area

Alarm ID	OP-089	Alarm Title	Register of subscribed channel located in system reserved area
Explanation	The initial register of subscribed channel is set in the system reserved area.		
Possible Cause	1. The initial target register R(n) of subscribed channel is set to CNC system interface area.		

Solution	1. Assign the correct register

2.81 OP-090 Information setting error of subscribing channel

Alarm ID	**OP-090**	**Alarm Title**	**Information setting error of subscribing channel**
Description	Information setting error of subscribing channel when applying specific functions.		
Possible Cause	1. When applying the subscribing channel as dual feedback inspection function, the corresponding channel data is not position feedback (includes position feedback, controller monitoring position feedback, controller monitoring 2nd position feedback)		
Solution	1. Please check if the subscribing channel data set by the axis card port number corresponding to Pr241~Pr260 axis dual feedback is position feedback.		

2.82 OP-091 System has serious error before the previous power off, re-calibrate before machining

Alarm ID	**OP-091**	**Alarm title**	**【System has serious error before the previous power off, re-calibrate before machining】**
Description	The controller detected a serious error in the system before the previous power off.		
Possible Cause	The system has an unexpected error before the previous power off.		
Solution	1. Check whether machining setting data is correct. 2. Reboot controller and driver. 3. Please contact Syntec OEM, to confirm the cause of the serious error before the previous power off.		

2.83 OP-092 Dual feedback port setting does not support dual feedback control mode

Alarm ID	**OP-092**	**Alarm title**	**【Dual feedback port setting does not support dual feedback control mode】**

Description	Dual feedback control mode does not support obtaining feedback data through subscription channel
Possible Cause	Dual feedback setting is subscription channel while dual feedback mode is control mode. ex. Pr241 sets 10001, Pr681 sets 0
Solution	1. Modify dual feedback mode Pr681~ to detection mode 2. Modify Pr241~ dual feedback port setting

2.84 OP-093 Incorrect dual feedback port setting

Alarm ID	**OP-093**	**Alarm title**	**【Incorrect dual feedback port setting】**
Description	Incorrect dual feedback port setting		
Possible Cause	Port number of Pr241~Pr260 is not supported		
Solution	Please refer to Pr241~Pr260 parameter manual instruction to set correct number.		

2.85 OP-094 Setpoint Axis major alarm occurred

Alarm ID	**OP-094**	**Alarm title**	**【Setpoint Axis major alarm occurred】**
Description	Setpoint Axis has major alarm, CNC will stop to avoid machine crashing.		
Possible Cause	Setpoint Axis has alarm or drive alarm.		
Solution	After clearing Setpoint Axis alarm and drive alarm, this alarm will be cleared.		

2.86 OP-095 System has serious exception, please reboot the system

Alarm ID	**OP-095**	**Alarm title**	**【System has serious exception, please reboot the system】**
Description	The controller detected a serious error in the system, please reboot system immediately.		
Possible Cause	The system has an unexpected error, that will cause system instability.		

Solution	Reboot controller and driver.

2.87 OP-096 Loader axis group use non Syntec M3 drive

Alarm ID	**OP-096**	**Alarm title**	**【Loader axis group use non Syntec M3 drive】**
Description	The axis of loader axis group (Pr733, Pr734) can use only Syntec M3 drive or virtual axis.		
Possible Cause	The axis of loader axis group is not Syntec M3 drive and virtual axis.		
Solution	The axis of loader axis group change to use Syntec M3 drive or modify parameter Pr21~ to be virtual axis.		

Note: OP-096 support version: 10.118.48Y, 10.118.52S, 10.118.56M, 10.118.60G, 10.118.66A and earlier.

2.88 OP-097 Independent axis group (Pr741=1) check error

Alarm ID	**OP-097**	**Alarm title**	**【Independent axis group (Pr741=1) check error】**
Description	When Pr741 is 1, the system will make axis group, which specified by Pr733~ and Pr737, into independent axis group. Furthermore, the system will check the axis, which is belonged to independent axis group, to inhibit "one axis for multiple axis groups", "axis exchange" and "axis coupling" setting. The independent axis group is also inhibited to use setpoint axis.		
Possible Cause	1. Pr741 is set accidentally as 1. 2. When Pr741 is 1, the axis of independent axis group is also belonged to another axis groups. 3. When Pr741 is 1, the "axis exchange" and "axis coupling" parameter is set to axis of independent axis group. 4. When Pr741 is 1, there is at least one setpoint axis that is belonged to independent axis group.		
Solution	1. Pr741 is set to 0. 2. Check Pr701~ setting and confirm the axis of independent axis group is only used by an unique independent axis group. 3. Check Pr3721~, Pr3821~ setting and make sure the axis ID can not specify axis of independent axis group. 4. Check the setting of 【*Belonging to axis group ID】 of setpoint axis parameter, which can not use independent axis group ID.		

2.89 OP-098 The version of the serial parameter file is incompatible

Alarm ID	OP-098	Alarm title	【The version of the serial parameter file is incompatible】
Description	The serial parameter definition file placed during the driver update is incompatible.		
Possible Cause	The driver firmware version does not match the controller version.		
Solution	1. Update the driver firmware version to the controller compatible version. 2. Update the controller software version to the driver compatible version.		

2.90 OP-099 Illegal Setting for Independent Axis Groups (Pr732=2)

Alarm ID	OP-099	Alarm Title	【Illegal Setting for Independent Axis Groups (Pr732=2)】
Description	If Pr732 is set to 2, the following restrictions apply： 1. Except that Pr742 is set to 1, one axis can only be associated to ONE axis group. 2. Inter-Axis Group use of the following functions are NOT allowed: a. Axis Exchange. b. Axis Coupling. c. ECAM.		
Possible Cause	1. Pr732 is unexpectedly set to 2. 2. When Pr732 is set to 2, Pr742 is not set to 1, and at least one axis is assigned to multiple axis groups (Pr701~). 3. When Pr732 is set to 2, there exists inter-axis group use of the following functions: a. Axis Exchange. b. Axis Coupling. c. ECAM.		
Solution	1. Modify Pr732 so that it is not equal to 2. 2. Modify Pr701~ so that each axis belongs uniquely to ONE axis group. If the axis needs to be used among multiple axis groups, change Pr742 to 1 and use it as a roaming axis. 3. Modify Pr3721~, Pr3821~ so that there is no inter-axis setting. 4. Check and modify the setting of ECAM so that there is no inter-axis setting.		

2.91 OP-100 Illegal Trigger of Halted Point Return

Alarm ID	OP-099	Alarm Title	【Illegal Trigger of Halted Point Return】
Description	When the axis group independent function is turned on (Pr732 is set to 2), if multiple axis groups execute custom pause point return at the same time, Then only the axis group that first triggered the return of the custom pause point can be executed correctly. The rest of the axis groups will not be executed (will not be cyclically started).		
Possible Cause	When the axis group independent function (Pr732 is set to 2) and the custom pause point return function (Pr3852 is set to 1) are both enabled, Simultaneously trigger multiple axis groups to execute custom pause point return.		
Solution	After the execution of the axis group that first triggered the return of the custom pause point is completed, the cycle start of other axis groups is triggered again.		

2.92 OP-101 EtherCAT-IO communication error

Alarm ID	OP-101	Alarm Title	【EtherCAT-IO communication error】
Description	EtherCAT-IO communication is abnormal continuously for over 10 ms. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Cause	1. EtherCAT communication cable is loose. 2. Noise interference.		
Solution	1. Check if the communication cable is normal. 2. Replace communication cables. 3. Reboot controller and IO device.		

2.93 OP-102 EtherCAT devices exceeds the number of supported stations

Alarm ID	OP-102	Alarm Title	【EtherCAT devices exceeds the number of supported stations】
Description	EtherCAT devices connected in series exceeds the number of stations supported by controller.		

Alarm ID	OP-102	Alarm Title	【EtherCAT devices exceeds the number of supported stations】
Possible Cause	More than 16 EtherCAT devices are connected in series.		
Solution	Please reduce the number of EtherCAT device stations connected in series.		

2.94 OP-103 Drivers connected don't support SPA3.0

Alarm ID	OP-103	Alarm Title	【Drivers connected don't support SPA3.0】
Description	The current configuration of controller and drivers don't support SPA3.0.		
Possible Cause	Controller doesn't connent any Syntec driver or software version of Syntec driver is lower than 4.4.0, 5.2.2.		
Solution	1. If there is no Syntec driver connected: change Pr3850 to be 0, use SPA2.0 instead. 2. If there is a connected Syntec driver: update software of Syntec driver to 4.4.0, 5.2.2 or higher.		

2.95 OP-104 Inclined Axes control configure failed

Alarm ID	OP-104	Alarm Title	【Inclined Axes control configure failed】
Description	Configuring inclined axes control fails.		
Possible Cause	1. Inclined axis or orthe axis's axis type is rotary axis. 2. Inclined axis or orthe axis is assigned to different coordinate or multiple coordinates. 3. Inclined axis or orthe axis is actived axis exchange function. 4. Inclined axis or orthe axis is actived roaming axis function. 5. Inclined axis or orthe axis is actived axis coupling function.		
Solution	1. Please modify Pr221~, and assign inclined axis or orthe axis's axis type is linear axis. 2. Please modify Pr701~， and assign inclined axis and orthe axis are same single coordinate. 3. Please modify Pr3721~ so that inclined axis or orthe axis isn't actived axis exchange. 4. Same as no.2 suggestion， when inclined axis and orthe axis are assigned same single coordinate, it don't deal with as roaming axis. 5. Please modify Pr3821~ so that inclined axis or orthe axis isn't assigned axis		

2.96 OP-105 Indexing Axis configure failed

Alarm ID	OP-105	Alarm Title	【Indexing Axis configure failed】
Description	Configuring indexing axis fails.		
Possible Cause	1. Indexing axis is assigned to non first group or multiple groups. 2. Axis type of indexing axis is linear axis or rotary axis Type C~E. 3. Indexing axis is assigned to spindle. 4. Indexing axis is actived axis exchange function. 5. Indexing axis is actived axis coupling function.		
Solution	1. Please modify Pr701~， and assign indexing axis to first group. 2. Please modify Pr221~, and assign axis type of indexing axis is rotary axis Type A or Type B. 3. Please modify Pr1621~, don't assign indexing axis as spindle. 4. Please modify Pr3721~ so that indexing axis isn't actived axis exchange. 5. Please modify Pr3821~ so that indexing axis isn't assigned to coupling axis.		

2.97 OP-106 Tension Control Axis ID Conflict

Alarm ID	OP-106	Alarm Title	【Tension Control Axis ID Conflict】
Description	Detecting tension control master and slave axis ID conflict with each other, or reuse at other tension control group.		
Possible Cause	• The value of Pr2901 and Pr2906(master ID of tension control) are same. • The value of Pr2902 and Pr2907(slave ID of tension control) are same. • One of master ID of tension control groups is same with slave ID of other group.		
Solution	• Change the conflicted axis ID as mentioned above to other axis ID which is not assigned to tension control, or set master ID and slave ID to 0 for disabling conflicted tension control.		

SYNTEC

3 System Alarm -SYS

Alarm ID	SYS-001	Alarm Title	Critical system error
Description	Unexpected error happened while controller running.		
Possible Cause	System memory operation failed.		
Solution	Do system backup immediately, contact Syntec OEM and restart the system.		

Alarm ID	SYS-002	Alarm Title	Important file error
Description	Error happened due to file lost or corruption.		
Possible Cause	1. Servo communication driver module loading failed. 2. IO driver loading failed. 3. Other driver module loading failed. 4. System ORX module loading failed. 5. System data loading failed. 6. Motion control module loading failed. 10. High speed high precision parameter data loading failed		
Solution	Do system backup immediately, contact Syntec OEM and restart the system.		

1. Sub Alarm SYS-002-1【Servo comm. driver loading failed】
   Servo communication modules such as M3 and ECAT.
2. Sub Alarm SYS-002-2【I/O driver loading failed】
3. Sub Alarm SYS-002-3【Other driver loading failed】
   Other driver module such as SRI.
4. Sub Alarm SYS-002-4【System ORX module loading failed】
   For example, System ORX module loading failed or initialize failed
5. Sub Alarm SYS-002-5【System data loading failed】
   For example, XML file loading failed.
6. Sub Alarm SYS-002-6【Motion control module loading failed】
   For example, the failure to load or initialize the plug-in path handling module.
7. Sub Alarm SYS-002-10【High speed high precision parameter data loading failed】
   For example, the failure to load or initialize the HSHPParam.dat.

3.1 System Alarm -SYS -Important Note

3.2 SYS-001 Critical system error

Alarm ID	SYS-001	Alarm Title	Critical system error

Description	Unexpected error happened while controller running.
Possible Cause	System memory operation failed.
Solution	Do system backup immediately, contact Syntec OEM and restart the system.

3.3 SYS-002 Important file error

Alarm ID	**SYS-002**	**Alarm Title**	**Important file error**
Description	Error happened due to file lost or corruption.		
Possible Cause	1. Servo communication driver module loading failed. 2. IO driver loading failed. 3. Other driver module loading failed. 4. System ORX module loading failed. 5. System data loading failed. 6. Motion control module loading failed. 10. High speed high precision parameter data loading failed		
Solution	Do system backup immediately, contact Syntec OEM and restart the system.		

1. Sub Alarm SYS-002-1【Servo comm. driver loading failed】
   Servo communication modules such as M3 and ECAT.
2. Sub Alarm SYS-002-2【I/O driver loading failed】
3. Sub Alarm SYS-002-3【Other driver loading failed】
   Other driver module such as SRI.
4. Sub Alarm SYS-002-4【System ORX module loading failed】
   For example, System ORX module loading failed or initialize failed
5. Sub Alarm SYS-002-5【System data loading failed】
   For example, XML file loading failed.
6. Sub Alarm SYS-002-6【Motion control module loading failed】
   For example, the failure to load or initialize the plug-in path handling module.
7. Sub Alarm SYS-002-10【High speed high precision parameter data loading failed】
   For example, the failure to load or initialize the HSHPParam.dat.

SYNTEC

4 Axial Motion Alarm -MOT

Alarm ID	**MOT-001**	**Alarm Title**	**A,B encoder feedback signal error**
Description	axis card detects A/B encoder feedback error.		
Possible Cause	Only happens when Pr9 *Servo board type sets as 0 (EMP2), when axis board type sets as EMP2, axis card will auto-detect A,B encoder feedback. If there is signal error, or has A signal but no B signal, this alarm will be issued.		
Solution	Check the servo cable or replace the axis board.		
Alarm ID	**MOT-002**	**Alarm Title**	**Error counter overflow**
Description	Axis board detects overflow in encoder feedback.		
Possible Cause	Only happen when servo type is set to 0(EMP2), 4(PMC4), 6(SERVO6), axis board will automatically detect the A, B encoder signals, if signal error or input signal too large, counter overflow alarm issued.		
Solution	Check the servo cable or replace the axis board.		
Alarm ID	**MOT-003**	**Alarm Title**	**Encoder module error**
Description	Currently no application.		
Possible Cause			
Solution			
Alarm ID	**MOT-004**	**Alarm Title**	**No index interrupt signal**
Description	Currently no application.		
Possible Cause			
Solution			
Alarm ID	**MOT-005**	**Alarm Title**	**DDA command overflow**

Description	Controller sends too many commands. In the one interpolation time interval, software calculates that the number of commands sent is more than 2047 pulses.
Possible Cause	1. DDA software time setting value is too long. 2. Motion velocity is too fast. 3. Servo resolution is set too high. 4. Backlash compensation or pitch compensation is too large. 5. Feed forward Compensation is enabled before booting.
Solution	1. Recommend that low interpolation time interval setting (parameter 3203) is not less than 2000. 2. Reduce the feedrate limit (Pr461-Pr480) 3. Reduce the servo resolution setting (driver and CNC Pr61-Pr80) 4. Set appropriate mechanical compensation time constant (parameter 1401~1420). 5. Cancel feed forward compensation function (parameter 581~600). 6. Please contact staff of machinery manufacturer to solve problem.
More description	In order to achieve the multi-axis coordinated control, SYNTEC's controller uses DDA (Digital Differential Analyzer), Cycle Time of DDA is set by parameter Pr 3203. In one Cycle time of DDA, every axial is allowed to send maximum 2047 pulses. Once exceeding this value, controller will send alarm

Alarm ID	**MOT-006**	**Alarm Title**	**Driver power supply status error**
Description	When controller outputs movement command, it detects driver power status error (Not Servo On or not power on). (This alarm is only appropriate for situation when Yaskawa driver blocks A.95x warning).		
Possible Cause	1. Driver's Power-amp voltage isn't stable. 2. Cut off driver power-amp voltage when emergency stop pushed. 3. Driver malfunction.		
Solution	1. Check if the power supply of driver is normal. 2. Check if the power cable of driver is loose or broken. 3. Replace the driver.		

Alarm ID	**MOT-007**	**Alarm Title**	**Driver power-off**
Description	Driver input power is cut off.		
Possible Cause	1. power failure. 2. Main power is shut down.		
Solution	Ensure the controller is powered off completely. Confirm the power supply and reboot.		

Alarm ID	MOT-008	Alarm Title	Loss Pulse
Description	1. After sending the last command, start to check whether the difference between the feedback and the command exceeds the loss pulse check window(Pr561~Pr580). If it exceeds, this alarm will be issued. 2. hardware model is not within the supported software version.		
Possible Cause	1. Kinematic occurs obstruction phenomenon. 2. Servo drive occurs unexpected Servo ON / OFF. 3. CPU board send the data to axis board unsuccessfully (CPU board or axis board has problem, the contact between CPU and axis board is not good). 4. Unstable position feedback at idle. 5. The cable that sends command from controller to servo driver has poor quality or is disconnected. 6. Controller doesn't set servo drive alarm check, controller continues to send motion command although the drive is abnormal. 7. Local interference. 8. Improper setting of Loss pulse check window (Pr561~Pr580) 9. 10, EZ series controller installed 9.242, 10.112 and below versions software, or option software is not activated.		
Solution	1. Do not shut down the controller when alarm occurs. Please check whether the value of No 8, 9, 10 in diagnose function is zero 2. Check whether the mechanical lubrication system is good. 3. Open the cover of axial to check whether foreign matter blocks the motion of axial. 4. Rotate screw to check whether machine is stuck (loading of driver) 5. Execute servo tuning to suppress the vibration frequency. 6. Check the drive servo-on and the servo-off of power or cable signal 7. If the setting value of No 8, 9, 10 in diagnostic function do not change, please take home search action (don't need to reboot), after that check whether parameters 24, 25, 26, 40, 41, 42 are equal to zero, if the parameters 24, 25, 26 are not equal to zero, the feedback loop has problems 8. If the parameters 40, 41, 42 are not equal to zero, command transmission from controller to the motor has been lost pulse. 9. If all parameters 24, 25, 40, 41, 42 are not zero, then the interference signal is relatively large, specifically in the machining process, the setting value of parameters 8, 9, 10 gradually become large. The reason is the contact point between CPU board and axis card is not and axis board is not good. Try to replace CPU board and axis board. 10. Adjust Loss pulse check window (Pr561~Pr580). Recommended values are as follows; a. linear axis : 100 b. rotary axis : 500 11. Confirm if hardware model is compatible with software version, or if option software Option2 is activated.		

More description	Settings of Pr561~Pr580 is the check the range of loss pulse. System Data 8 [X axis following error value] System Data 9 [Y axis following error value] System Data 10 [Z axis following error value] System Data 24 [X axis absolute position feedback value] System Data 25 [Y axis absolute position feedback value] System Data 26 [Z axis absolute position feedback value] System Data 40 [X axis absolute position command value] System Data 41 [Y axis absolute position command value] System Data 42 [Z axis absolute position command value]

Alarm ID	**MOT-009**	**Alarm Title**	**Servo Driver Alarm**
Description	Drive issued alarm.		
Possible Cause	Drive alarm mostly is because of external Causes; i.e. 1. Driver overheat. 2. Encoder wiring error. 3. Internal parameters is set wrong. 4. Servo motor is incompatible. 5. driver malfunction.		
Solution	Follow the troubleshoot steps in driver alarm manual to solve alarm.		

Alarm ID	**MOT-010**	**Alarm Title**	**Servo position command comm. error**
Description	Once the communication between Kernel and axis card has errors, software will check whether the queue value in internal IC of axis control is not zero.		
Possible Cause	1. There is only one axis card, but parameter sets two axis cards, and servo axis points to the second axis card. 2. One axis card has errors in case controller has two more axis cards. 3. Two or more axis cards, IRQ11 Jump is plugged. In diagnosis function, number 23 is not equal to 100. 4. Servo board pulse source parameter(Pr11) setting error.		
Solution	1. Check whether the parameter setting Pr11, Pr13 are consistent with the hardware feature. 2. Check axis card jump setting. 3. Change axis card.		
More description	Each interpolation time interval, Kernel (core) software will check whether the QUEUE value FLAG is correct. After one FILTER, if it reads the error value, alarm will appear and diagnose function number 68 will be added 1.		

New Old

Versions: 10.118.40O, 10.118.41O, 10.118.48 and later.

Alarm ID	MOT-011	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in drive communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded. If it is M2 communication, add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

New Old

Versions: 10.118.40N, 10.118.41N, 10.118.47 and older.

Alarm ID	MOT-011	Alarm title	Drive communication error
Description	Abnormal driver communication		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the driver is loose. 3. The quality of the communication cable is poor, or there is noise interference. 4. The Serial Bus M2 communication cable is an older version of the cable with poor noise resistance ability. 5. The communication time setting of RTEX driver is different from Pr3203. 6. The EtherCAT drive does not have a matching ESI file.		

Solution	1. Check whether the driver parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the driver. 3. Check whether the machine is properly grounded. If it is M2 communication, add a terminal resistor to the communication port of the end driver. 4. Check if there is a yellow label sticker written V4 on the M2 communication cable of the serial bus. 5. Set the communication time of RTEX driver to the same as Pr3203. 6. Check if the ESI file is imported correctly. 7. Contact Syntec OEM.

Alarm ID	**MOT–012**	**Alarm title**	**Driver homing error**
Description	Serial Bus Driver homing unsuccessfully.		
Possible Reason	Homing method (Pr961) is set incorrectly or driver doesn’t support homing function.		
solution	Check whether homing method is set correctly or driver supports homing function.		

Alarm ID	**MOT-013**	**Alarm Title**	**Axial tuning failed. Reboot controller.**
Description	After one serial bus axis tuning function fails, under un-reboot condition triggers axis JOG, MPG JOG or Homing functions.		
Possible Reason	Serial bus axis tuning fails.		
Solution	Reboot controller.		

Alarm ID	**MOT-014**	**Alarm Title**	**Enabled too many serial axis**
Description	Over system serial bus axis number limit.		
Possible Reason	When interpolation time sets too short or using serial axis are too much and cause it’s over limit serial bus axis number in interpolation time. And this alarm is issued.		
Solution	Configure the interpolation time to increase or decrease the number of serial axis used.		

Alarm ID	**MOT-016**	**Alarm Title**	**Absolute type data error. Check the encoder power and reboot the driver**
Description	Detects absolute type encoder is abnormal, and controller has reset. Please reboot driver.		
Possible Reason	1. First time absolute encoder connects to power, do encoder setting. 2. Encoder runs out of power. Position data is cleared. 3. After driver connects to power, encoder battery voltage is too low.		
Solution	1. Check if the battery and connection status of the absolute encoder is proper. 2. Check if the battery voltage is stable and complies with the rating voltage required by the absolute encoder. 3. After the above check is completed, reboot controller and driver.		

Alarm ID	**MOT-017**	**Alarm Title**	**First Positive software limit exceed**
Description	Axial machine coordinate is over positive software stroke limit which is set by Pr2401~Pr2440.		
Possible Cause	Stroke movement of machine table exceeds the set value.		
Solution	1. Reset 2. Move axis in negative direction to leave the software stroke limit protection.		

Alarm ID	**MOT-018**	**Alarm Title**	**First Negative software limit exceed**
Description	Axial machine coordinate is over negative software stroke limit which is set by Pr2401~Pr2440.		
Possible Cause	Stroke movement of machine table exceeds the set value.		
Solution	1. Reset. 2. Move axis in positive direction, to leave the software stroke limit protection.		

Alarm ID	**MOT-019**	**Alarm Title**	**Following error exceed**
Description	1. Because of the characteristics of servo, servo motor location, there is no way to respond the command of controller immediately, so a slow phenomenon appears, when this latency is not in allowed range, controller will send out the alarm. 2. Hardware type doesn't support software version.		

Possible Cause	1. Mechanism movement is not smooth. 2. Cable contact is poor. 3. Setting values of acceleration and deceleration time are too small. 4. Servo On /Off Relay is interfered. 5. Internal loop in driver parameter gains too small. (Under 10.116.10 and 10.117.10 or newer version, gain parameter (Kp) in controller and driver will synchronize automatically, so it doesn't need to consider this factor). 6. Encoder resolution and electric gear ratio is wrongly set. 7. Drive or motor malfunction. 8. Encoder or cable between encoder and controller is abnormal. 9. System Data number 23 is not equal to 100. 10. 10 and EZ serial controller installs to 9.242, 10.112 or older version, or option software function is not activated.
Solution	1. Add lubricating oil to mechanism. 2. Confirm wire connecting is normal. 3. Increase combination or acceleration and deceleration time. 4. When controller under dry run mode, open wiring cabinet to check whether servo on/off relay pulses is abnormal. 5. Increase Inner loop gain of driver. 6. Confirm hardware type and software version is matching, or confirm option software function Option 2 is activated. 7. Contact to Syntec OEM.
More description	Maximum velocity setting value of G00 and home search is equal to setting parameter divided by Kp. This value multiplied by 2 is the setting range of controller. Reasonable following error: Ferr= speech in command/ setting value of loop gain Alarm allowed values= {max[(velocity of first stage in home search process), velocity G00 of each axis]/Kp}*2 i.e. Feedrate 1000mm/min, loop gain 30, accuracy 1um, Ferr = 1000*1000÷60÷30=555 System Data 32 [X axis reasonable following error] System Data 33 [Y axis reasonable following error] System Data 34 [Z axis reasonable following error]

Alarm ID	**MOT-020**	**Alarm Title**	**Cannot return to control mode when moving**
Description	When motor starts to servo on, such as emergency stop released, canceling monitor mode (C31 ~) or serial spindle switching to C-Axis mode, zero speed check for the motor does not pass within 400 ms.		
Possible Cause	1. Instant machine movement by hand during motor servo on procedure. 2. The drive gain setting is poor. Therefore, motor will be trembled during servo on procedure. 3. Zero speed check window(Pr901~) is set too small.		

Solution	1. Avoid man-made movement. 2. Check the drive's position loop gain and speed loop gain setting. 3. Enlarge Pr901~ setting. Note, after removing alarm: • For incremental system, press reset can remove the alarm. • For absolute encoder system, system enters Not Ready state, need to reboot controller to remove the alarm.

Alarm ID	**MOT-021**	**Alarm Title**	**Must re-homing**
Description	When user must re-homing, the controller issue this alarm.		
Possible Cause	1.MOT-020 [Cannot return to control mode when moving] is triggered. 2.MOT-022 [Home position inaccurate] is triggered. 3.Pr41~、Pr121~、Pr161~、Pr221~ is changed.		
Solution	1.Please remove alarm MOT-020 [Cannot return to control mode when moving] . Press reset or reboot controller. 2.Please remove alarm MOT-022 [Home position inaccurate]. Press reset or reboot controller. 3.Press reset and re-homing according to current axis setting.		

Alarm ID	**MOT-022**	**Alarm Title**	**Home position inaccurate**
Description	After boot and complete homing for the first time, controller will set servo command (System Data 40~) and machine coordinate (Diagnose variable 72~) to zero. Or set an offset value according to Pr881~. After booting, at the N(N>1) times of searching home, home grid will be compared to the result of the first time searching home, if the error is over 0.1 turn of motor, the controller will send alarm.		
Possible Cause	1. Homing signal of motor is abnormal. 2. Stopper, coupling or bearings is not locked tightly. 3. Home grid function (Pr941~) is not activated. 4. Under serial bus environment, when that axis is also as spindle, Pr881~ and Pr1771~ setting is difference, will cause homing position change after homing.		
Solution	1. Move motor in the same direction and observe whether position counter index changes normally. 2. Check whether the mechanism components are fixed properly. 3. If it’s possible cause 4., assuming sixth axis setting is as first spindle, please sets Pr886 and Pr1771 to same value.		

Alarm ID	MOT-023	Alarm Title	Fatal following error exceed
Description	Because of the characteristics of servo, servo motor's positioning cannot respond immediately to controller's command, and delay phenomenon will appear, when this delay phenomenon is not in the allowed limit, controller will send alarm.		
Possible Cause	1. Servo motor doesn't receive control due to external force. 2. Parameter of drive - inner loop gain is too small. 3. Parameters of acceleration and deceleration time is set too short. 4. Encoder is abnormal or connecting encoder to controller is abnormal.		
Solution	1. Check the external motion of machine table. 2. Check the setting parameter of drive. 3. Check the acceleration and deceleration setting of each axis, Pr401, Pr541~Pr560. 4. Maintain the connection between encoder and servo drives.		
More description	Maximum velocity value of G00 and home search is equal to setting parameter divided by Kp. This value multiplied by 4 is setting range of controller. Reasonable following error: Ferr= speech in command/ loop gain Alarm allowed values= {max[(velocity of first stage in home search process), velocity G00 of each axis]/Kp}*4 System Data 32 [X axis reasonable following error] System Data 33 [Y axis reasonable following error] System Data 34 [Z axis reasonable following error]		

Alarm ID	MOT-024	Alarm Title	Fatal dual feedback error exceed
Description	If controller discovers that the command and the second command of encoder feedback exceed allowable limit set in Pr3817, controller will send this alarm.		
Possible Cause	1. Mechanism interference. 2. Motor parameter setting error. 3. Linear scale parameter setting error. 4. Servo motor feedback signal abnormal or has interference. 5. Linear scale signal abnormal or has interference. 6. Linear scale reader loose and signal receiving unstable.		
Solution	1. Check external motion mechanism. 2. Check if motor resolution setting Pr61~, Pr81~ and Pr61~ is correct. 3. Check if linear scale resolution setting Pr261~ and Pr301~ is correct. 4. Check if motor encoder function well, or move motor feedback direction away from high power electromagnetic devices. 5. Check if linear scale power is stable, or move linear scale feedback direction away from high power electromagnetic devices. 6. Confirm reader will not be affected by mechanism stroke movement.		

Alarm ID	**MOT-025**	**Alarm Title**	**Positive hardware limit exceed**
Description	Servo motor touches the positive hardware limit switch in movement.		
Possible Cause	1. Machine table exceeds protection point. 2. Hardware stroke switches are damaged or broken 3. Controller input signal error.		
Solution	1. Use MPG mode to move machine table to opposite direction once discovering that machine table stops at the switch. 2. If machine table is not at the switch, check switch device, limit switch wiring, IO terminal, 24V power supply of terminal. 3. Check whether IO card is abnormal.		
Alarm ID	**MOT-026**	**Alarm Title**	**Negative hardware limit exceed**
Description	Servo motor touches the negative hardware stroke limit in moving.		
Possible Cause	1. Machine table exceeds protection point. 2. Hardware stroke switches are damaged or broken. 3. Input signal has errors.		
Solution	1. Use MPG mode to move machine table in opposite direction once the machine table stops at the switch. 2. If machine table is not at the switch, check limit switch, switch wiring, IO terminal, 24V power supply of terminal. 3. Check whether IO card is normal.		
Alarm ID	**MOT–027**	**Alarm Title**	**program error in PLC axis**
Description	PLC axis program syntax error.		
Possible Cause	PLC axis program syntax error.		
Solution	Check syntax of PLC axis program.		
Alarm ID	**MOT-028**	**Alarm Title**	**System memory too low**
Description	When CNC axis and PLC axis exchange, system remaining memory is too low.		
Possible Cause	During machining process, switch axis into PLC axis.		

Solution	Contact machinery manufacturers.		
Advance Description	Kernel software will check value of system data number 7 ‘System remaining memory’ at any time. When remaining value is too low, this alarm will be issued.		
Alarm ID	**MOT-029**	**Alarm Title**	**Miss index in homing**
Description	When searching home, if motor does not find out motor index signal after leaving home DOG more than 5 pitches, controller will send this alarm.		
Possible Cause	1. Can’t read the index signal. 2. The setting of homing 2nd travel feedrate is too fast. 3. The setting of motor reduction ratio is too big. 4. The distance between index signal and HomeDog is more than 5 pitches.		
Solution	1. Check motor index signal wiring; observe system data no. 48(X), 49(Y), 50(Z) to check whether index signal is read. If no, please check whether wiring is normal. 2. Reduce setting value of the homing 2nd travel federate (Pr841~Pr843)		
More description	When searching home, machine will use the velocity setting value of the first stage to move to home DOG, and stop. After that machine moves backward with velocity of the second stage. After leaving home DOG to move backward, it start to search the nearest motor index signal. In the second stage, controller will calculate according to resolution of encoder. If controller leaves home DOG more than 5 pitches and can not find out the index signal. Controller will send alarm.		
Alarm ID	**MOT-030**	**Alarm Title**	**Homing zero speed check failed**
Description	When motor touches HomeDog, motor cannot stop completely.		
Possible Cause	1. Improper drive gain setting, and makes motor vibrating. 2. Resonance phenomenon caused by motor running.		
Solution	1. Check the position loop gain and velocity loop gain setting of driver. 2. Activate the resonance frequency inhibition ability of driver. 3. Contact machinery manufacturers for help.		
More description	When searching home, machine will use the velocity setting value of the first stage to move to home DOG, and stop once it meets home DOG. After that machine moves backward with velocity of the second stage. After leaving home DOG to move backward, it start to search the nearest motor index signal. At the first stage to find the home DOG, motor will decrease velocity to stop. After 0.1 second command stops, if system data no. 8(X), 9(Y), 10(Z)-error register receives values bigger than zero speed check window(Pr901~Pr920), controller will send alarm.		

Alarm ID	MOT-031	Alarm Title	Static dual feedback error exceed
Description	After the controller stops sending the motion command, during the time set by Pr3805, the system will check whether dual feedback error exceeds allowed limit set by Pr1421~Pr1440. If yes, controller will send alarm.		
Possible Cause	1. Mechanism interference. 2. Motor parameter setting error. 3. Linear scale parameter setting error. 4. Motor feedback signal abnormal or is interfered. 5. Linear scale signal abnormal or is interfered. 6. Linear scale reader is loose and cause the feedback unstable.		
Solution	1. Check the external motion mechanism. 2. Check if motor resolution setting Pr61~, Pr81~ and Pr61~ is correct. 3. Check if the linear scale resolution setting Pr261~ and Pe301~ is correct. 4. Check if motor encoder works well, or move motor feedback line away from high power electromagnetic devices. 5. Check if linear scale power is stable, or move linear scale feedback line away from high power electromagnetic devices. 6. Confirm the reader is not interfered by the mechanism movement.		

Alarm ID	MOT-032	Alarm Title	Z axis Following spindle error exceed
Description	For tracking Tapping feeds, feeding axis opposite direction will feed more than one pitch distance.		
Possible Cause	Spindle feedback cable wiring is opposite.		
Solution	1. Swap the position feedback A+ and A- which inverter sends to the controller. 2. Change inverter related parameters.		

Alarm ID	MOT-033	Alarm Title	Absolute encoder read error
Description	When using absolute encoder, the communication between the controller and the driver fails.		
Possible Cause	1. The driver is not powered on. 2. The communication cable between the controller and driver is loose. 3. The absolute adapter board is damaged.		
Solution	1. Check if the power supply of driver is normal. 2. Check if the wiring is correct. 3. Change the absolute adapter board.		

Alarm ID	MOT-034	Alarm Title	Absolute home position has not been set. Please reset it.
Description	When using absolute encoder, need to set up absolute home position in controller.		
Possible Cause	1. The absolute home position has not been set or it is lost due to insufficient voltage. 2. Pr41~, Pr61~, Pr81~, Pr121~, Pr161~, Pr201~, Pr221~ and etc. parameters is modified. 3. Detect Yaskawa encoder sends abnormal alarm. (For example: A.C80, A.CA0) 4. Execute multi turns reset function on the controller. 5. An abnormal absolute position status of Syntec drive is detected.		
Solution	1. For possible reasons 1, 2, and 4: (Again) sets absolute home position. 2. For possible reason 3, 5: After solving abnormal encoder issue, sets up absolute home position again.		
More description	Absolute home position setting step: • Step 1: Move axis to the pre-set mechanical origin. • Step 2: Enter home position setting page, path: Parameter setting > Serial Parameter > Abosolute home position • Step 3: Choose the axis to be set with direction key, and click 'set machine origin'.		

Alarm ID	MOT-035	Alarm Title	absolute position battery failure
Description	When using absolute encoder, the communication between the controller and driver is successful, but the motor encoder position value is 0.		
Possible Cause	The battery of absolute encoder has no sufficient power.		
Solution	Change the battery.		

Alarm ID	MOT-036	Alarm Title	Can't leave home dog switch
Description	When searching home, after stop and return, if the axis moving exceed Pr981~ setting and still can't leave home dog switch.		
Possible Cause	HomeDog is damaged.		
Solution	Use the electrical multimeter to check whether the sensor of HomeDog is damaged or wiring is short circuit.		

More description	When searching home, machine will use setting value of the 1st phase feedrate to move towards home DOG (Pr881~Pr900 Axial home offset), and stop until reaches home dog switch. In the 2nd phase, controller will calculate according to encoder resolution. If controller leaves home DOG exceed the setting of Pr981~Pr1000 Axis homing 2nd protect revolution (encoder type) and cannot leave index signal, controller will send alarm.

Alarm ID	**MOT–037**	**Alarm Title**	**Second Positive software limit exceed**
Description	Axis machine coordinate exceeds Pr2501~Pr2540 set Second Positive software limit value.		
Possible Cause	The motion of machine table exceeds set value.		
Solution	1. Reset to remove alarm. 2. Move axis in negative direction to leave the software stroke protection range.		

Alarm ID	**MOT–038**	**Alarm Title**	**Second Negative software limit exceed**
Description	Axis machine coordinate exceeds Pr2501~Pr2540 set Second Negative software limit value.		
Possible Cause	The motion of machine table exceeds setting value.		
Solution	1. Reset to remove alarm. 2. Move axis in positive direction to leave the software stroke protection range.		

Alarm ID	**MOT-039**	**Alarm Title**	**Tapping severe over travel**
Description	Tracking Tapping theory is Z axis moves along with spindle feedback. If driver occurs an alarm that cause motor dry runs during tapping period, Z axis will still tap with motor feedback, and there might have risk of machine crash. Therefore, when Z axis is higher than the hole top (R point) or about 10 pitches from the hole bottom. System will send this alarm and stop (pause) the machine. Not supported version: 10.114.33E, 10.114.38F, 10.115.43D, 10.115.46A, 10.115.47 or above version.		
Possible Cause	Motor dry runs because of alarm.		
Solution	Check the cause of driver alarm.		

Alarm ID	MOT-040	Alarm Title	Dual feedback self-inspection error exceed
Description	After activating dual feedback, between linear scales' each index signals, accumulated A, B pulse count difference exceed Pr3818 setting.		
Possible Cause	1. Linear scale power is unstable. 2. Linear scale feedback is interfered. 3. Linear scale's adapter is interfered by magnetic field or electric field. 4. Linear scale's sensor reader is loose and cause unstable feedback.		
Solution	1. Independently us the power supply of linear scale. 2. Check if the case at CNC side is power pollution free, or the isolation cable is fallen. 3. Keep the linear scale adapter away from the heavy electricity area, or cover it with cooper sheet to reduce external interference. 4. Confirm the sensor reader will not be affected by the mechanism travel movement.		

Unable to render include or excerpt-include. Could not retrieve page.

Alarm ID	MOT-042	Alarm Title	Over the 3rd negative software travel limit
Description	Axis mechanical coordinate is over negative software stroke limit which is set by Pr2441~Pr2480.		
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press reset. 2. Move axis in positive direction, to leave the software stroke protection range.		

Alarm ID	MOT-043	Alarm Title	Enter the 1st software stroke limit protection range
Description	Axis machine coordinate enter software stroke limit's protection range which is set by Pr2401~Pr2440.		
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press reset. 2. Axis move in opposite direction, and leave the software stroke limit protection range.		

Alarm ID	**MOT-044**	**Alarm Title**	**Enter the 2nd software stroke limit protection range**
Description	Axis mechanical coordinate enter software stroke limit's protection range which is set by Pr2501~Pr2540.		
Possible Cause	Bed moves over setting limit.		
Solution	1. Press the reset. 2. Move the axis in opposite direction, to leave the software stroke protection range.		
Alarm ID	**MOT-045**	**Alarm Title**	**Enter the 3rd software stroke limit protection scope**
Description	Axis mechanical coordinate enter software stroke limit's protection range which is set by Pr2441~Pr2480.		
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press the reset. 2. Move axis in opposite direction, and leave the software stroke protection range.		

Alarm ID	**MOT-046**	**Alarm Title**	**Origin switch signal abnormal**
Description	For incremental type home searching, set 0 or 2 in home mode (Pr961~), system will move towards origin switch direction according to Pr861~ setting, and stop with deceleration order when finds original point switch. If machine runs over block because deceleration distance is too long, when system continues moving in opposite direction with 2nd phase homing feedrate, it needs to wait for origin switch's signal again shows up and disappear, then it can start finding index signal. When moving in opposite direction, if origin switch's signal doesn't appear, alarm will be issued if exceeds 'five times distance of 1st phase homing feedrate * G00 acceleration and deceleration time/ 2'.		
Possible Cause	1. Origin switch contact is poor. 2. Noise interference.		
Solution	1. Check if the origin switch signals are interfered by noise. 2. Check if the origin switch contact is poor contact.		

Alarm ID	**MOT-050**	**Alarm Title**	**Illegal control mode switch**
Description	Do illegal control mode switch to axis.		
Possible Cause	1. Switch the axis control mode from speed control into torque/ gap control. 2. Switch the axis control mode from torque control into speed/ gap/ spindle orientation control. 3. Switch the axis control mode from gap control mode into speed/ torque/ spindle orientation control. 4. Switch the axis control mode from spindle orientation control into torque/ gap control.		
Solution	Switch the axis control mode correctly.		
Alarm ID	**MOT–051**	**Alarm Title**	**Inhibit to cycle start while moving**
Description	Before all manual commands are sent, prohibit cycle start to prevent operation error.		
Possible Cause	Manual command (JOG, INJOG, and MPGJOG) cannot be sent successfully.		
Solution	Reset to remove alarm. Wait until machine stops, then cycle start.		
Alarm ID	**MOT-052**	**Alarm Title**	**Driver power-amp voltage not input**
Description	Driver power-amp power is not suppled, please check power cable wiring.		
Possible Cause	Driver power-amp power is not suppled.		
Solution	Check if the three phase of driver electricity is input correctly.		
Alarm ID	**MOT-053**	**Alarm Title**	**No absolute type functions for an increment encoder**
Description	Driver side's encoder is incremental or when its setting is incremental, controller side can’t start using absolute homing function.		
Possible Cause	Encoder is incremental or use as incremental. Sets Pr201~Pr220 as absolute encoder.		

Alarm ID	MOT-053	Alarm Title	No absolute type functions for an increment encoder
Solution	1. Check the encoder type. In case of a incremental encoder, configure the Pr201~Pr220 as an incremental encoder. 2. Check the driver internal and the related parameter setting of encoder, and configure the encoder in the driver as an increment type. 3. After modification, reboot controller and driver.		

Alarm ID	MOT-054	Alarm Title	Encoder type (Pr201~Pr220) changed
Description	Encoder type has changed.		
Possible Cause	Modified Pr201~Pr220 or driver internal and encoder related parameter setting.		
Solution	Reboot controller and driver.		

Alarm ID	MOT-055	Alarm Title	Rotation axis type C and double feedback unsupported by the absolute type
Description	When using absolute encoder, related parameter setting is wrong.		
Possible Cause	1. When using absolute encoder (Pr201~ sets as 3), axis's axial type stetting will be rotary axial type C (Pr201~ sets as 3). 2. When using absolute encoder (Pr201~ sets as 3) and start dual feedback function simultaneously (Pr241~ not sets as 0).		
Solution	1. When using an absolute encoder, configure the axis type of axial direction as the non-rotation axis type C and other types (Pr221~≠3). 2. Turn off the dual feedback function (Pr241~=0) or use an increment encoder (Pr201~=0).		
Advanced Explanation	1. When this alarm occurs will trigger 'MOT-034 first homing setting incomplete, absolute position can't operate', user need to reset home position. 2. This alarm won't be cleared until related parameter reset correctly. Whether need to reboot depend on parameter. a. Pr221~ effective after press reset. b. Pr201~ Pr241~ effective after reboot.		

Alarm ID	MOT-056	Alarm Title	Single-turn absolute encoder cannot be used as an increment type. Reconfigure the encoder parameter.
Description	Single-turn absolute encoder can’t be used as incremental encoder.		
Possible Cause	When encoder is absolute encoder, set Pr201~Pr220 as incremental encoder.		
Solution	1. Check the encoder type. In case of a single-turn absolute type, configure the Pr201~Pr220 as an absolute encoder. 2. Check the driver internal and encoder related parameter setting, set the use of the encoder in the driver as increment type. 3. After modification, reboot controller and driver.		

Alarm ID	MOT-057	Alarm Title	Drive cannot servo on
Description	Drive cannot servo on.		
Possible Cause	When controller send servo on signal, but drive can’t enter servo on state in 20 seconds. Controller will send this alarm.		
Solution	1. Please contact the drive supplier. 2. After resolving the problem that the drive cannot servo on, reset the controller to clear the alarm.		

Alarm ID	MOT-058	Alarm Title	To permanent save the driver parameter setting, execute under not ready state
Description	Pr181~, Pr1771 has synchronized into driver parameter, but axis is servo on and can’t execute save parameters.		
Possible Cause	When modified P1181~ and Pr1771~, axis is servo on.		
Solution	Execute permanent save of driver parameter under not ready state.		

Alarm ID	MOT-059	Alarm Title	Please modified resolution related parameters from the page of serial parameters
Description	The axis of the Syntec driver can synchronize the resolution related setting. If the sensor type of controller and driver are the same, the related controller parameters will be synchronized from the driver. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. The matched condition of sensor type (single feedback): • Driver parameter Pn-911 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-911 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 The matched condition of sensor type (driver dual feedback or semi-closed loop): • Driver parameter Pn-931 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-931 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 Axial resolution related parameters: • Pr61~Pr80 Axial encoder resolution • Pr81~Pr100 Axial encoder scaling factor Driver resolution related parameters: • Pn-902 Encoder Resolution • Pn-922 2nd Encoder Resolution • Pn-20E Electronic Gear Ratio (Numerator) • Pn-210 Electronic Gear Ratio (Denominator) Version information: • This alarm is supported in 10.118.41Q, 10.118.48B, 10.118.49 and later versions.		
Possible Cause	Modified the resolution related parameters in controller after the resolution synchronization of the Syntec driver.		
Solution	1. Reset the controller. 2. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. Then restart the controller and the driver.		
Alarm ID	**MOT-060**	**Alarm Title**	**Torque application configuration conflicted with driver configuration (Pn002.0)**
Description	Torque application setting is conflicted with Yaskawa driver parameter setting.		

Alarm ID	MOT-060	Alarm Title	Torque application configuration conflicted with driver configuration (Pn002.0)
Possible Cause	1. Controller needs to use torque limit mode, but Yaskawa driver parameter setting sets as torque compensation mode. 2. Controller needs to use torque compensation mode, but Yaskawa driver parameter setting sets as torque limit mode.		
Solution	Check if the driver parameter configuration is correct.		

Alarm ID	MOT-061	Alarm Title	Gap control mode unsupported by axis
Description	Axis doesn't support switching into gap control mode.		
Possible Cause	1. Only Syntec driver supports gap control mode. 2. Syntec driver's firmware is too old.		
Solution	1. Use Syntec driver. 2. Check the firmware version of the Syntec driver.		

Alarm ID	MOT-062	Alarm Title	Gap control mode cannot be switched during axial movement
Description	Axis is moving, can't change from position control into gap control.		
Possible Cause	When axis is moving, received the mode changing command.		
Solution	Mode can be switched after the axial movement stops completely.		

Alarm ID	MOT-063	Alarm Title	Axial sensor type not matching with home searching method
Description	Axis sensor type isn't 'Absolute encoder', but homing method setting is 'Absolute encoder index' or 'Absolute with origin switch', and the parameter is not matched.		
Possible Cause	Axis sensor type (Pr201~Pr220) sets as 0~2, and the corresponding home searching method (Pr961~Pr980) sets as 4 or 5.		
Solution	In case the axial sensor type (Pr201~Pr220) is configured to 0~2, the home searching method (Pr961~Pr980) can be configured to 0~3, not 4 or 5.		

Alarm ID	MOT-064	Alarm Title	Axis driver doesn’t support selected friction compensation
Description	That axis driver doesn’t support the selected friction compensation.		
Possible Cause	When Pr2921~ is set to 1, but driver does not support speed spike compensation, the alarm will be issued; when Pr2921~ is set to 2, but driver does not support torque spike compensation, the alarm will be issued.		
Solution	1. Set Pr2921~ as the friction compensation that the axis driver supports. Press reset can remove this alarm. 2. Drivers that support speed spike compensation include: a. Yaskawa M2/M3 serial bus driver (Controller support versions: 10.116.54K, 10.118.0F, 10.118.6 and later versions). b. Syntec M3 serial bus driver (Controller support versions: 10.118.12G, 10.118.16 and later versions, driver support versions: 2.8.3, 2.9.0, 2.10.0 and later versions). c. Panasonic RTEX serial bus A6N driver (Controller support versions: 10.118.21 and 10.118.12O and later versions). d. EtherCAT serial bus drivers that support speed spike compensation (Controller support versions: 10.118.48I, 10.118.52C, 10.118.54 and later versions), please refer to EtherCAT Drive Application Manual. 3. Drivers that support torque spike compensation include: a. Syntec M3 serial bus driver (Controller support versions: 10.118.40A, 10.118.41, and later versions, driver supported versions: 2.14.105, 2.15.0, 3.0 .0 and later versions). 4. RETX related regulation, please refer to RTEX Driver Application Manual.		

Alarm ID	MOT-065	Alarm Title	Axis driver doesn’t support power-off retract
Description	Driver doesn’t support power-off retract function.		
Possible Cause	Driver doesn’t support power-off retract function. When Pr1041~ Pr1060 setting is not 0 and this alarm will be issued.		
Solution	1. Only Syntec Bus M3 Driver supports power-off retract function. 2. Confirm Syntec Serial Bus M3 Driver’s firmware version. Support firmware versions are 2.10.1, 2.11.0 and above version.		

Alarm ID	MOT-066	Alarm Title	Axis driver firmware version is not supported
Description	Controller doesn’t support axis driver's firmware version.		

Alarm ID	MOT-066	Alarm Title	Axis driver firmware version is not supported
Possible Cause	Driver’s firmware version is too old.		
Solution	Upgrade driver firmware version. For Panasonic RETX A6N firmware version, upgrade to 1.23 or above version.		

Alarm ID	MOT-067	Alarm Title	Syntec encoder’s firmware upgrade finished. Please reboot driver.
Description	Syntec encoder's firmware upgrade is finished. Need to reboot driver.		
Possible Cause	Execute axis encoder's firmware upgrade.		
Solution	Reboot driver and controller.		

Alarm ID	MOT-068	Alarm Title	Syntec encoder information reading timeout
Description	Syntec encoder information reading timeout.		
Possible Cause	1. Communication between controller and driver is abnormal. 2. Driver condition is abnormal.		
Solution	1. Check if communication is loose. 2. Reboot driver.		

Alarm ID	MOT-069	Alarm Title	Axial feedback abnormal
Description	If the difference between the axial feedback value exceeds 4 times of the maximum speed set by the controller, the number of abnormal times will be accumulated by 1 each time. This alarm will be issued when two consecutive abnormalities occur; The maximum speed is the maximum value among Pr461 ~, Pr521 ~, Pr621 ~, Pr821 ~, Pr1801 ~(when using as servo spindle).		
Possible Cause	1. Communication between controller and driver is abnormal. 2. The following parameter settings are inappropriate; a. Pr461~Pr480 Maximum feed rate for axial rapid movement (G00) b. Pr521~Pr540 Axial JOG speed c. Pr621~Pr640 Axial maximum cutting feedrate d. Pr821~Pr840 Axial homing feedrate 3. If using servo spindle, Pr1801~Pr1810 Spindle maximum speed setting is inappropriate		

Alarm ID	MOT-069	Alarm Title	Axial feedback abnormal
Solution	1. Check if communication is connect well. 2. Re-set above mentioned parameters.		

Alarm ID	MOT-070	Alarm Title	Axial command abnormal
Description	The planned controller command exceeds 1.2 times of the distance that can be moved at the axial maximum speed; The maximum speed is the maximum value among Pr461~, Pr521~, Pr621~, Pr821~, Pr1801~ (when using as servo spindle).		
Possible Cause	1. The following parameter settings are inappropriate; a. Pr461~Pr480 Maximum feed rate for axial rapid movement (G00) b. Pr521~Pr540 Axial JOG speed c. Pr621~Pr640 Axial maximum cutting feedrate d. Pr821~Pr840 Axial homing feedrate 2. If using servo spindle, Pr1801~Pr1810 Spindle maximum speed setting is inappropriate		
Solution	Re-set above mentioned parameters.		

Alarm ID	MOT-071	Alarm Title	Interpolation severely overtime
Description	The interpolation timeout exceeds 10ms, causing controller fail to communicate with driver normally; If within 10ms, controller will perform proxy interpolation and continue to communicate with driver.		
Possible Cause	1. Frequent data transmission in the cloud, causes the controller system to be extremely busy, e.g. using dipole structure (the front stage simulator is connected to the back stage controller) 2. Parameter Pr3251 is set to 5.		
Solution	1. Reboot controller and driver. 2. .Check if Pr3251 is set to 5. If so, please take a look at "Pr3251 Touch Screen" page, and set Pr3251 to other value. 3. Please upgrade the BSP version to 9.35 or above. 4. Please contact OEM Syntec.		

Alarm ID	MOT-072	Alarm Title	Software stroke limit setting error when using single turn absolute encoder
Description	Pr2401~Pr2440 is not activated or is incorrectly set when using the single turn absolute encoder type.		
Possible Cause	1. The encoder type is single turn absolute. 2. When using the single turn absolute encoder type, the software stroke limit should be activated and its value must be set within the distance in one motor revolution.		
Solution	1. Correct Pr201~ encoder type. 2. When Pr201~ is single turn absolute encoder type, make sure that Pr2401~Pr2440 are not default values, i.e. the negative and positive limits are not -999999999 and 999999999 respectively. And the positive limit should be larger than the negative limit. Also, the stroke limit should be set within the distance in one motor revolution. 3. After correction, please reboot the controller and the drive.		

Alarm ID	MOT-073	Alarm Title	Fail to preload servo drive parameter
Description	The controller fails to preload necessary parameters from the drive during booting.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. Mismatch between the controller and the drive version.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please upgrade the controller version. 5. Please upgrade the drive version. 6. Please contact OEM Syntec.		

Alarm ID	MOT-074	Alarm Title	Fail to read absolute position from servo drive
Description	The controller fails to read the absolute position from the drive.		
Possible Cause	1. The controller can't communicate with the drive normally.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Please contact OEM Syntec.		

Alarm ID	MOT-075	Alarm Title	Serial communication initialize timeout
Description	The communication between the controller and the drive is established, but fails to complete the initialization process in 20 seconds.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. The drive issues critical alarms.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please lower the interpolation time (Pr3203), then reboot the controller and the drive. 5. Please contact OEM Syntec.		
Alarm ID	**MOT-076**	**Alarm Title**	**Servo drive parameter preload timeout**
Description	The communication between the controller and the drive is established, but fails to preload parameters in 20 seconds.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. Take too much time to read drive parameters.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please lower the interpolation time (Pr3203) and reboot the controller and the drive. 5. Please lower the PLC scan time (Pr3204) and reboot the controller and the drive. 6. Please contact OEM Syntec.		
Alarm ID	**MOT-077**	**Alarm Title**	**Fail to recover machine position**
Description	Fail to recover machine position from absolute encoder or Fram in 20 seconds during booting.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. The drive issues absolute encoder battery alarms.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot drive's absolute encoder battery alarms. 4. Please contact OEM Syntec.		

Alarm ID	MOT-079	Alarm 标题	【Inconsistent Resolution Settings】
说明	The resolutions of the controller and the drive are not set consistently. The settings are considered consistent if the following equation is satisfied: 1. Single feedback: Resolution (Pr61~) * Encoder Scaling Factor (Pr81~) = Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E). 2. Dual feedback or Semi-Closed Loop: Resolution (Pr61~) * Encoder Scaling Factor (Pr81~) = 2nd Encoder Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E).		
可能原因	1. Improper controller parameter settings: a. Encoder Resolution. b. Encoder Scaling Factor. 2. Improper drive parameter settings: a. Electronic Gear Ratio. b. Encoder Resolution.		
排除方法	1. Change the controller parameters so that the equation mentioned above is satisfied: a. Axes: Pr61~, Pr81~. b. Servo Spindles or Serial Rotary Spindles: Pr1651~, Pr1661~. 2. Change the drive parameters so that the equation mentioned above is satisfied: a. Electronic Gear Ratio: Pn-20E, Pn-210. b. Encoder Resolution: Pn-902, Pn-922. 3. To clear the alarm, a. Reset the controller if ONLY the controller parameter(s) is(are) modified. b. Restart both of the controller and the drive if the drive parameter(s) is(are) modified.		

Unable to render include or excerpt-include. Could not retrieve page.

Alarm ID	MOT–081	Alarm title	The hardware doesn't receive communication packet
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

Alarm ID	MOT–082	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

Alarm ID	MOT-083	Alarm title	Driver software misses communication packet
Description	Response packet Watchdog(M3) or Working Counter(EtherCAT) is wrong for over 10ms in M3 communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet Watchdog(M3) or Working Counter(EtherCAT) is wrong		
Solution	Contact Syntec OEM.		

Alarm ID	MOT-084	Alarm title	Friction compensation mode of drive and controller are discrepant
Description	The friction compensation of the controller (based on Pr2921~) conflicts with the one of the driver.		
Possible Reason	This alarm happens if any of the following occurs while Syntec driver is connected: 1. The friction compensation of the controller is selected as "Torque compensation" (Pr2921~ is set as 2) while the friction compensation of Syntec driver is deactivated (Pn290 is set as 0). 2. The friction compensation of the controller is NOT selected as "Torque compensation" (Pr2921~ is other than 2) while the friction compensation of Syntec driver is activated (Pn290 is set as 1).		
Solution	1. Press RESET to clear this alarm. This makes the controller re-activate the compensation mode of Pr2921~ and activate/ deactivate the friction compensation (Pn290) of Syntec driver accordingly. 2. Contact Syntec OEM.		

Alarm ID	MOT-085	Alarm title	Trigger PLC MoveAx components for the roaming axis in borrowing
Description	When the PLC MoveAx component is triggered, the axis cannot be borrowed by other axis groups or PLC Rn subprogram components.		
Reason	When Pr742 is set to 1, for the roaming axis that has been borrowed by other axis groups or PLC Rn subprogram components, trigger the PLC MoveAx component to move.		
Solution	After confirming that other axis groups or PLC Rn subprogram components have removed the roaming axis, trigger the PLC MoveAx component to move.		
Alarm ID	MOT-086	Alarm Title	Drive is abnormally switch to servo off state when running
Description	Drive is abnormally switch to servo off state when running.		
Possible Cause	When drive is abnormally changed from servo on state to servo off state, controller will send this alarm.		
Solution	1. Check if the drive power supply is normal. 2. Check if the wiring is correct. 3. After resolving the problem that the drive cannot servo on, reset the controller to clear the alarm.		
Alarm ID	MOT-087	Alarm Title	Inaccurate stop position. Please check if there are any mechanisms stuck
Description	After sending the last command, start to check whether the difference between the feedback and the command exceeds the loss pulse check window(Pr561~Pr580). If it exceeds, this alarm will be issued		
Possible Cause	1. Mechanical movement was hindered. 2. Unstable position feedback at idle. 3. Improper setting of Loss pulse check window (Pr561~Pr580)		
Solution	1. Check whether the mechanical lubrication system and track lubrication are good. 2. Open the cover of axial to check if any foreign matter blocks the motion of axial. 3. Rotate screw to check if the mechanism is stuck (loading of driver) 4. Use the automatic tuning function to adjust the servo and suppress the vibration frequency. 5. Adjust Loss pulse check window (Pr561~Pr580). Recommended values are as follows: a. linear axis : 100 b. rotary axis : 500		

Alarm ID	MOT-088	Alarm Title	Servo on failed. Unable to execute operation.
Description	The servo can not servo on, after cycle start or other manual function is triggered.		
Possible Cause	1. When power on, axis doesn't complete magnetic pole angle detection which cause servo disabled. 2. Driver alarms or abnormal power supplication will cause servo driver become servo off. 3. Driver is malfunction.		
Solution	1. After power on, wait for a while before operation. 2. Check if the servo drive power supply status is normal and if there is driver alarm. 3. Check the status of the servo drive.		

Alarm ID	MOT-089	Alarm Title	Not allow the PLC axis controls the indexing axis
Description	The indexing axis is operated using PLC control component.		
Possible Cause	Use PLC Run or PLC Move to control the indexing axis.		
Solution	Remove the use of PLC Run or PLC Move to control the indexing axis in the ladder diagram.		

Alarm ID	MOT-090	Alarm Title	【Forbid to set as linear axis type when using single turn absolute encoder】
Description	While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0).		
Possible Cause	1. The encoder type is single turn absolute(Pr201 ~ sets as 4). 2. While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0).		
Solution	1. Correct Pr201~ encoder type. 2. While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0). 3. After correction, please reboot the controller and the drive.		

Alarm ID	MOT-901	Alarm Title	Electrical angle detection is in progress, please wait for a few seconds before operating.
Description	Electrical angle detection is in progress, please wait for a few seconds before operating. If Cycle Start (C0) or home request is still triggered during alarm, the command will be executed after electrical angle detection is completed.		
Possible Cause	Electrical angle detection is in progress.		
Solution	After electrical angle detection is completed, you can continue to complete the actions.		

4.1 MOT-001 A,B encoder feedback signal error

Alarm ID	MOT-001	Alarm Title	A,B encoder feedback signal error
Description	axis card detects A/B encoder feedback error.		
Possible Cause	Only happens when Pr9 *Servo board type sets as 0 (EMP2), when axis board type sets as EMP2, axis card will auto-detect A,B encoder feedback. If there is signal error, or has A signal but no B signal, this alarm will be issued.		
Solution	Check the servo cable or replace the axis board.		

4.2 MOT-002 Error counter overflow

Alarm ID	MOT-002	Alarm Title	Error counter overflow
Description	Axis board detects overflow in encoder feedback.		
Possible Cause	Only happen when servo type is set to 0(EMP2), 4(PMC4), 6(SERVO6), axis board will automatically detect the A, B encoder signals, if signal error or input signal too large, counter overflow alarm issued.		
Solution	Check the servo cable or replace the axis board.		

4.3 MOT-003 Encoder module error

Alarm ID	MOT-003	Alarm Title	Encoder module error

Description	Currently no application.
Possible Cause	
Solution	

4.4 MOT-004 No index interrupt signal

Alarm ID	**MOT-004**	**Alarm Title**	**No index interrupt signal**
Description	Currently no application.		
Possible Cause			
Solution			

4.5 MOT-005 DDA command overflow

Alarm ID	**MOT-005**	**Alarm Title**	**DDA command overflow**
Description	Controller sends too many commands. In the one interpolation time interval, software calculates that the number of commands sent is more than 2047 pulses.		
Possible Cause	1. DDA software time setting value is too long. 2. Motion velocity is too fast. 3. Servo resolution is set too high. 4. Backlash compensation or pitch compensation is too large. 5. Feed forward Compensation is enabled before booting.		
Solution	1. Recommend that low interpolation time interval setting (parameter 3203) is not less than 2000. 2. Reduce the feedrate limit (Pr461-Pr480) 3. Reduce the servo resolution setting (driver and CNC Pr61-Pr80) 4. Set appropriate mechanical compensation time constant (parameter 1401~1420). 5. Cancel feed forward compensation function (parameter 581~600). 6. Please contact staff of machinery manufacturer to solve problem.		
More description	In order to achieve the multi-axis coordinated control, SYNTEC's controller uses DDA (Digital Differential Analyzer), Cycle Time of DDA is set by parameter Pr 3203. In one Cycle time of DDA, every axial is allowed to send maximum 2047 pulses. Once exceeding this value, controller will send alarm		

4.6 MOT-006 Driver power supply status error

Alarm ID	MOT-006	Alarm Title	Driver power supply status error
Description	When controller outputs movement command, it detects driver power status error (Not Servo On or not power on). (This alarm is only appropriate for situation when Yaskawa driver blocks A.95x warning).		
Possible Cause	1. Driver's Power-amp voltage isn't stable. 2. Cut off driver power-amp voltage when emergency stop pushed. 3. Driver malfunction.		
Solution	1. Check if the power supply of driver is normal. 2. Check if the power cable of driver is loose or broken. 3. Replace the driver.		

4.7 MOT-007 Driver power-off

Alarm ID	MOT-007	Alarm Title	Driver power-off
Description	Driver input power is cut off.		
Possible Cause	1. power failure. 2. Main power is shut down.		
Solution	Ensure the controller is powered off completely. Confirm the power supply and reboot.		

4.8 MOT-008 Loss Pulse

Alarm ID	MOT-008	Alarm Title	Loss Pulse
Description	1. After sending the last command, start to check whether the difference between the feedback and the command exceeds the loss pulse check window(Pr561~Pr580). If it exceeds, this alarm will be issued. 2. hardware model is not within the supported software version.		

Possible Cause	1. Kinematic occurs obstruction phenomenon. 2. Servo drive occurs unexpected Servo ON / OFF. 3. CPU board send the data to axis board unsuccessfully (CPU board or axis board has problem, the contact between CPU and axis board is not good). 4. Unstable position feedback at idle. 5. The cable that sends command from controller to servo driver has poor quality or is disconnected. 6. Controller doesn't set servo drive alarm check, controller continues to send motion command although the drive is abnormal. 7. Local interference. 8. Improper setting of Loss pulse check window (Pr561~Pr580) 9. 10, EZ series controller installed 9.242, 10.112 and below versions software, or option software is not activated.
Solution	1. Do not shut down the controller when alarm occurs. Please check whether the value of No 8, 9, 10 in diagnose function is zero 2. Check whether the mechanical lubrication system is good. 3. Open the cover of axial to check whether foreign matter blocks the motion of axial. 4. Rotate screw to check whether machine is stuck (loading of driver) 5. Execute servo tuning to suppress the vibration frequency. 6. Check the drive servo-on and the servo-off of power or cable signal 7. If the setting value of No 8, 9, 10 in diagnostic function do not change, please take home search action (don't need to reboot), after that check whether parameters 24, 25, 26, 40, 41, 42 are equal to zero, if the parameters 24, 25, 26 are not equal to zero, the feedback loop has problems 8. If the parameters 40, 41, 42 are not equal to zero, command transmission from controller to the motor has been lost pulse. 9. If all parameters 24, 25, 40, 41, 42 are not zero, then the interference signal is relatively large, specifically in the machining process, the setting value of parameters 8, 9, 10 gradually become large. The reason is the contact point between CPU board and axis card is not and axis board is not good. Try to replace CPU board and axis board. 10. Adjust Loss pulse check window (Pr561~Pr580). Recommended values are as follows; a. linear axis : 100 b. rotary axis : 500 11. Confirm if hardware model is compatible with software version, or if option software Option2 is activated.

More description	Settings of Pr561~Pr580 is the check the range of loss pulse. System Data 8 [X axis following error value] System Data 9 [Y axis following error value] System Data 10 [Z axis following error value] System Data 24 [X axis absolute position feedback value] System Data 25 [Y axis absolute position feedback value] System Data 26 [Z axis absolute position feedback value] System Data 40 [X axis absolute position command value] System Data 41 [Y axis absolute position command value] System Data 42 [Z axis absolute position command value]

4.9 MOT-009 Servo Driver Alarm

Alarm ID	**MOT-009**	**Alarm Title**	**Servo Driver Alarm**
Description	Drive issued alarm.		
Possible Cause	Drive alarm mostly is because of external Causes; i.e. 1. Driver overheat. 2. Encoder wiring error. 3. Internal parameters is set wrong. 4. Servo motor is incompatible. 5. driver malfunction.		
Solution	Follow the troubleshoot steps in driver alarm manual to solve alarm.		

4.10 MOT-010 Servo position command comm. error

Alarm ID	**MOT-010**	**Alarm Title**	**Servo position command comm. error**
Description	Once the communication between Kernel and axis card has errors, software will check whether the queue value in internal IC of axis control is not zero.		
Possible Cause	1. There is only one axis card, but parameter sets two axis cards, and servo axis points to the second axis card. 2. One axis card has errors in case controller has two more axis cards. 3. Two or more axis cards, IRQ11 Jump is plugged. In diagnosis function, number 23 is not equal to 100. 4. Servo board pulse source parameter(Pr11) setting error.		

Solution	1. Check whether the parameter setting Pr11, Pr13 are consistent with the hardware feature. 2. Check axis card jump setting. 3. Change axis card.
More description	Each interpolation time interval, Kernel (core) software will check whether the QUEUE value FLAG is correct. After one FILTER, if it reads the error value, alarm will appear and diagnose function number 68 will be added 1.

4.11 MOT-011 Drive communication error

New Old

Versions: 10.118.40O, 10.118.41O, 10.118.48 and later.

Alarm ID	**MOT–011**	**Alarm title**	**Drive communication error**
Description	At least 2 errors listed below occur for over 10ms totally in drive communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded. If it is M2 communication, add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

New Old

Versions: 10.118.40N, 10.118.41N, 10.118.47 and older.

Alarm ID	**MOT–011**	**Alarm title**	**Drive communication error**
Description	Abnormal driver communication		

Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the driver is loose. 3. The quality of the communication cable is poor, or there is noise interference. 4. The Serial Bus M2 communication cable is an older version of the cable with poor noise resistance ability. 5. The communication time setting of RTEX driver is different from Pr3203. 6. The EtherCAT drive does not have a matching ESI file.
Solution	1. Check whether the driver parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the driver. 3. Check whether the machine is properly grounded. If it is M2 communication, add a terminal resistor to the communication port of the end driver. 4. Check if there is a yellow label sticker written V4 on the M2 communication cable of the serial bus. 5. Set the communication time of RTEX driver to the same as Pr3203. 6. Check if the ESI file is imported correctly. 7. Contact Syntec OEM.

4.12 MOT-012 Driver homing error

Alarm ID	**MOT–012**	**Alarm title**	**Driver homing error**
Description	Serial Bus Driver homing unsuccessfully.		
Possible Reason	Homing method (Pr961) is set incorrectly or driver doesn’t support homing function.		
solution	Check whether homing method is set correctly or driver supports homing function.		

4.13 MOT-013 Axial tuning failed. Reboot controller.

Alarm ID	**MOT-013**	**Alarm Title**	**Axial tuning failed. Reboot controller.**
Description	After one serial bus axis tuning function fails, under un-reboot condition triggers axis JOG, MPG JOG or Homing functions.		
Possible Reason	Serial bus axis tuning fails.		
Solution	Reboot controller.		

4.14 MOT-014 Enabled too many serial axis

Alarm ID	MOT-014	Alarm Title	Enabled too many serial axis
Description	Over system serial bus axis number limit.		
Possible Reason	When interpolation time sets too short or using serial axis are too much and cause it's over limit serial bus axis number in interpolation time. And this alarm is issued.		
Solution	Configure the interpolation time to increase or decrease the number of serial axis used.		

4.15 MOT-016 Absolute type data error. Check the encoder power and reboot the driver

Alarm ID	MOT-016	Alarm Title	Absolute type data error. Check the encoder power and reboot the driver
Description	Detects absolute type encoder is abnormal, and controller has reset. Please reboot driver.		
Possible Reason	1. First time absolute encoder connects to power, do encoder setting. 2. Encoder runs out of power. Position data is cleared. 3. After driver connects to power, encoder battery voltage is too low.		
Solution	1. Check if the battery and connection status of the absolute encoder is proper. 2. Check if the battery voltage is stable and complies with the rating voltage required by the absolute encoder. 3. After the above check is completed, reboot controller and driver.		

4.16 MOT-017 First Positive software limit exceed

Alarm ID	MOT-017	Alarm Title	First Positive software limit exceed
Description	Axial machine coordinate is over positive software stroke limit which is set by Pr2401~Pr2440.		
Possible Cause	Stroke movement of machine table exceeds the set value.		
Solution	1. Reset 2. Move axis in negative direction to leave the software stroke limit protection.		

4.17 MOT-018 First Negative software limit exceed

Alarm ID	MOT-018	Alarm Title	First Negative software limit exceed
Description	Axial machine coordinate is over negative software stroke limit which is set by Pr2401~Pr2440.		
Possible Cause	Stroke movement of machine table exceeds the set value.		
Solution	1. Reset. 2. Move axis in positive direction, to leave the software stroke limit protection.		

4.18 MOT-019 Following error exceed

Alarm ID	MOT-019	Alarm Title	Following error exceed
Description	1. Because of the characteristics of servo, servo motor location, there is no way to respond the command of controller immediately, so a slow phenomenon appears, when this latency is not in allowed range, controller will send out the alarm. 2. Hardware type doesn't support software version.		
Possible Cause	1. Mechanism movement is not smooth. 2. Cable contact is poor. 3. Setting values of acceleration and deceleration time are too small. 4. Servo On /Off Relay is interfered. 5. Internal loop in driver parameter gains too small. (Under 10.116.10 and 10.117.10 or newer version, gain parameter (Kp) in controller and driver will synchronize automatically, so it doesn't need to consider this factor). 6. Encoder resolution and electric gear ratio is wrongly set. 7. Drive or motor malfunction. 8. Encoder or cable between encoder and controller is abnormal. 9. System Data number 23 is not equal to 100. 10. 10 and EZ serial controller installs to 9.242, 10.112 or older version, or option software function is not activated.		
Solution	1. Add lubricating oil to mechanism. 2. Confirm wire connecting is normal. 3. Increase combination or acceleration and deceleration time. 4. When controller under dry run mode, open wiring cabinet to check whether servo on/off relay pulses is abnormal. 5. Increase Inner loop gain of driver. 6. Confirm hardware type and software version is matching, or confirm option software function Option 2 is activated. 7. Contact to Syntec OEM.		

More description	Maximum velocity setting value of G00 and home search is equal to setting parameter divided by Kp. This value multiplied by 2 is the setting range of controller. Reasonable following error: Ferr= speech in command/ setting value of loop gain Alarm allowed values= {max[(velocity of first stage in home search process), velocity G00 of each axis]/Kp}*2 i.e. Feedrate 1000mm/min, loop gain 30, accuracy 1um, Ferr = 1000*1000÷60÷30=555 System Data 32 [X axis reasonable following error] System Data 33 [Y axis reasonable following error] System Data 34 [Z axis reasonable following error]

4.19 MOT-020 Cannot return to control mode when moving

Alarm ID	**MOT-020**	**Alarm Title**	**Cannot return to control mode when moving**
Description	When motor starts to servo on, such as emergency stop released, canceling monitor mode (C31 ~) or serial spindle switching to C-Axis mode, zero speed check for the motor does not pass within 400 ms.		
Possible Cause	1. Instant machine movement by hand during motor servo on procedure. 2. The drive gain setting is poor. Therefore, motor will be trembled during servo on procedure. 3. Zero speed check window(Pr901~) is set too small.		
Solution	1. Avoid man-made movement. 2. Check the drive's position loop gain and speed loop gain setting. 3. Enlarge Pr901~ setting. Note, after removing alarm: • For incremental system, press reset can remove the alarm. • For absolute encoder system, system enters Not Ready state, need to reboot controller to remove the alarm.		

4.20 MOT-021 Must re-homing

Alarm ID	**MOT-021**	**Alarm Title**	**Must re-homing**
Description	When user must re-homing, the controller issue this alarm.		

Possible Cause	1.MOT-020 [Cannot return to control mode when moving] is triggered. 2.MOT-022 [Home position inaccurate] is triggered. 3.Pr41~、Pr121~、Pr161~、Pr221~ is changed.
Solution	1.Please remove alarm MOT-020 [Cannot return to control mode when moving] . Press reset or reboot controller. 2.Please remove alarm MOT-022 [Home position inaccurate]. Press reset or reboot controller. 3.Press reset and re-homing according to current axis setting.

4.21 MOT-022 Home position inaccurate

Alarm ID	**MOT-022**	**Alarm Title**	**Home position inaccurate**
Description	After boot and complete homing for the first time, controller will set servo command (System Data 40~) and machine coordinate (Diagnose variable 72~) to zero. Or set an offset value according to Pr881~. After booting, at the N(N>1) times of searching home, home grid will be compared to the result of the first time searching home, if the error is over 0.1 turn of motor, the controller will send alarm.		
Possible Cause	1. Homing signal of motor is abnormal. 2. Stopper, coupling or bearings is not locked tightly. 3. Home grid function (Pr941~) is not activated. 4. Under serial bus environment, when that axis is also as spindle, Pr881~ and Pr1771~ setting is difference, will cause homing position change after homing.		
Solution	1. Move motor in the same direction and observe whether position counter index changes normally. 2. Check whether the mechanism components are fixed properly. 3. If it's possible cause 4., assuming sixth axis setting is as first spindle, please sets Pr886 and Pr1771 to same value.		

4.22 MOT-023 Fatal following error exceed

Alarm ID	**MOT-023**	**Alarm Title**	**Fatal following error exceed**
Description	Because of the characteristics of servo, servo motor's positioning cannot respond immediately to controller's command, and delay phenomenon will appear, when this delay phenomenon is not in the allowed limit, controller will send alarm.		

Possible Cause	1. Servo motor doesn't receive control due to external force. 2. Parameter of drive - inner loop gain is too small. 3. Parameters of acceleration and deceleration time is set too short. 4. Encoder is abnormal or connecting encoder to controller is abnormal.
Solution	1. Check the external motion of machine table. 2. Check the setting parameter of drive. 3. Check the acceleration and deceleration setting of each axis, Pr401, Pr541~Pr560. 4. Maintain the connection between encoder and servo drives.
More description	Maximum velocity value of G00 and home search is equal to setting parameter divided by Kp. This value multiplied by 4 is setting range of controller. Reasonable following error: Ferr= speech in command/ loop gain Alarm allowed values= {max[(velocity of first stage in home search process), velocity G00 of each axis]/Kp}*4 System Data 32 [X axis reasonable following error] System Data 33 [Y axis reasonable following error] System Data 34 [Z axis reasonable following error]

4.23 MOT-024 Fatal dual feedback error exceed

Alarm ID	**MOT-024**	**Alarm Title**	**Fatal dual feedback error exceed**
Description	If controller discovers that the command and the second command of encoder feedback exceed allowable limit set in Pr3817, controller will send this alarm.		
Possible Cause	1. Mechanism interference. 2. Motor parameter setting error. 3. Linear scale parameter setting error. 4. Servo motor feedback signal abnormal or has interference. 5. Linear scale signal abnormal or has interference. 6. Linear scale reader loose and signal receiving unstable.		
Solution	1. Check external motion mechanism. 2. Check if motor resolution setting Pr61~, Pr81~ and Pr61~ is correct. 3. Check if linear scale resolution setting Pr261~ and Pr301~ is correct. 4. Check if motor encoder function well, or move motor feedback direction away from high power electromagnetic devices. 5. Check if linear scale power is stable, or move linear scale feedback direction away from high power electromagnetic devices. 6. Confirm reader will not be affected by mechanism stroke movement.		

4.24 MOT-025 Positive hardware limit exceed

Alarm ID	MOT-025	Alarm Title	Positive hardware limit exceed
Description	Servo motor touches the positive hardware limit switch in movement.		
Possible Cause	1. Machine table exceeds protection point. 2. Hardware stroke switches are damaged or broken 3. Controller input signal error.		
Solution	1. Use MPG mode to move machine table to opposite direction once discovering that machine table stops at the switch. 2. If machine table is not at the switch, check switch device, limit switch wiring, IO terminal, 24V power supply of terminal. 3. Check whether IO card is abnormal.		

4.25 MOT-026 Negative hardware limit exceed

Alarm ID	MOT-026	Alarm Title	Negative hardware limit exceed
Description	Servo motor touches the negative hardware stroke limit in moving.		
Possible Cause	1. Machine table exceeds protection point. 2. Hardware stroke switches are damaged or broken. 3. Input signal has errors.		
Solution	1. Use MPG mode to move machine table in opposite direction once the machine table stops at the switch. 2. If machine table is not at the switch, check limit switch, switch wiring, IO terminal, 24V power supply of terminal. 3. Check whether IO card is normal.		

4.26 MOT-027 program error in PLC axis

Alarm ID	MOT–027	Alarm Title	program error in PLC axis
Description	PLC axis program syntax error.		
Possible Cause	PLC axis program syntax error.		
Solution	Check syntax of PLC axis program.		

4.27 MOT-028 System memory too low

Alarm ID	MOT-028	Alarm Title	System memory too low
Description	When CNC axis and PLC axis exchange, system remaining memory is too low.		
Possible Cause	During machining process, switch axis into PLC axis.		
Solution	Contact machinery manufacturers.		
Advance Description	Kernel software will check value of system data number 7 'System remaining memory' at any time. When remaining value is too low, this alarm will be issued.		

4.28 MOT-029 Miss index in homing

Alarm ID	MOT-029	Alarm Title	Miss index in homing
Description	When searching home, if motor does not find out motor index signal after leaving home DOG more than 5 pitches, controller will send this alarm.		
Possible Cause	1. Can't read the index signal. 2. The setting of homing 2nd travel feedrate is too fast. 3. The setting of motor reduction ratio is too big. 4. The distance between index signal and HomeDog is more than 5 pitches.		
Solution	1. Check motor index signal wiring; observe system data no. 48(X), 49(Y), 50(Z) to check whether index signal is read. If no, please check whether wiring is normal. 2. Reduce setting value of the homing 2nd travel federate (Pr841~Pr843)		
More description	When searching home, machine will use the velocity setting value of the first stage to move to home DOG, and stop. After that machine moves backward with velocity of the second stage. After leaving home DOG to move backward, it start to search the nearest motor index signal. In the second stage, controller will calculate according to resolution of encoder. If controller leaves home DOG more than 5 pitches and can not find out the index signal. Controller will send alarm.		

4.29 MOT-030 Homing zero speed check failed

Alarm ID	MOT-030	Alarm Title	Homing zero speed check failed
Description	When motor touches HomeDog, motor cannot stop completely.		

Possible Cause	1. Improper drive gain setting, and makes motor vibrating. 2. Resonance phenomenon caused by motor running.
Solution	1. Check the position loop gain and velocity loop gain setting of driver. 2. Activate the resonance frequency inhibition ability of driver. 3. Contact machinery manufacturers for help.
More description	When searching home, machine will use the velocity setting value of the first stage to move to home DOG, and stop once it meets home DOG. After that machine moves backward with velocity of the second stage. After leaving home DOG to move backward, it start to search the nearest motor index signal. At the first stage to find the home DOG, motor will decrease velocity to stop. After 0.1 second command stops, if system data no. 8(X), 9(Y), 10(Z)-error register receives values bigger than zero speed check window(Pr901~Pr920), controller will send alarm.

4.30 MOT-031 Static dual feedback error exceed

Alarm ID	**MOT-031**	**Alarm Title**	**Static dual feedback error exceed**
Description	After the controller stops sending the motion command, during the time set by Pr3805, the system will check whether dual feedback error exceeds allowed limit set by Pr1421~Pr1440. If yes, controller will send alarm.		
Possible Cause	1. Mechanism interference. 2. Motor parameter setting error. 3. Linear scale parameter setting error. 4. Motor feedback signal abnormal or is interfered. 5. Linear scale signal abnormal or is interfered. 6. Linear scale reader is loose and cause the feedback unstable.		
Solution	1. Check the external motion mechanism. 2. Check if motor resolution setting Pr61~, Pr81~ and Pr61~ is correct. 3. Check if the linear scale resolution setting Pr261~ and Pe301~ is correct. 4. Check if motor encoder works well, or move motor feedback line away from high power electromagnetic devices. 5. Check if linear scale power is stable, or move linear scale feedback line away from high power electromagnetic devices. 6. Confirm the reader is not interfered by the mechanism movement.		

4.31 MOT-032 Z axis Following spindle error exceed

Alarm ID	**MOT-032**	**Alarm Title**	**Z axis Following spindle error exceed**

Description	For tracking Tapping feeds, feeding axis opposite direction will feed more than one pitch distance.
Possible Cause	Spindle feedback cable wiring is opposite.
Solution	1. Swap the position feedback A+ and A- which inverter sends to the controller. 2. Change inverter related parameters.

4.32 MOT-033 Absolute encoder read error

Alarm ID	**MOT-033**	**Alarm Title**	**Absolute encoder read error**
Description	When using absolute encoder, the communication between the controller and the driver fails.		
Possible Cause	1. The driver is not powered on. 2. The communication cable between the controller and driver is loose. 3. The absolute adapter board is damaged.		
Solution	1. Check if the power supply of driver is normal. 2. Check if the wiring is correct. 3. Change the absolute adapter board.		

4.33 MOT-034 Absolute home position has not been set. Please reset it.

Alarm ID	**MOT-034**	**Alarm Title**	**Absolute home position has not been set. Please reset it.**
Description	When using absolute encoder, need to set up absolute home position in controller.		
Possible Cause	1. The absolute home position has not been set or it is lost due to insufficient voltage. 2. Pr41~, Pr61~, Pr81~, Pr121~, Pr161~, Pr201~, Pr221~ and etc. parameters is modified. 3. Detect Yaskawa encoder sends abnormal alarm. (For example: A.C80, A.CA0) 4. Execute multi turns reset function on the controller. 5. An abnormal absolute position status of Syntec drive is detected.		
Solution	1. For possible reasons 1, 2, and 4: (Again) sets absolute home position. 2. For possible reason 3, 5: After solving abnormal encoder issue, sets up absolute home position again.		

More description	Absolute home position setting step: • Step 1: Move axis to the pre-set mechanical origin. • Step 2: Enter home position setting page, path: Parameter setting > Serial Parameter > Abosolute home position • Step 3: Choose the axis to be set with direction key, and click 'set machine origin'.

4.34 MOT-035 absolute position battery failure

Alarm ID	**MOT-035**	**Alarm Title**	**absolute position battery failure**
Description	When using absolute encoder, the communication between the controller and driver is successful, but the motor encoder position value is 0.		
Possible Cause	The battery of absolute encoder has no sufficient power.		
Solution	Change the battery.		

4.35 MOT-036 Can't leave home dog switch

Alarm ID	**MOT-036**	**Alarm Title**	**Can't leave home dog switch**
Description	When searching home, after stop and return, if the axis moving exceed Pr981~ setting and still can't leave home dog switch.		
Possible Cause	HomeDog is damaged.		
Solution	Use the electrical multimeter to check whether the sensor of HomeDog is damaged or wiring is short circuit.		
More description	When searching home, machine will use setting value of the 1st phase feedrate to move towards home DOG (Pr881~Pr900 Axial home offset), and stop until reaches home dog switch. In the 2nd phase, controller will calculate according to encoder resolution. If controller leaves home DOG exceed the setting of Pr981~Pr1000 Axis homing 2nd protect revolution (encoder type) and cannot leave index signal, controller will send alarm.		

4.36 MOT-037 Second Positive software limit exceed

Alarm ID	MOT–037	Alarm Title	Second Positive software limit exceed
Description	Axis machine coordinate exceeds Pr2501~Pr2540 set Second Positive software limit value.		
Possible Cause	The motion of machine table exceeds set value.		
Solution	1. Reset to remove alarm. 2. Move axis in negative direction to leave the software stroke protection range.		

4.37 MOT-038 Second Negative software limit exceed

Alarm ID	MOT–038	Alarm Title	Second Negative software limit exceed
Description	Axis machine coordinate exceeds Pr2501~Pr2540 set Second Negative software limit value.		
Possible Cause	The motion of machine table exceeds setting value.		
Solution	1. Reset to remove alarm. 2. Move axis in positive direction to leave the software stroke protection range.		

4.38 MOT-039 Tapping severe over travel

Alarm ID	MOT-039	Alarm Title	Tapping severe over travel
Description	Tracking Tapping theory is Z axis moves along with spindle feedback. If driver occurs an alarm that cause motor dry runs during tapping period, Z axis will still tap with motor feedback, and there might have risk of machine crash. Therefore, when Z axis is higher than the hole top (R point) or about 10 pitches from the hole bottom. System will send this alarm and stop (pause) the machine. Not supported version: 10.114.33E, 10.114.38F, 10.115.43D, 10.115.46A, 10.115.47 or above version.		
Possible Cause	Motor dry runs because of alarm.		
Solution	Check the cause of driver alarm.		

4.39 MOT-040 Dual feedback self-inspection error exceed

Alarm ID	MOT-040	Alarm Title	Dual feedback self-inspection error exceed
Description	After activating dual feedback, between linear scales' each index signals, accumulated A, B pulse count difference exceed Pr3818 setting.		
Possible Cause	1. Linear scale power is unstable. 2. Linear scale feedback is interfered. 3. Linear scale's adapter is interfered by magnetic field or electric field. 4. Linear scale's sensor reader is loose and cause unstable feedback.		
Solution	1. Independently us the power supply of linear scale. 2. Check if the case at CNC side is power pollution free, or the isolation cable is fallen. 3. Keep the linear scale adapter away from the heavy electricity area, or cover it with cooper sheet to reduce external interference. 4. Confirm the sensor reader will not be affected by the mechanism travel movement.		

4.40 MOT-041 Exceed the 3rd positive software travel limit

Alarm ID	MOT-041	Alarm Title	Exceed the 3rd positive software travel limit
Description	Axis mechanical coordinate is over positive software stroke limit which is set by Pr2441~Pr2480.		
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press reset. 2. Move axis in negative direction, to leave the software stroke protection range.		

4.41 MOT-042 Exceed the 3rd negative software travel limit

Alarm ID	MOT-042	Alarm Title	Over the 3rd negative software travel limit
Description	Axis mechanical coordinate is over negative software stroke limit which is set by Pr2441~Pr2480.		
Possible Cause	Machine bed moves over setting limit.		

Alarm ID	MOT-042	Alarm Title	Over the 3rd negative software travel limit
Solution	1. Press reset. 2. Move axis in positive direction, to leave the software stroke protection range.		

4.42 MOT-043 Enter the 1st software stroke limit protection range

Alarm ID	MOT-043	Alarm Title	Enter the 1st software stroke limit protection range
Description	Axis machine coordinate enter software stroke limit's protection range which is set by Pr2401~Pr2440.		
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press reset. 2. Axis move in opposite direction, and leave the software stroke limit protection range.		

4.43 MOT-044 Enter the 2nd software stroke limit protection range

Alarm ID	MOT-044	Alarm Title	Enter the 2nd software stroke limit protection range
Description	Axis mechanical coordinate enter software stroke limit's protection range which is set by Pr2501~Pr2540.		
Possible Cause	Bed moves over setting limit.		
Solution	1. Press the reset. 2. Move the axis in opposite direction, to leave the software stroke protection range.		

4.44 MOT-045 Enter the 3rd software stroke limit protection scope

Alarm ID	MOT-045	Alarm Title	Enter the 3rd software stroke limit protection scope
Description	Axis mechanical coordinate enter software stroke limit's protection range which is set by Pr2441~Pr2480.		

Alarm ID	MOT-045	Alarm Title	Enter the 3rd software stroke limit protection scope
Possible Cause	Machine bed moves over setting limit.		
Solution	1. Press the reset. 2. Move axis in opposite direction, and leave the software stroke protection range.		

4.45 MOT-046 Origin switch signal abnormal

Alarm ID	MOT-046	Alarm Title	Origin switch signal abnormal
Description	For incremental type home searching, set 0 or 2 in home mode (Pr961~), system will move towards origin switch direction according to Pr861~ setting, and stop with deceleration order when finds original point switch. If machine runs over block because deceleration distance is too long, when system continues moving in opposite direction with 2nd phase homing feedrate, it needs to wait for origin switch's signal again shows up and disappear, then it can start finding index signal. When moving in opposite direction, if origin switch's signal doesn't appear, alarm will be issued if exceeds 'five times distance of 1st phase homing feedrate * G00 acceleration and deceleration time/ 2'.		
Possible Cause	1. Origin switch contact is poor. 2. Noise interference.		
Solution	1. Check if the origin switch signals are interfered by noise. 2. Check if the origin switch contact is poor contact.		

4.46 MOT-050 Illegal control mode switch

Alarm ID	MOT-050	Alarm Title	Illegal control mode switch
Description	Do illegal control mode switch to axis.		

Alarm ID	MOT-050	Alarm Title	Illegal control mode switch
Possible Cause	1. Switch the axis control mode from speed control into torque/ gap control. 2. Switch the axis control mode from torque control into speed/ gap/ spindle orientation control. 3. Switch the axis control mode from gap control mode into speed/ torque/ spindle orientation control. 4. Switch the axis control mode from spindle orientation control into torque/ gap control.		
Solution	Switch the axis control mode correctly.		

4.47 MOT-051 Inhibit to cycle start while moving

Alarm ID	MOT–051	Alarm Title	Inhibit to cycle start while moving
Description	Before all manual commands are sent, prohibit cycle start to prevent operation error.		
Possible Cause	Manual command (JOG, INJOG, and MPGJOG) cannot be sent successfully.		
Solution	Reset to remove alarm. Wait until machine stops, then cycle start.		

4.48 MOT-052 Driver power-amp voltage not input

Alarm ID	MOT-052	Alarm Title	Driver power-amp voltage not input
Description	Driver power-amp power is not suppled, please check power cable wiring.		
Possible Cause	Driver power-amp power is not suppled.		
Solution	Check if the three phase of driver electricity is input correctly.		

4.49 MOT-053 No absolute type functions for an increment encoder

Alarm ID	MOT-053	Alarm Title	No absolute type functions for an increment encoder
Description	Driver side's encoder is incremental or when its setting is incremental, controller side can’t start using absolute homing function.		

Alarm ID	MOT-053	Alarm Title	No absolute type functions for an increment encoder
Possible Cause	Encoder is incremental or use as incremental. Sets Pr201~Pr220 as absolute encoder.		
Solution	1. Check the encoder type. In case of a incremental encoder, configure the Pr201~Pr220 as an incremental encoder. 2. Check the driver internal and the related parameter setting of encoder, and configure the encoder in the driver as an increment type. 3. After modification, reboot controller and driver.		

4.50 MOT-054 Encoder type (Pr201~Pr220) changed

Alarm ID	MOT-054	Alarm Title	Encoder type (Pr201~Pr220) changed
Description	Encoder type has changed.		
Possible Cause	Modified Pr201~Pr220 or driver internal and encoder related parameter setting.		
Solution	Reboot controller and driver.		

4.51 MOT-055 Rotation axis type C and double feedback unsupported by the absolute type

Alarm ID	MOT-055	Alarm Title	Rotation axis type C and double feedback unsupported by the absolute type
Description	When using absolute encoder, related parameter setting is wrong.		
Possible Cause	1. When using absolute encoder (Pr201~ sets as 3), axis's axial type stetting will be rotary axial type C (Pr201~ sets as 3). 2. When using absolute encoder (Pr201~ sets as 3) and start dual feedback function simultaneously (Pr241~ not sets as 0).		
Solution	1. When using an absolute encoder, configure the axis type of axial direction as the non-rotation axis type C and other types (Pr221~≠3). 2. Turn off the dual feedback function (Pr241~=0) or use an increment encoder (Pr201~=0).		

Alarm ID	MOT-055	Alarm Title	Rotation axis type C and double feedback unsupported by the absolute type
Advanced Explanation	1. When this alarm occurs will trigger 'MOT-034 first homing setting incomplete, absolute position can't operate', user need to reset home position. 2. This alarm won't be cleared until related parameter reset correctly. Whether need to reboot depend on parameter. a. Pr221~ effective after press reset. b. Pr201~ Pr241~ effective after reboot.		

4.52 MOT-056 Single-turn absolute encoder cannot be used as an increment type. Reconfigure the encoder parameter.

Alarm ID	MOT-056	Alarm Title	Single-turn absolute encoder cannot be used as an increment type. Reconfigure the encoder parameter.
Description	Single-turn absolute encoder can't be used as incremental encoder.		
Possible Cause	When encoder is absolute encoder, set Pr201~Pr220 as incremental encoder.		
Solution	1. Check the encoder type. In case of a single-turn absolute type, configure the Pr201~Pr220 as an absolute encoder. 2. Check the driver internal and encoder related parameter setting, set the use of the encoder in the driver as increment type. 3. After modification, reboot controller and driver.		

4.53 MOT-057 Drive cannot servo on

Alarm ID	MOT-057	Alarm Title	Drive cannot servo on
Description	Drive cannot servo on.		
Possible Cause	When controller send servo on signal, but drive can't enter servo on state in 20 seconds. Controller will send this alarm.		
Solution	1. Please contact the drive supplier. 2. After resolving the problem that the drive cannot servo on, reset the controller to clear the alarm.		

4.54 MOT-058 To permanent save the driver parameter setting, execute under not ready state

Alarm ID	**MOT-058**	**Alarm Title**	**To permanent save the driver parameter setting, execute under not ready state**
Description	Pr181~, Pr1771 has synchronized into driver parameter, but axis is servo on and can't execute save parameters.		
Possible Cause	When modified P1181~ and Pr1771~, axis is servo on.		
Solution	Execute permanent save of driver parameter under not ready state.		

SYNTEC

4.55 MOT-059 Please modified resolution related parameters from the page of serial parameters

Alarm ID	MOT-059	Alarm Title	Please modified resolution related parameters from the page of serial parameters
Description	The axis of the Syntec driver can synchronize the resolution related setting. If the sensor type of controller and driver are the same, the related controller parameters will be synchronized from the driver. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. The matched condition of sensor type (single feedback): • Driver parameter Pn-911 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-911 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 The matched condition of sensor type (driver dual feedback or semi-closed loop): • Driver parameter Pn-931 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-931 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 Axial resolution related parameters: • Pr61~Pr80 Axial encoder resolution • Pr81~Pr100 Axial encoder scaling factor Driver resolution related parameters: • Pn-902 Encoder Resolution • Pn-922 2nd Encoder Resolution • Pn-20E Electronic Gear Ratio (Numerator) • Pn-210 Electronic Gear Ratio (Denominator) Version information: • This alarm is supported in 10.118.41Q, 10.118.48B, 10.118.49 and later versions.		
Possible Cause	Modified the resolution related parameters in controller after the resolution synchronization of the Syntec driver.		
Solution	1. Reset the controller. 2. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. Then restart the controller and the driver.		

4.56 MOT-060 Torque application configuration conflicted with driver configuration (Pn002.0)

Alarm ID	MOT-060	Alarm Title	Torque application configuration conflicted with driver configuration (Pn002.0)
Description	Torque application setting is conflicted with Yaskawa driver parameter setting.		
Possible Cause	1. Controller needs to use torque limit mode, but Yaskawa driver parameter setting sets as torque compensation mode. 2. Controller needs to use torque compensation mode, but Yaskawa driver parameter setting sets as torque limit mode.		
Solution	Check if the driver parameter configuration is correct.		

4.57 MOT-061 Gap control mode unsupported by axis

Alarm ID	MOT-061	Alarm Title	Gap control mode unsupported by axis
Description	Axis doesn't support switching into gap control mode.		
Possible Cause	1. Only Syntec driver supports gap control mode. 2. Syntec driver's firmware is too old.		
Solution	1. Use Syntec driver. 2. Check the firmware version of the Syntec driver.		

4.58 MOT-062 Gap control mode cannot be switched during axial movement

Alarm ID	MOT-062	Alarm Title	Gap control mode cannot be switched during axial movement
Description	Axis is moving, can't change from position control into gap control.		
Possible Cause	When axis is moving, received the mode changing command.		
Solution	Mode can be switched after the axial movement stops completely.		

4.59 MOT-063 Axial sensor type not matching with home searching method

Alarm ID	MOT-063	Alarm Title	Axial sensor type not matching with home searching method
Description	Axis sensor type isn't 'Absolute encoder', but homing method setting is 'Absolute encoder index' or 'Absolute with origin switch', and the parameter is not matched.		
Possible Cause	Axis sensor type (Pr201~Pr220) sets as 0~2, and the corresponding home searching method (Pr961~Pr980) sets as 4 or 5.		
Solution	In case the axial sensor type (Pr201~Pr220) is configured to 0~2, the home searching method (Pr961~Pr980) can be configured to 0~3, not 4 or 5.		

4.60 MOT-064 Axis driver doesn't support selected friction compensation

Alarm ID	MOT-064	Alarm Title	Axis driver doesn't support selected friction compensation
Description	That axis driver doesn't support the selected friction compensation.		
Possible Cause	When Pr2921~ is set to 1, but driver does not support speed spike compensation, the alarm will be issued; when Pr2921~ is set to 2, but driver does not support torque spike compensation, the alarm will be issued.		
Solution	1. Set Pr2921~ as the friction compensation that the axis driver supports. Press reset can remove this alarm. 2. Drivers that support speed spike compensation include: a. Yaskawa M2/M3 serial bus driver (Controller support versions: 10.116.54K, 10.118.0F, 10.118.6 and later versions). b. Syntec M3 serial bus driver (Controller support versions: 10.118.12G, 10.118.16 and later versions, driver support versions: 2.8.3, 2.9.0, 2.10.0 and later versions). c. Panasonic RTEX serial bus A6N driver (Controller support versions: 10.118.21 and 10.118.12O and later versions). d. EtherCAT serial bus drivers that support speed spike compensation (Controller support versions: 10.118.48I, 10.118.52C, 10.118.54 and later versions), please refer to EtherCAT Drive Application Manual. 3. Drivers that support torque spike compensation include: a. Syntec M3 serial bus driver (Controller support versions: 10.118.40A, 10.118.41, and later versions, driver supported versions: 2.14.105, 2.15.0, 3.0 .0 and later versions). 4. RETX related regulation, please refer to RTEX Driver Application Manual.		

4.61 MOT-065 Axis driver doesn’t support power-off retract

Alarm ID	MOT-065	Alarm Title	Axis driver doesn’t support power-off retract
Description	Driver doesn’t support power-off retract function.		
Possible Cause	Driver doesn’t support power-off retract function. When Pr1041~ Pr1060 setting is not 0 and this alarm will be issued.		
Solution	1. Only Syntec Bus M3 Driver supports power-off retract function. 2. Confirm Syntec Serial Bus M3 Driver’s firmware version. Support firmware versions are 2.10.1, 2.11.0 and above version.		

4.62 MOT-066 Axis driver firmware version is not supported

Alarm ID	MOT-066	Alarm Title	Axis driver firmware version is not supported
Description	Controller doesn’t support axis driver's firmware version.		
Possible Cause	Driver’s firmware version is too old.		
Solution	Upgrade driver firmware version. For Panasonic RETX A6N firmware version, upgrade to 1.23 or above version.		

4.63 MOT-067 Syntec encoder’s firmware upgrade finished. Please reboot driver.

Alarm ID	MOT-067	Alarm Title	Syntec encoder’s firmware upgrade finished. Please reboot driver.
Description	Syntec encoder's firmware upgrade is finished. Need to reboot driver.		
Possible Cause	Execute axis encoder's firmware upgrade.		
Solution	Reboot driver and controller.		

4.64 MOT-068 Syntec encoder information reading timeout

Alarm ID	MOT-068	Alarm Title	Syntec encoder information reading timeout
Description	Syntec encoder information reading timeout.		
Possible Cause	1. Communication between controller and driver is abnormal. 2. Driver condition is abnormal.		
Solution	1. Check if communication is loose. 2. Reboot driver.		

4.65 MOT-069 Axial feedback abnormal

Alarm ID	MOT-069	Alarm Title	Axial feedback abnormal
Description	If the difference between the axial feedback value exceeds 4 times of the maximum speed set by the controller, the number of abnormal times will be accumulated by 1 each time. This alarm will be issued when two consecutive abnormalities occur; The maximum speed is the maximum value among Pr461 ~, Pr521 ~, Pr621 ~, Pr821 ~, Pr1801 ~(when using as servo spindle).		
Possible Cause	1. Communication between controller and driver is abnormal. 2. The following parameter settings are inappropriate; a. Pr461~Pr480 Maximum feed rate for axial rapid movement (G00) b. Pr521~Pr540 Axial JOG speed c. Pr621~Pr640 Axial maximum cutting feedrate d. Pr821~Pr840 Axial homing feedrate 3. If using servo spindle, Pr1801~Pr1810 Spindle maximum speed setting is inappropriate		
Solution	1. Check if communication is connect well. 2. Re-set above mentioned parameters.		

4.66 MOT-070 Axial command abnormal

Alarm ID	MOT-070	Alarm Title	Axial command abnormal
Description	The planned controller command exceeds 1.2 times of the distance that can be moved at the axial maximum speed; The maximum speed is the maximum value among Pr461~, Pr521~, Pr621~, Pr821~, Pr1801~ (when using as servo spindle).		

Alarm ID	MOT-070	Alarm Title	Axial command abnormal
Possible Cause	1. The following parameter settings are inappropriate; a. Pr461~Pr480 Maximum feed rate for axial rapid movement (G00) b. Pr521~Pr540 Axial JOG speed c. Pr621~Pr640 Axial maximum cutting feedrate d. Pr821~Pr840 Axial homing feedrate 2. If using servo spindle, Pr1801~Pr1810 Spindle maximum speed setting is inappropriate		
Solution	Re-set above mentioned parameters.		

4.67 MOT-071 Interpolation severely overtime

Alarm ID	MOT-071	Alarm Title	Interpolation severely overtime
Description	The interpolation timeout exceeds 10ms, causing controller fail to communicate with driver normally; If within 10ms, controller will perform proxy interpolation and continue to communicate with driver.		
Possible Cause	1. Frequent data transmission in the cloud, causes the controller system to be extremely busy, e.g. using dipole structure (the front stage simulator is connected to the back stage controller) 2. Parameter Pr3251 is set to 5.		
Solution	1. Reboot controller and driver. 2. .Check if Pr3251 is set to 5. If so, please take a look at "Pr3251 Touch Screen" page, and set Pr3251 to other value. 3. Please upgrade the BSP version to 9.35 or above. 4. Please contact OEM Syntec.		

4.68 MOT-072 Software stroke limit setting error when using single turn absolute encoder

Alarm ID	MOT-072	Alarm Title	Software stroke limit setting error when using single turn absolute encoder
Description	Pr2401~Pr2440 is not activated or is incorrectly set when using the single turn absolute encoder type.		

Alarm ID	MOT-072	Alarm Title	Software stroke limit setting error when using single turn absolute encoder
Possible Cause	1. The encoder type is single turn absolute. 2. When using the single turn absolute encoder type, the software stroke limit should be activated and its value must be set within the distance in one motor revolution.		
Solution	1. Correct Pr201~ encoder type. 2. When Pr201~ is single turn absolute encoder type, make sure that Pr2401~Pr2440 are not default values, i.e. the negative and positive limits are not -999999999 and 999999999 respectively. And the positive limit should be larger than the negative limit. Also, the stroke limit should be set within the distance in one motor revolution. 3. After correction, please reboot the controller and the drive.		

4.69 MOT-073 Fail to preload servo drive parameter

Alarm ID	MOT-073	Alarm Title	Fail to preload servo drive parameter
Description	The controller fails to preload necessary parameters from the drive during booting.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. Mismatch between the controller and the drive version.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please upgrade the controller version. 5. Please upgrade the drive version. 6. Please contact OEM Syntec.		

4.70 MOT-074 Fail to read absolute position from servo drive

Alarm ID	MOT-074	Alarm Title	Fail to read absolute position from servo drive
Description	The controller fails to read the absolute position from the drive.		
Possible Cause	1. The controller can't communicate with the drive normally.		

Alarm ID	MOT-074	Alarm Title	Fail to read absolute position from servo drive
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Please contact OEM Syntec.		

4.71 MOT-075 Serial communication initialize timeout

Alarm ID	MOT-075	Alarm Title	Serial communication initialize timeout
Description	The communication between the controller and the drive is established, but fails to complete the initialization process in 20 seconds.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. The drive issues critical alarms.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please lower the interpolation time (Pr3203), then reboot the controller and the drive. 5. Please contact OEM Syntec.		

4.72 MOT-076 Servo drive parameter preload timeout

Alarm ID	MOT-076	Alarm Title	Servo drive parameter preload timeout
Description	The communication between the controller and the drive is established, but fails to preload parameters in 20 seconds.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. Take too much time to read drive parameters.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot all critical drive alarms. 4. Please lower the interpolation time (Pr3203) and reboot the controller and the drive. 5. Please lower the PLC scan time (Pr3204) and reboot the controller and the drive. 6. Please contact OEM Syntec.		

4.73 MOT-077 Fail to recover machine position

Alarm ID	MOT-077	Alarm Title	Fail to recover machine position
Description	Fail to recover machine position from absolute encoder or Fram in 20 seconds during booting.		
Possible Cause	1. The controller can't communicate with the drive normally. 2. The drive issues absolute encoder battery alarms.		
Solution	1. Check if the axis is connected to a third-party drive. If it is, check if the drive causes the alarm. 2. Reboot the controller and the drive. 3. Troubleshoot drive's absolute encoder battery alarms. 4. Please contact OEM Syntec.		

4.74 MOT-079 Inconsistent Resolution Settings

Alarm ID	MOT-079	Alarm 标题	【Inconsistent Resolution Settings】
说明	The resolutions of the controller and the drive are not set consistently. The settings are considered consistent if the following equation is satisfied: 1. Single feedback: Resolution (Pr61~) * Encoder Scaling Factor (Pr81~) = Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E). 2. Dual feedback or Semi-Closed Loop: Resolution (Pr61~) * Encoder Scaling Factor (Pr81~) = 2nd Encoder Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E).		
可能原因	1. Improper controller parameter settings: a. Encoder Resolution. b. Encoder Scaling Factor. 2. Improper drive parameter settings: a. Electronic Gear Ratio. b. Encoder Resolution.		
排除方法	1. Change the controller parameters so that the equation mentioned above is satisfied: a. Axes: Pr61~, Pr81~. b. Servo Spindles or Serial Rotary Spindles: Pr1651~, Pr1661~. 2. Change the drive parameters so that the equation mentioned above is satisfied: a. Electronic Gear Ratio: Pn-20E, Pn-210. b. Encoder Resolution: Pn-902, Pn-922. 3. To clear the alarm, a. Reset the controller if ONLY the controller parameter(s) is(are) modified. b. Restart both of the controller and the drive if the drive parameter(s) is(are) modified.		

4.75 MOT-080 Poor contact of communication wire

Alarm ID	MOT-080	Alarm title	Poor contact of communication wire
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

4.76 MOT-081 The hardware doesn't receive communication packet

Alarm ID	MOT–081	Alarm title	The hardware doesn't receive communication packet
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

4.77 MOT-082 The communication packet is disturbed by noise

Alarm ID	MOT–082	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		

Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.

4.78 MOT-083 Driver software misses communication packet

Alarm ID	**MOT-083**	**Alarm title**	**Driver software misses communication packet**
Description	Response packet Watchdog(M3) or Working Counter(EtherCAT) is wrong for over 10ms in M3 communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet Watchdog(M3) or Working Counter(EtherCAT) is wrong		
Solution	Contact Syntec OEM.		

4.79 MOT-084 Friction compensation mode of drive and controller are discrepant

Alarm ID	**MOT-084**	**Alarm title**	**Friction compensation mode of drive and controller are discrepant**
Description	The friction compensation of the controller (based on Pr2921~) conflicts with the one of the driver.		
Possible Reason	This alarm happens if any of the following occurs while Syntec driver is connected: 1. The friction compensation of the controller is selected as "Torque compensation" (Pr2921~ is set as 2) while the friction compensation of Syntec driver is deactivated (Pn290 is set as 0). 2. The friction compensation of the controller is NOT selected as "Torque compensation" (Pr2921~ is other than 2) while the friction compensation of Syntec driver is activated (Pn290 is set as 1).		
Solution	1. Press RESET to clear this alarm. This makes the controller re-activate the compensation mode of Pr2921~ and activate/ deactivate the friction compensation (Pn290) of Syntec driver accordingly. 2. Contact Syntec OEM.		

4.80 MOT-085 Trigger PLC MoveAx component for the roaming axis in borrowing

Alarm ID	MOT-085	Alarm title	Trigger PLC MoveAx components for the roaming axis in borrowing
Description	When the PLC MoveAx component is triggered, the axis cannot be borrowed by other axis groups or PLC Rn subprogram components.		
Reason	When Pr742 is set to 1, for the roaming axis that has been borrowed by other axis groups or PLC Rn subprogram components, trigger the PLC MoveAx component to move.		
Solution	After confirming that other axis groups or PLC Rn subprogram components have removed the roaming axis, trigger the PLC MoveAx component to move.		

4.81 MOT-086 Drive is abnormally switch to servo off state when running

Alarm ID	MOT-086	Alarm Title	Drive is abnormally switch to servo off state when running
Description	Drive is abnormally switch to servo off state when running.		
Possible Cause	When drive is abnormally changed from servo on state to servo off state, controller will send this alarm.		
Solution	1. Check if the drive power supply is normal. 2. Check if the wiring is correct. 3. After resolving the problem that the drive cannot servo on, reset the controller to clear the alarm.		

4.82 MOT-087 Inaccurate stop position. Please check if there are any mechanisms are stuck

Alarm ID	MOT-087	Alarm Title	Inaccurate stop position. Please check if there are any mechanisms stuck
Description	After sending the last command, start to check whether the difference between the feedback and the command exceeds the loss pulse check window(Pr561~Pr580). If it exceeds, this alarm will be issued		

Possible Cause	1. Mechanical movement was hindered. 2. Unstable position feedback at idle. 3. Improper setting of Loss pulse check window (Pr561~Pr580)
Solution	1. Check whether the mechanical lubrication system and track lubrication are good. 2. Open the cover of axial to check if any foreign matter blocks the motion of axial. 3. Rotate screw to check if the mechanism is stuck (loading of driver) 4. Use the automatic tuning function to adjust the servo and suppress the vibration frequency. 5. Adjust Loss pulse check window (Pr561~Pr580). Recommended values are as follows: a. linear axis : 100 b. rotary axis : 500

4.83 MOT-088 Servo on failed. Unable to execute operation.

Alarm ID	**MOT-088**	**Alarm Title**	**Servo on failed. Unable to execute operation.**
Description	The servo can not servo on, after cycle start or other manual function is triggered.		
Possible Cause	1. When power on, axis doesn't complete magnetic pole angle detection which cause servo disabled. 2. Driver alarms or abnormal power supplication will cause servo driver become servo off. 3. Driver is malfunction.		
Solution	1. After power on, wait for a while before operation. 2. Check if the servo drive power supply status is normal and if there is driver alarm. 3. Check the status of the servo drive.		

4.84 MOT-089 Not allow the PLC axis controls the indexing axis

Alarm ID	**MOT-089**	**Alarm Title**	**Not allow the PLC axis controls the indexing axis**
Description	The indexing axis is operated using PLC control component.		
Possible Cause	Use PLC Run or PLC Move to control the indexing axis.		
Solution	Remove the use of PLC Run or PLC Move to control the indexing axis in the ladder diagram.		

4.85 MOT-090 Forbid to set as linear axis type when using single turn absolute encoder

Alarm ID	**MOT-090**	**Alarm Title**	**【Forbid to set as linear axis type when using single turn absolute encoder】**
Description	While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0).		
Possible Cause	1. The encoder type is single turn absolute(Pr201 ~ sets as 4). 2. While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0).		
Solution	1. Correct Pr201~ encoder type. 2. While using single turn absolute encoder, axis type is prohibited to set as linear axis type(Pr221 ~ sets as 0). 3. After correction, please reboot the controller and the drive.		

4.86 MOT-901 Electrical angle detection is in progress, please wait for a few seconds before operating.

Alarm ID	**MOT-901**	**Alarm Title**	**Electrical angle detection is in progress, please wait for a few seconds before operating.**
Description	Electrical angle detection is in progress, please wait for a few seconds before operating. If Cycle Start (C0) or home request is still triggered during alarm, the command will be executed after electrical angle detection is completed.		
Possible Cause	Electrical angle detection is in progress.		
Solution	After electrical angle detection is completed, you can continue to complete the actions.		

5 Spindle Alarm - SPD

Alarm ID	**SPD–001**	**Alarm Title**	**A, B Encoder Feedback Signal Error**
Description	Axial card detects A, B encoder has feedback signal error.		
Possible Cause	Only happens when Pr9 parameter axis board type sets as 0 (EMP2). When axis board type sets as EMP2, axis board internally will automatically detects A, B encoder signal. If the signal is wrong, or there is a phase A signal but no phase B signal, then this alarm is issued.		
Solution	Check servo cable or change axis board.		

Alarm ID	**SPD–002**	**Alarm Title**	**Error counter overflows**
Description	Axis card detects encoder feedback overflows.		
Possible Cause	Only happen when Pr9 parameter axis card type sets as 0(EMP2), 4(PMC4), and 6(SERVO6). When axis card type sets as 0(EMP2), 4(PMC4), and 6(SERVO6), axis card inside will detects A, B encoder signal. If signal is wrong or input signal is too big, and this alarm will be issued.		
Solution	Check servo cable or change axis card.		

Alarm ID	**SPD–003**	**Alarm Title**	**Encoder module error**
Description	Currently no application.		
Possible Cause			
Solution			

Alarm ID	**SPD–004**	**Alarm Title**	**NO interrupted index signal**
Description	No application currently.		
Possible Cause			
Solution			

Alarm ID	SPD–005	Alarm Title	Pulse signal is over rated value
Description	The command count send from controller is too much. Software calculate that over 2047 pluses in a interpolation time needs to be send.		
Possible Cause	1. Software interpolation time sets too long. 2. Movement speed is too fast. 3. Servo resolution sets too high. 4. Backlash compensation value or pitch compensation value sets too large. 5. Start feedforward compensation function.		
Solution	1. Decrease software interpolation time setting (Pr3203), nott recommend to set lower than 2000. 2. Lower movement speed limit setting. (Pr461~Pr480) 3. Reduce servo resolution. (Driver and Pr61~Pr80) 4. Sets appropriate mechanism compensation time constant. (Pr1401~Pr1420) 5. Reduce or close feedforward compensation function. (Pr581~Pr600) 6. Please contact machine manufacturer.		

Alarm ID	SPD–006	Alarm Title	Can’t clear index condition
Description	No application currently.		
Possible Cause			
Solution			

Alarm ID	SPD–009	Alarm Title	Driver Alarm
Description	Driver sends an alarm signal.		
Possible Cause	Driver alarm is mostly because of external reasons: For example: 1. Driver temperature is too high. 2. Encoder cable wiring is wrong. 3. Internal parameter setting is wrong. 4. Not matched with servo motor. 5. Driver malfunction.		
Solution	Please follow driver alarm manual for troubleshooting.		

Alarm ID	SPD–017	Alarm Title	Spindle Orientation Position check error
Description	Spindle orientation (C61) triggered and spindle is stopped. Spindle orientation error is over one degree.		
Possible Cause	1. During spindle orientation, inverter module position control mode is not activated. 2. Bad spindle motor tuning and caused the motor stills under jitter correction state after finish orientation. 3. Motor feedback signal lose position command. 4. Spindle zero speed check window (Pr901~Pr916) sets too large.		
Solution	1. Confirm inverter module's control mode. 2. Retune the spindle motor. 3. Confirm motor feedback cable's quality, or contact motor supplier to check if the encoder is abnormal. 4. Reduce zero check window.		
Advanced Explanation	Observe System Data No. 52 when motor operate in the same direction, if the variation will increase or decrease by a multiple of Pr1651*Pr1661? If it's not, it means motor feedback signal lose position command. Please increase the quality of motor feedback cable, or contact motor supplier to check if encoder is abnormal.		

Alarm ID	SPD–018	Alarm Title	Can't find spindle orientation index signal
Description	During spindle orientation, can't find index signal over a certain time.		
Possible Cause	1. If system data no. 52 doesn't change with rotation of motor, it means controller can't read motor index signal. 2. If system data no. 52 change with the rotation of motor, it means controller can read motor index signal, but because spindle rotation speed is not reduced to half of the rated speed, so it can't execute orientation.		
Solution	1. Please increase quality of motor feedback cable, or contact motor supplier to check if encoder is abnormal. 2. Please check if belt is loose or gear setting ratio is correct.		

Alarm ID	SPD-020	Alarm Title	Spindle can't take correct index signal as reference
Description	Spindle can't use hardware and software's index signal.		

Alarm ID	SPD-020	Alarm Title	Spindle can't take correct index signal as reference
Possible Cause	1. If encoder sets on the motor side (Pr1811 sets as 1). When gear number of motor (Pr1681~): gear number of ballscrew (Pr1682~) is not integer ratio, it can't correspond all motor position to all screw position, so the index signal may be inaccurate. If encoder is on the motor side (Pr1811 sets as 1). When axis type (Pr221~) sets as 3 or 5, because the mechanical coordinate is not between 0~360, so it can't calculate the index signal position from feedback. 2. When the spindle rotates more than one revolution, a. If the diagnosis variable 52 has changed, check whether the Pr1993 is set too small. b. If the diagnostic variable 52 remains unchanged, check whether i. the encoder signal cable is not connected properly. ii. axis board pin is fault. iii. encoder Z phase signal is fault. 3. Because the index signal returned by the drive is unreasonable, the controller cannot execute the threading function.		
Solution	1. The controller doesn't support threading or homing function under this condition, please use other rotation type axial. 2. Refer to the following: a. Enlargement Pr1993 setting value. b. Or i. Connect the encoder signal cable securely. ii. Replace the axis board. iii. Replace the encoder Z phase signal. 3. Please contact the drive vendor.		

Alarm ID	SPD–021	Alarm Title	Do tapping when neutral
Description	When spindle switch to neutral, and execute tapping command.		
Possible Cause	Spindle doesn't change to the right gear position.		
Solution	Change spindle's gear (R24) to 1~3 gear, then execute tapping command.		

Alarm ID	SPD–022	Alarm Title	Pr1791~ sets as 4. Only support Syntec Inverter Module
Description	When Pr1791~ sets as 4, it has to be used only with Syntec Spindle Motor Module.		
Possible Cause	Use Yaskawa or other brand's serial bus driver as spindle.		
Solution	Change Pr1791~ into 1 or 3.		

<table>
<tr><th>Alarm ID</th><th>SPD–023</th><th>Alarm Title</th><th>Spindle rotation speed hasn’t reached yet. Please check parameter setting.</th></tr>
<tr><td>Description</td><td colspan="3">
After enables spindle rotation speed reach function (Pr1991 not sets as 0), spindle rotation speed has to reach to user’s setting spindle speed during the check time, or this alarm will be issued.

(Spindle rotation speed reach check time= Max(30, 3 * Pr1831 * 1000), it means take the maximum value between 30 seconds and 3 times of spindle acceleration/ deceleration time.)

If you want to use Pr1991 spindle rotation speed reach check function, even if it is inverter spindle, it also needs to trigger spindle clockwise/counter-clockwise rotation signal C64/C65, R583.x/R584.x to turn the system spindle into rotation condition.
Adjust inverter spindle’s rotation speed reach check regulation to be the same with servo spindle. When Pr1991 is not equals to 0, spindle speed reach conditions are as following:
<table>
<tr><th>Spindle clockwise / counter-clockwise rotation signal C64/ C65, R583.x/R584.x</th><th>Established conditions of spindle rotation reach</th></tr>
<tr><td rowspan="2">On</td><td>The speed difference between spindle feedback (Fbk) and speed command (Cmd) is smaller than speed command (Cmd) times spindle speed reach check range’s percentage. (Pr1992 or Pr1993)</td></tr>
<tr><td>ie | Fbk - Cmd | < Cmd * (Pr1992 or Pr1993) * 0.01</td></tr>
<tr><td rowspan="2">Off</td><td>Spindle feedback (Fbk) is smaller than 50RPM.</td></tr>
<tr><td>i.e. Fbk < 50 (RPM)</td></tr>
</table>

Note: If spindle clockwise/counter-clockwise rotation signal is off it will also check, in order to process when spindle clockwise/counter-clockwise rotation signal status turn from on to off, have to ensure spindle speed has reduce to certain low speed, and will be identified as spindle speed reached.
When spindle doesn’t install encoder (Pr1711~ sets as 0), spindle speed reach conditions are as following:
<table>
<tr><th>Spindle Type</th><th>Established conditions of spindle rotation reach</th></tr>
<tr><td>Serial Bus Spindle</td><td>Directly shows driver’s information if spindle speed has reached or not.</td></tr>
</table>

</td></tr>
</table>

<table>
<tr><td>Alarm ID</td><td>SPD–023</td><td>Alarm Title</td><td>Spindle rotation speed hasn’t reached yet. Please check parameter setting.</td></tr>
<tr><td></td><td colspan="3"><table>
<tr><th>Spindle Type</th><th>Established conditions of spindle rotation reach</th></tr>
<tr><td>General Pulse Spindle</td><td>When S59 bit sets as on, controller will consider that spindle speed has reached.
If you want to do spindle rotation speed reach check, send an O bit to TB16IN from driver, and PLC can gets this information and execute speed reach check in PLC.</td></tr>
</table></td></tr>
<tr><td>Possible Cause</td><td colspan="3">When spindle speed check function enables, spindle speed can’t reach to target speed during rotation speed check’s set time.
(Spindle rotation speed reach check time= Max(30, 3 * Pr1831 * 1000), it means take the maximum value between 30 seconds and 3 times of spindle acceleration/ deceleration time.)</td></tr>
<tr><td>Solution</td><td colspan="3">1. Please confirm spindle rotation speed check function’s related parameter setting is correct. (Pr1991, Pr1992, Pr1993)
2. Please confirm spindle acceleration/ deceleration speed setting is correct. (Pr1831~)
3. Please confirm spindle clockwise / counter-clockwise rotation signal C64/C65, R583.x/ R584.x triggered normally.
4. Please confirm spindle works well.
5. Please confirm spindle feedback works well.</td></tr>
<tr><td>Alarm ID</td><td>SPD–024</td><td>Alarm Title</td><td>Fail to do spindle zero speed check</td></tr>
<tr><td>Description</td><td colspan="3">This alarm will be triggered while a spindle fails to pass zero speed check over 5 seconds and the feedback speed is not decreasing. The following situations require zero speed check:
1. For a serial spindles with no servo command, the spindle will enter servo-off state after passing zero speed check.
2. When exiting the orientation mode, a spindle will do coordinate synchronization after passing zero speed check.
3. Under circumstances of emergency stop (while C36 is on) or critical alarms, a spindle will enter servo-off state after passing zero speed check.
Note: The zero speed check window is 3 rpm if the emergency stop or a critical alarm is triggered, otherwise it is based on Pr901~Pr920 Axis zero speed check window.</td></tr>
<tr><td>Possible Cause</td><td colspan="3">1. Insufficient control performance of spindle motor.
2. Abnormity of spindle motor feedback.
3. Pr901~Pr920 Axis zero speed check window is too narrow. (For Serial Bus Line especially)</td></tr>
</table>

Alarm ID	SPD–024	Alarm Title	Fail to do spindle zero speed check
Solution	1. Please re-tune servo drive. 2. Check whether the motor feedback of the spindle is functioned well. 3. Increase Pr901~Pr920 Axis zero speed check window, a setting value within 0.1 to 0.5 degree is recommended. (The suggested setting value = (0.1~0.5) * spindle resolution/ 360)		
Advanced Explanation	Check the fluctuation of speed and loading rate of the spindle when it is under constant rotating command. A speed fluctuation beyond Pr901~Pr920 Axis zero speed check window -normally led by improper servo tuning- would make it fail to pass zero speed check.		

Alarm ID	SPD–025	Alarm Title	**Please modified resolution related parameters from the page of serial parameters**
Description	The axis of the Syntec driver can synchronize the resolution related setting. If the sensor type of controller and driver are the same, the related controller parameters will be synchronized from the driver. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. The matched condition of sensor type (single feedback): • Driver parameter Pn-911 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-911 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 The matched condition of sensor type (driver dual feedback or semi-closed loop): • Driver parameter Pn-931 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-931 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 Axial resolution related parameters: • Pr61~Pr80 Axial encoder resolution • Pr81~Pr100 Axial encoder scaling factor Driver resolution related parameters: • Pn-902 Encoder Resolution • Pn-922 2nd Encoder Resolution • Pn-20E Electronic Gear Ratio (Numerator) • Pn-210 Electronic Gear Ratio (Denominator) Version information: • This alarm is supported in 10.118.41Q, 10.118.48B, 10.118.49 and later versions.		
Possible Cause	Modified the resolution related parameters in controller after the resolution synchronization of the Syntec driver.		

Alarm ID	SPD–025	Alarm Title	Please modified resolution related parameters from the page of serial parameters
Solution	1. Reset the controller. 2. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. Then restart the controller and the driver.		
Alarm ID	SPD–026	Alarm Title	Spindle orientation offset. Please fix this parameter in Syntec spindle into zero.
Description	If the spindle is Syntec serial bus spindle, then the modification of spindle orientation offset parameter (Pr1751~Pr1756) is invalid.		
Possible Cause	When controller detects that the spindle is Syntec serial bus, corresponding spindle orientation offset parameter (Pr1751~Pr1756) set value should be non-zero.		
Solution	Sets corresponding spindle orientation offset parameter as zero. Press to reset to remove this alarm.		
Alarm ID	SPD–027	Alarm Title	This spindle type doesn't support spindle orientation function
Description	This spindle type doesn't support spindle orientation function.		
Possible Cause	This spindle type doesn't support spindle orientation function.		
Solution	Set Pr1791~ as other spindle types (0~5), which support spindle orientation function.		
Alarm ID	SPD–028	Alarm Title	This spindle type doesn't support spindle tapping function
Description	This spindle type doesn't support spindle tapping function.		
Possible Cause	This spindle type doesn't support spindle tapping function.		
Solution	Set Pr1791~ as other spindle types (0~5), which support spindle tapping function.		
Alarm ID	SPD–029	Alarm Title	This spindle type doesn't support C axis mode
Description	This spindle type doesn't support C axis mode.		

Alarm ID	SPD–029	Alarm Title	This spindle type doesn't support C axis mode
Possible Cause	This spindle type doesn't support C axis mode.		
Solution	Set Pr1791~ as servo spindle types 1~4, which support C axis mode.		

Alarm ID	SPD–030	Alarm Title	This product doesn't support Syntec Spindle
Description	This product doesn't support Syntec Spindle.		
Possible Cause	This spindle of this controller product doesn't support Syntec driver.		
Solution	1. Please change spindle into other non-Syntec driver. 2. Please contact sales personnel to buy other types of controller.		

Alarm ID	SPD–031	Alarm Title	The spindle does not support the dual winding switching function.
Description	This spindle type or drive type does not support the dual winding switching function.		
Possible Cause	1. This spindle does not support the winding switching function. 2. The driver does not support the winding switching function.		
Solution	1. Set R631 and R632 to zero. 2. The spindle uses a Syntec driver. 3. Set Pr1791~ to 2 and 4. 4. Update the drive to version 2.14.0 or above.		

Alarm ID	SPD–032	Alarm Title	dual winding switch failed
Description	An abnormality occurred during the dual winding switching process.		
Possible Cause	1. The state of R631.x is changed during the winding switching. 2. When R632.x is updating, its status is different from R602.x. 3. The winding switching process takes more than five seconds. 4. The drive does not turn on the dual coil parameters switching function.		

Alarm ID	SPD–032	Alarm Title	dual winding switch failed
Solution	1. Switch R631.x to the low speed state. This requirement is removed after (included) software version 10.118.85. 2. Switch R632.x to the state to which R602.x belongs. 3. When the above conditions are met, after the spindle stops, press reset to remove the alarm.		

Alarm ID	SPD–033	Alarm Title	Y-Delta zero speed start check failed
Description	The controller will start the spindle in the high speed coil state. This alarm is removed after (included) software version 10.118.85.		
Possible Cause	1. R631.x is in the high speed state when servo-on. 2. Both R631.x and R602.x are in the low speed state when servo-on, but R632.x is in the high speed state.		
Solution	1. Set R631.x to the low speed state. 2. When both R631.x and R602.x are in the low speed state, check that R632.x should be in the low speed state.		

Alarm ID	SPD–036	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

Alarm ID	SPD-037	Alarm 标题	【Inconsistent Resolution Settings】
Description	The resolutions of the controller and the drive are not set consistently. The settings are considered consistent if the following equation is satisfied: 1. Single feedback: Spindle Encoder Resolution (Pr1651~) * Spindle Encoder Scaling Factor (Pr1661~) = Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E). 2. Dual feedback or Semi-Closed Loop: Spindle Encoder Resolution (Pr1651~) * Spindle Encoder Scaling Factor (Pr1661~) = 2nd Encoder Resolution (Pn-922) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E).		
Possible Cause	1. Improper controller parameter settings: a. Spindle Encoder Resolution. b. Spindle Encoder Scaling Factor. 2. Improper driver parameter settings: a. Electronic Gear Ratio. b. Encoder Resolution.		
Solution	1. Change the controller parameters Pr1651~ and Pr1661~ so that the equation mentioned above is satisfied. 2. Change the drive parameters so that the equation mentioned above is satisfied: a. Electronic Gear Ratio: Pn-20E, Pn-210. b. Encoder Resolution: Pn-902, Pn-922. 3. To clear the alarm, a. Reset the controller if ONLY the controller parameter(s) is(are) modified. b. Restart both of the controller and the driver if the driver parameter(s) is(are) modified.		

Alarm ID	SPD-038	Alarm title	Poor contact of communication wire
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

Alarm ID	SPD–039	Alarm title	The hardware doesn't receive communication packet

Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		
Alarm ID	**SPD–040**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		
Alarm ID	**SPD–041**	**Alarm title**	**Driver software misses communication packet**
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		
Alarm ID	**SPD–042**	**Alarm title**	**【The drive does not support control parameters switch】**
Description	The drive does not support control parameters switch but Pr1994 was set 1.		
Possible Reason	It is not a Syntec drive and Pr1994 set 1.		

Solution	1. Use the Syntec drive which support control parameters switch. 2. Set Pr1994 to 0.

P.S. Valid version of SPD-042 : Before 10.118.30H, 10.118.32H, 10.118.40L, 10.118.41L, 10.118.47.

Alarm ID	**SPD–043**	**Alarm title**	**【The control parameters switch failed】**
Description	The drive failed to switch the control parameters.		
Possible Reason	1. Enabling the following functions may cause the control parameters switch to fail. : a. V/F control b. Auto tunning 2. Abnormal communication may cause control parameters switch to fail.		
Solution	1. Disable the mentioned functions. 2. Please check communication system.		

Alarm ID	**SPD–044**	**Alarm Title**	**Spindle is stalled. Please check environment and parameter setting.**
Description	When the spindle rotation signal is triggered, the spindle speed cannot reach 15% of user's setting spindle speed during the stall check time. **(Spindle stall check time= Max(15, 3 * Pr1831~ * 1000), it means take the maximum value between 15 seconds and 3 times of spindle acceleration/deceleration time.)** • This alarm only protects servo spindle from stalling.		
Possible Cause	1. The mechanism is stuck or loading is too heavy. 2. Insufficient control performance of spindle motor. 3. Abnormity of spindle motor feedback.		
Solution	1. Please confirm whether the mechanism is stuck or loading is too heavy. 2. Please confirm spindle acceleration/ deceleration speed setting is correct. (Pr1831~) 3. Please confirm spindle works well. 4. Please re-tune servo drive.		

5.1 SPD-020 Spindle can't take correct index signal as reference

Alarm ID	**SPD-020**	**Alarm Title**	**Spindle can't take correct index signal as reference**
Description	Spindle can't use hardware and software's index signal.		

Alarm ID	SPD-020	Alarm Title	Spindle can't take correct index signal as reference
Possible Cause	1. If encoder sets on the motor side (Pr1811 sets as 1). When gear number of motor (Pr1681~): gear number of ballscrew (Pr1682~) is not integer ratio, it can't correspond all motor position to all screw position, so the index signal may be inaccurate. If encoder is on the motor side (Pr1811 sets as 1). When axis type (Pr221~) sets as 3 or 5, because the mechanical coordinate is not between 0~360, so it can't calculate the index signal position from feedback. 2. When the spindle rotates more than one revolution, a. If the diagnosis variable 52 has changed, check whether the Pr1993 is set too small. b. If the diagnostic variable 52 remains unchanged, check whether i. the encoder signal cable is not connected properly. ii. axis board pin is fault. iii. encoder Z phase signal is fault. 3. Because the index signal returned by the drive is unreasonable, the controller cannot execute the threading function.		
Solution	1. The controller doesn't support threading or homing function under this condition, please use other rotation type axial. 2. Refer to the following: a. Enlargement Pr1993 setting value. b. Or i. Connect the encoder signal cable securely. ii. Replace the axis board. iii. Replace the encoder Z phase signal. 3. Please contact the drive vendor.		

5.2 SPD-034 Dynamic Inspection Setting Error

Alarm ID	SPD-034	Alarm Topic	【Dynamic Inspection Setting Error】
Description	When enable the spindle dynamic equilibrium inspection function, not set software, hardware deployment properly.		
Probable Reason	1. The encoder port did not connect with Syntec acceleration gauge. 2. The setting error of encoder parameter in Syntec driver parameter. 3. The driver version did not support Syntec acceleration gauge.		

Exclusion Method	1. Please following the setting step below: a. Use Syntec acceleration gauge. b. Ensure Syntec driver parameter Pn-810 setting value. i. If the configuration is semi-closed loop, please set Pn-810 as 1. 1. Set the Syntec acceleration gauge port number to Syntec driver parameter Pn-901. 2. Set Syntec driver parameter Pn-900 as 13. 3. Set Syntec driver parameter Pn-90D as 3. ii. If the configuration is other, set Pn-810 as 0. 1. Set the Syntec acceleration gauge port number to Syntec driver parameter Pn-921. 2. Set Syntec driver parameter Pn-920 as 13. 3. Set Syntec driver parameter Pn-92D as 3. c. Do power outage and reopen the controller and driver, 2. Please upload the driver to 2.5.1 or afterward version.

5.3 SPD-035 Dynamic Equilibrium Function Timeout

Alarm ID	**SPD-035**	**Alarm Topic**	**【Dynamic Equilibrium Function Timeout】**
Description	Timeout occurred in spindle dynamic equilibrium function.		
Probable Reasons	1. The spindle steady speed inspection fail. The probable reason is following: a. Spindle encoder is abnormal. b. The spindle rotation does not reach steady speed standard. The dynamic equilibrium function request the difference between the target speed and the return speed is within 10RPM. 2. When open dynamite equilibrium inspection function, you could also open analysis platform （or OpenGuide） oscilloscope function. 3. When open dynamite equilibrium adjustment function, you could also open analysis platform （or OpenGuide） oscilloscope function.		
Exclusion method	1. To exclude the steady speed inspection failure, you can refer to following method: a. Please check whether the spindle is adjusted to steady speed error is less than 10RPM. b. Please check whether spindle encoder is operating correctly. c. Please check the spindle would not frequently accelerate or decelerate when doing dynamic equilibrium function. 2. Choose one of the following options.: a. Please close analysis platform（or OpenGuide） and open dynamic equilibrium inspection. b. Please close dynamic equilibrium inspection. 3. Choose one of the following options.: a. Please close analysis platform（or OpenGuide） and open dynamic equilibrium adjustment. b. Please close dynamic equilibrium adjustment.		

5.4 SPD–001 A, B Encoder Feedback Signal Error

Alarm ID	SPD–001	Alarm Title	A, B Encoder Feedback Signal Error
Description	Axial card detects A, B encoder has feedback signal error.		
Possible Cause	Only happens when Pr9 parameter axis board type sets as 0 (EMP2). When axis board type sets as EMP2, axis board internally will automatically detects A, B encoder signal. If the signal is wrong, or there is a phase A signal but no phase B signal, then this alarm is issued.		
Solution	Check servo cable or change axis board.		

5.5 SPD–002 Error counter overflows

Alarm ID	SPD–002	Alarm Title	Error counter overflows
Description	Axis card detects encoder feedback overflows.		
Possible Cause	Only happen when Pr9 parameter axis card type sets as 0(EMP2), 4(PMC4), and 6(SERVO6). When axis card type sets as 0(EMP2), 4(PMC4), and 6(SERVO6), axis card inside will detects A, B encoder signal. If signal is wrong or input signal is too big, and this alarm will be issued.		
Solution	Check servo cable or change axis card.		

5.6 SPD–003 Encoder module error

Alarm ID	SPD–003	Alarm Title	Encoder module error
Description	Currently no application.		
Possible Cause			
Solution			

5.7 SPD–004 NO interrupted index signal

Alarm ID	SPD–004	Alarm Title	NO interrupted index signal
Description	No application currently.		
Possible Cause			
Solution			

5.8 SPD–005 Pulse signal is over rated value

Alarm ID	SPD–005	Alarm Title	Pulse signal is over rated value
Description	The command count send from controller is too much. Software calculate that over 2047 pluses in a interpolation time needs to be send.		
Possible Cause	1. Software interpolation time sets too long. 2. Movement speed is too fast. 3. Servo resolution sets too high. 4. Backlash compensation value or pitch compensation value sets too large. 5. Start feedforward compensation function.		
Solution	1. Decrease software interpolation time setting (Pr3203), nott recommend to set lower than 2000. 2. Lower movement speed limit setting. (Pr461~Pr480) 3. Reduce servo resolution. (Driver and Pr61~Pr80) 4. Sets appropriate mechanism compensation time constant. (Pr1401~Pr1420) 5. Reduce or close feedforward compensation function. (Pr581~Pr600) 6. Please contact machine manufacturer.		

5.9 SPD–006 Can’t clear index condition

Alarm ID	SPD–006	Alarm Title	Can’t clear index condition
Description	No application currently.		
Possible Cause			
Solution			

5.10 SPD–009 Driver Alarm

Alarm ID	SPD–009	Alarm Title	Driver Alarm
Description	Driver sends an alarm signal.		
Possible Cause	Driver alarm is mostly because of external reasons: For example: 1. Driver temperature is too high. 2. Encoder cable wiring is wrong. 3. Internal parameter setting is wrong. 4. Not matched with servo motor. 5. Driver malfunction.		
Solution	Please follow driver alarm manual for troubleshooting.		

5.11 SPD–017 Spindle Orientation Position check error

Alarm ID	SPD–017	Alarm Title	Spindle Orientation Position check error
Description	Spindle orientation (C61) triggered and spindle is stopped. Spindle orientation error is over one degree.		
Possible Cause	1. During spindle orientation, inverter module position control mode is not activated. 2. Bad spindle motor tuning and caused the motor stills under jitter correction state after finish orientation. 3. Motor feedback signal lose position command. 4. Spindle zero speed check window (Pr901~Pr916) sets too large.		
Solution	1. Confirm inverter module’s control mode. 2. Retune the spindle motor. 3. Confirm motor feedback cable's quality, or contact motor supplier to check if the encoder is abnormal. 4. Reduce zero check window.		
Advanced Explanation	Observe System Data No. 52 when motor operate in the same direction, if the variation will increase or decrease by a multiple of Pr1651*Pr1661? If it’s not, it means motor feedback signal lose position command. Please increase the quality of motor feedback cable, or contact motor supplier to check if encoder is abnormal.		

5.12 SPD–018 Can’t find spindle orientation index signal

Alarm ID	SPD–018	Alarm Title	Can’t find spindle orientation index signal
Description	During spindle orientation, can’t find index signal over a certain time.		
Possible Cause	1. If system data no. 52 doesn’t change with rotation of motor, it means controller can’t read motor index signal. 2. If system data no. 52 change with the rotation of motor, it means controller can read motor index signal, but because spindle rotation speed is not reduced to half of the rated speed, so it can’t execute orientation.		
Solution	1. Please increase quality of motor feedback cable, or contact motor supplier to check if encoder is abnormal. 2. Please check if belt is loose or gear setting ratio is correct.		

5.13 SPD–021 Do tapping when neutral

Alarm ID	SPD–021	Alarm Title	Do tapping when neutral
Description	When spindle switch to neutral, and execute tapping command.		
Possible Cause	Spindle doesn’t change to the right gear position.		
Solution	Change spindle’s gear (R24) to 1~3 gear, then execute tapping command.		

5.14 SPD–022 Pr1791~ sets as 4. Only support Syntec Inverter Module

Alarm ID	SPD–022	Alarm Title	Pr1791~ sets as 4. Only support Syntec Inverter Module
Description	When Pr1791~ sets as 4, it has to be used only with Syntec Spindle Motor Module.		
Possible Cause	Use Yaskawa or other brand’s serial bus driver as spindle.		
Solution	Change Pr1791~ into 1 or 3.		

5.15 SPD–023 Spindle rotation speed hasn't reached yet. Please check parameter setting.

<table>
<tr><th>Alarm ID</th><th>SPD–023</th><th>Alarm Title</th><th>Spindle rotation speed hasn't reached yet. Please check parameter setting.</th></tr>
<tr><td>Description</td><td colspan="3">After enables spindle rotation speed reach function (Pr1991 not sets as 0), spindle rotation speed has to reach to user's setting spindle speed during the check time, or this alarm will be issued.
(Spindle rotation speed reach check time= Max(30, 3 * Pr1831 * 1000), it means take the maximum value between 30 seconds and 3 times of spindle acceleration/ deceleration time.)
• If you want to use Pr1991 spindle rotation speed reach check function, even if it is inverter spindle, it also needs to trigger spindle clockwise/counter-clockwise rotation signal C64/C65, R583.x/R584.x to turn the system spindle into rotation condition.
• Adjust inverter spindle's rotation speed reach check regulation to be the same with servo spindle. When Pr1991 is not equals to 0, spindle speed reach conditions are as following:
<table>
<tr><th>Spindle clockwise / counter-clockwise rotation signal C64/ C65, R583.x/R584.x</th><th>Established conditions of spindle rotation reach</th></tr>
<tr><td rowspan="2">On</td><td>The speed difference between spindle feedback (Fbk) and speed command (Cmd) is smaller than speed command (Cmd) times spindle speed reach check range's percentage. (Pr1992 or Pr1993)</td></tr>
<tr><td>ie | Fbk - Cmd | < Cmd * (Pr1992 or Pr1993) * 0.01</td></tr>
<tr><td rowspan="2">Off</td><td>Spindle feedback (Fbk) is smaller than 50RPM.</td></tr>
<tr><td>i.e. Fbk < 50 (RPM)</td></tr>
</table>
• Note: If spindle clockwise/counter-clockwise rotation signal is off it will also check, in order to process when spindle clockwise/counter-clockwise rotation signal status turn from on to off, have to ensure spindle speed has reduce to certain low speed, and will be identified as spindle speed reached.
• When spindle doesn't install encoder (Pr1711~ sets as 0), spindle speed reach conditions are as following:</td></tr>
</table>

<table>
<tr><td>Alarm ID</td><td>SPD–023</td><td>Alarm Title</td><td>Spindle rotation speed hasn’t reached yet. Please check parameter setting.</td></tr>
<tr><td></td><td colspan="3"><table>
<tr><th>Spindle Type</th><th>Established conditions of spindle rotation reach</th></tr>
<tr><td>Serial Bus Spindle</td><td>Directly shows driver’s information if spindle speed has reached or not.</td></tr>
<tr><td>General Pulse Spindle</td><td>When S59 bit sets as on, controller will consider that spindle speed has reached.
If you want to do spindle rotation speed reach check, send an O bit to TB16IN from driver, and PLC can gets this information and execute speed reach check in PLC.</td></tr>
</table></td></tr>
<tr><td>Possible Cause</td><td colspan="3">When spindle speed check function enables, spindle speed can’t reach to target speed during rotation speed check’s set time.
(Spindle rotation speed reach check time= Max(30, 3 * Pr1831 * 1000), it means take the maximum value between 30 seconds and 3 times of spindle acceleration/ deceleration time.)</td></tr>
<tr><td>Solution</td><td colspan="3">1. Please confirm spindle rotation speed check function’s related parameter setting is correct. (Pr1991, Pr1992, Pr1993)
2. Please confirm spindle acceleration/ deceleration speed setting is correct. (Pr1831~)
3. Please confirm spindle clockwise / counter-clockwise rotation signal C64/C65, R583.x/ R584.x triggered normally.
4. Please confirm spindle works well.
5. Please confirm spindle feedback works well.</td></tr>
</table>

SYNTEC

5.16 SPD–024 Fail to do spindle zero speed check

Alarm ID	SPD–024	Alarm Title	Fail to do spindle zero speed check
Description	This alarm will be triggered while a spindle fails to pass zero speed check over 5 seconds and the feedback speed is not decreasing. The following situations require zero speed check: 1. For a serial spindles with no servo command, the spindle will enter servo-off state after passing zero speed check. 2. When exiting the orientation mode, a spindle will do coordinate synchronization after passing zero speed check. 3. Under circumstances of emergency stop (while C36 is on) or critical alarms, a spindle will enter servo-off state after passing zero speed check. Note: The zero speed check window is 3 rpm if the emergency stop or a critical alarm is triggered, otherwise it is based on Pr901~Pr920 Axis zero speed check window.		
Possible Cause	1. Insufficient control performance of spindle motor. 2. Abnormity of spindle motor feedback. 3. Pr901~Pr920 Axis zero speed check window is too narrow. (For Serial Bus Line especially)		
Solution	1. Please re-tune servo drive. 2. Check whether the motor feedback of the spindle is functioned well. 3. Increase Pr901~Pr920 Axis zero speed check window, a setting value within 0.1 to 0.5 degree is recommended. (The suggested setting value = (0.1~0.5) * spindle resolution/ 360)		
Advanced Explanation	Check the fluctuation of speed and loading rate of the spindle when it is under constant rotating command. A speed fluctuation beyond Pr901~Pr920 Axis zero speed check window -normally led by improper servo tuning- would make it fail to pass zero speed check.		

SYNTEC

5.17 SPD–025 Please modified resolution related parameters from the page of serial parameters

Alarm ID	SPD–025	Alarm Title	Please modified resolution related parameters from the page of serial parameters
Description	The axis of the Syntec driver can synchronize the resolution related setting. If the sensor type of controller and driver are the same, the related controller parameters will be synchronized from the driver. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. The matched condition of sensor type (single feedback): • Driver parameter Pn-911 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-911 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 The matched condition of sensor type (driver dual feedback or semi-closed loop): • Driver parameter Pn-931 = 1 (Rotary), Pr201~Pr220 Axial sensor type. = 0, 3, 4 • Driver parameter Pn-931 = 2 (Linear), Pr201~Pr220 Axial sensor type. = 1, 5 Axial resolution related parameters: • Pr61~Pr80 Axial encoder resolution • Pr81~Pr100 Axial encoder scaling factor Driver resolution related parameters: • Pn-902 Encoder Resolution • Pn-922 2nd Encoder Resolution • Pn-20E Electronic Gear Ratio (Numerator) • Pn-210 Electronic Gear Ratio (Denominator) Version information: • This alarm is supported in 10.118.41Q, 10.118.48B, 10.118.49 and later versions.		
Possible Cause	Modified the resolution related parameters in controller after the resolution synchronization of the Syntec driver.		
Solution	1. Reset the controller. 2. If the resolution related parameters need to be modified, please modify the parameters in the page of serial parameters. Then restart the controller and the driver.		

5.18 SPD–026 Spindle orientation offset. Please fix this parameter in Syntec spindle into zero.

Alarm ID	**SPD–026**	**Alarm Title**	**Spindle orientation offset. Please fix this parameter in Syntec spindle into zero.**
Description	If the spindle is Syntec serial bus spindle, then the modification of spindle orientation offset parameter (Pr1751~Pr1756) is invalid.		
Possible Cause	When controller detects that the spindle is Syntec serial bus, corresponding spindle orientation offset parameter (Pr1751~Pr1756) set value should be non-zero.		
Solution	Sets corresponding spindle orientation offset parameter as zero. Press to reset to remove this alarm.		

5.19 SPD–027 This spindle type doesn’t support spindle orientation function

Alarm ID	**SPD–027**	**Alarm Title**	**This spindle type doesn’t support spindle orientation function**
Description	This spindle type doesn’t support spindle orientation function.		
Possible Cause	This spindle type doesn’t support spindle orientation function.		
Solution	Set Pr1791~ as other spindle types (0~5), which support spindle orientation function.		

5.20 SPD–028 This spindle type doesn’t support spindle tapping function

Alarm ID	**SPD–028**	**Alarm Title**	**This spindle type doesn’t support spindle tapping function**
Description	This spindle type doesn’t support spindle tapping function.		
Possible Cause	This spindle type doesn’t support spindle tapping function.		
Solution	Set Pr1791~ as other spindle types (0~5), which support spindle tapping function.		

5.21 SPD–029 This spindle type doesn’t support C axis mode

Alarm ID	SPD–029	Alarm Title	This spindle type doesn’t support C axis mode
Description	This spindle type doesn’t support C axis mode.		
Possible Cause	This spindle type doesn’t support C axis mode.		
Solution	Set Pr1791~ as servo spindle types 1~4, which support C axis mode.		

5.22 SPD–030 This product doesn’t support Syntec Spindle

Alarm ID	SPD–030	Alarm Title	This product doesn’t support Syntec Spindle
Description	This product doesn’t support Syntec Spindle.		
Possible Cause	This spindle of this controller product doesn’t support Syntec driver.		
Solution	1. Please change spindle into other non-Syntec driver. 2. Please contact sales personnel to buy other types of controller.		

5.23 SPD–031 The spindle does not support the dual winding switching function.

Alarm ID	SPD–031	Alarm Title	The spindle does not support the dual winding switching function.
Description	This spindle type or drive type does not support the dual winding switching function.		
Possible Cause	1. This spindle does not support the winding switching function. 2. The driver does not support the winding switching function.		
Solution	1. Set R631 and R632 to zero. 2. The spindle uses a Syntec driver. 3. Set Pr1791~ to 2 and 4. 4. Update the drive to version 2.14.0 or above.		

5.24 SPD–032 dual winding switch failed

Alarm ID	SPD–032	Alarm Title	dual winding switch failed
Description	An abnormality occurred during the dual winding switching process.		
Possible Cause	1. The state of R631.x is changed during the winding switching. 2. When R632.x is updating, its status is different from R602.x. 3. The winding switching process takes more than five seconds. 4. The drive does not turn on the dual coil parameters switching function.		
Solution	1. Switch R631.x to the low speed state. This requirement is removed after (included) software version 10.118.85. 2. Switch R632.x to the state to which R602.x belongs. 3. When the above conditions are met, after the spindle stops, press reset to remove the alarm.		

5.25 SPD–033 Y-Delta zero speed start check failed

Alarm ID	SPD–033	Alarm Title	Y-Delta zero speed start check failed
Description	The controller will start the spindle in the high speed coil state. This alarm is removed after (included) software version 10.118.85.		
Possible Cause	1. R631.x is in the high speed state when servo-on. 2. Both R631.x and R602.x are in the low speed state when servo-on, but R632.x is in the high speed state.		
Solution	1. Set R631.x to the low speed state. 2. When both R631.x and R602.x are in the low speed state, check that R632.x should be in the low speed state.		

5.26 SPD–036 Drive communication error

Alarm ID	SPD–036	Alarm title	Drive communication error

Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.

5.27 SPD–037 Inconsistent Resolution Settings

Alarm ID	**SPD-037**	**Alarm 标题**	**【Inconsistent Resolution Settings】**
Description	The resolutions of the controller and the drive are not set consistently. The settings are considered consistent if the following equation is satisfied: 1. Single feedback: Spindle Encoder Resolution (Pr1651~) * Spindle Encoder Scaling Factor (Pr1661~) = Resolution (Pn-902) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E). 2. Dual feedback or Semi-Closed Loop: Spindle Encoder Resolution (Pr1651~) * Spindle Encoder Scaling Factor (Pr1661~) = 2nd Encoder Resolution (Pn-922) * Denominator of Electronic Gear Ratio (Pn-210) / Numerator of Electronic Gear Ratio (Pn-20E).		
Possible Cause	1. Improper controller parameter settings: a. Spindle Encoder Resolution. b. Spindle Encoder Scaling Factor. 2. Improper driver parameter settings: a. Electronic Gear Ratio. b. Encoder Resolution.		
Solution	1. Change the controller parameters Pr1651~ and Pr1661~ so that the equation mentioned above is satisfied. 2. Change the drive parameters so that the equation mentioned above is satisfied: a. Electronic Gear Ratio: Pn-20E, Pn-210. b. Encoder Resolution: Pn-902, Pn-922. 3. To clear the alarm, a. Reset the controller if ONLY the controller parameter(s) is(are) modified. b. Restart both of the controller and the driver if the driver parameter(s) is(are) modified.		

5.28 SPD–038 Poor contact of communication wire

Alarm ID	SPD-038	Alarm title	Poor contact of communication wire
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

5.29 SPD–039 The hardware doesn't receive communication packet

Alarm ID	SPD–039	Alarm title	The hardware doesn't receive communication packet
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

5.30 SPD–040 The communication packet is disturbed by noise

Alarm ID	SPD–040	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		

Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.

5.31 SPD–041 Driver software misses communication packet

Alarm ID	**SPD–041**	**Alarm title**	**Driver software misses communication packet**
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

5.32 SPD–042 The drive does not support control parameters switch

Alarm ID	**SPD–042**	**Alarm title**	**【The drive does not support control parameters switch】**
Description	The drive does not support control parameters switch but Pr1994 was set 1.		
Possible Reason	It is not a Syntec drive and Pr1994 set 1.		
Solution	1. Use the Syntec drive which support control parameters switch. 2. Set Pr1994 to 0.		

P.S. Valid version of SPD-042 : Before 10.118.30H, 10.118.32H, 10.118.40L, 10.118.41L, 10.118.47.

5.33 SPD–043 The control parameters switch failed

Alarm ID	**SPD–043**	**Alarm title**	**【The control parameters switch failed】**
Description	The drive failed to switch the control parameters.		

Possible Reason	1. Enabling the following functions may cause the control parameters switch to fail. : a. V/F control b. Auto tunning 2. Abnormal communication may cause control parameters switch to fail.
Solution	1. Disable the mentioned functions. 2. Please check communication system.

5.34 SPD–044 Spindle is stalled. Please check environment and parameter setting.

Alarm ID	**SPD–044**	**Alarm Title**	**Spindle is stalled. Please check environment and parameter setting.**
Description	When the spindle rotation signal is triggered, the spindle speed cannot reach 15% of user's setting spindle speed during the stall check time. **(Spindle stall check time= Max(15, 3 * Pr1831~ * 1000), it means take the maximum value between 15 seconds and 3 times of spindle acceleration/deceleration time.)** • This alarm only protects servo spindle from stalling.		
Possible Cause	1. The mechanism is stuck or loading is too heavy. 2. Insufficient control performance of spindle motor. 3. Abnormity of spindle motor feedback.		
Solution	1. Please confirm whether the mechanism is stuck or loading is too heavy. 2. Please confirm spindle acceleration/ deceleration speed setting is correct. (Pr1831~) 3. Please confirm spindle works well. 4. Please re-tune servo drive.		

SYNTEC

6 Syntax Compiler Alarm - COM

Alarm ID	**COM–001**	**Alarm Title**	**EOF in comment**
Description	The symbol "(*" and "*)" must be used in pairs, if the program uses "(*" as the beginning of the comment, but doesn't use "*)" at the end of the comment. System will issue this alarm.		
Possible Cause	Programming error.		
Solution	Using symbol "(*" before command and symbol "*)" after command in MACRO.		

Alarm ID	**COM–002**	**Alarm Title**	**No string ending character**
Description	In MACRO the string in brackets of PRINT command has no " string ending character.		
Possible Cause	MACRO programming error.		
Solution	Check PRINT command in MACRO.		

Alarm ID	**COM–003**	**Alarm Title**	**Syntax error**
Description	MACRO has syntax error when controller interprets it.		
Possible Cause	1. MACRO Programming error. 2. Use APP syntax incorrectly in G code macro, for example: G200 X10. APP "appname". 3. Use APP syntax in M/T code 4. The single line syntax in MACRO program is too complicated to parse		
Solution	1. Check MACRO syntax according to line number shown in the alarm. 2. The APP argument must be the first of the G code argument, for example: G200 APP"appname" X10. 3. M/T code does not support using APP syntax, please rewrite the program. 4. Split calculations to simplify statements.		

Alarm ID	**COM–004**	**Alarm Title**	**Illegal variable**
Description	To save in a variable which is not accessible in system.		
Possible Cause	Variable usage error.		

Solution	Check variable in MACRO and confirm the variable is accessible in system.

Alarm ID	**COM–005**	**Alarm Title**	**Expression too complex**
Description	When MACRO is too complicated.		
Possible Cause	MACRO programming error.		
Solution	Check whether MACRO logic is clear and correct.		

Alarm ID	**COM–006**	**Alarm Title**	**EXIT statement outside loop statement**
Description	If EXIT command is not inside loop command, system will issue alarm.		
Possible Cause	MACRO Programming error.		
Solution	Check whether EXIT command in MACRO program is used correctly.		

Alarm ID	**COM–007**	**Alarm Title**	**Repeat loop too deep**
Description	IF Loop command in MACRO such as REPEAT loop, REPEAT loop, WHILE loop, FOR loop repeats more than 10 times, system will issue this alarm.		
Possible Cause	MACRO Programming error.		
Solution	Modify MACRO program to avoid too many loop commands.		

Alarm ID	**COM–008**	**Alarm Title**	**absent statement ending character ';'**
Description	Program doesn’t have ending character";" when MACRO command finishes.		
Possible Cause	MACRO Programming error.		
Solution	Check MACRO program to confirm whether it has the ending character.		

Unable to render include or excerpt-include. Could not retrieve page.

Alarm ID	**COM–010**	**Alarm Title**	**absent right ')'**

Description	In program, symbol “(” and “)” must be used in pairs.		
Possible Cause	MACRO programming error		
Solution	Check MACRO program to confirm whether using “(” and “)” symbol is used in pairs.		
Alarm ID	**COM–011**	**Alarm Title**	**absent right ']'**
Description	In MACRO program, symbol “[” and “]” must be used in pairs.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if “[” and “]” is used in pairs.		
Alarm ID	**COM–012**	**Alarm Title**	**absent 'FOR' keyword in FOR statement**
Description	In MACRO, incorrectly uses TO to define loop condition in FOR loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm TO is correctly used in FOR loop.		
Alarm ID	**COM–013**	**Alarm Title**	**absent 'DO' keyword in FOR statement**
Description	In MACRO, FOR loop uses DO to define Implement task in loop incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if DO is used correctly in FOR loop.		
Alarm ID	**COM–014**	**Alarm Title**	**absent 'END_FOR' keyword in FOR statement**
Description	In MACRO, END_FOR is not used correctly in FOR loop to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_FOR is correctly used in FOR loop.		

Alarm ID	**COM–015**	**Alarm Title**	**absent 'UNTIL' keyword in REPEAT statement**
Description	In MACRO, REPEAT loop uses UNTIL to define loop condition incorrectly.		
Possible Cause	MACRO programming error		
Solution	Check MACRO program to confirm if UNTIL is used correctly in REPEAT loop.		
Alarm ID	**COM–016**	**Alarm Title**	**absent 'END_REPEAT' keyword in REPEAT statement**
Description	In MACRO, REPEAT loop doesn’t have END_REPEAT to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_REPEAT is used correctly in REPEAT loop.		
Alarm ID	**COM–017**	**Alarm Title**	**absent 'DO' keyword in WHILE statement**
Description	In MACRO, WHILE loop uses DO to define task incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if DO is correctly used in WHILE loop.		
Alarm ID	**COM–018**	**Alarm Title**	**absent 'END_WHILE' keyword in WHILE statement**
Description	In MACRO, WHILE loop doesn’t have END_WHILE to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_WHILE is used correctly to end the WHILE loop.		
Alarm ID	**COM–019**	**Alarm Title**	**absent 'THEN' keyword in IF statement**
Description	In MACRO, IF uses THEN to define task incorrectly.		

Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END is correctly used in IF loop.		
Alarm ID	**COM–020**	**Alarm Title**	**absent 'END_IF' or 'ELSE' keyword in IF statement**
Description	In MACRO, IF loop doesn’t use ELSE or END_IF correctly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if ELSE or END_IF is used correctly in IF loop.		
Alarm ID	**COM–021**	**Alarm Title**	**absent 'END_IF' keyword in IF statement**
Description	In MACRO, IF loop uses END_IF to finish loop incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, END_IF is correctly used in IF loop.		
Alarm ID	**COM–022**	**Alarm Title**	**absent 'OF' keyword in CASE statement**
Description	In MACRO, OF is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if OF is used correctly in CASE command.		
Alarm ID	**COM–023**	**Alarm Title**	**absent 'END_CASE' or 'ELSE' keyword in CASE statement**
Description	In MACRO program, ELSE or END_CASE is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if ELSE or END_CASE is used correctly in CASE command.		

Alarm ID	COM–024	Alarm Title	absent 'END_CASE' keyword in CASE statement
Description	In MACRO, END_CASE keyword is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, ensure that END_CASE is used before CASE statement ending.		

Unable to render include or excerpt-include. Could not retrieve page.

Alarm ID	COM-026	Alarm Title	Illegal MOVJ instruction input
Description	Illegal MOVJ command input.		
Possible Cause	MOVJ command format error.		
Solution	Check if MOVJ command format is correct.		

Alarm ID	COM-027	Alarm Title	Invalid Character
Description	Only ASCII characters are acceptable in NC files. Using non-ASCII characters is thus unacceptable. Note: Listed below are the special cases in which non-ASCII characters are considered acceptable: 1. Comment. 2. Arguments of MACRO functions that are of string type.		
Possible Cause	There are non-ASCII characters in the NC file. Common cases include: 1. Using full width semicolons instead of half width ones. 2. Writing comments without using comment characters. 3. Using full-width spaces instead of half width ones.		
Solution	Please check the machining program to make sure that only ASCII is used in comments and macros. Recommended to check: 1. Make sure that full-width semicolons is not used. 2. Make sure that all the program comments are added with comment syntax. 3. Make sure that full-width space is not used. Suggested exclusions: • Confirm the line number of the stopped processing file, delete and re-enter the processing file of this line on the controller.		

Alarm ID	COM-028	Alarm Title	Illegal use of ' / '
Description	Incorrectly use single block optional skip function " / ".		
Possible Cause	MACRO programming error, function syntax or variable calculation is written after " / ".		
Solution	Check the incorrect line to ensure no function syntax or variable operation is written after " / ".		

6.1 COM–001 EOF in comment

Alarm ID	COM–001	Alarm Title	EOF in comment
Description	The symbol "(*" and "*)" must be used in pairs, if the program uses "(*" as the beginning of the comment, but doesn't use "*)" at the end of the comment. System will issue this alarm.		
Possible Cause	Programming error.		
Solution	Using symbol "(*" before command and symbol "*)" after command in MACRO.		

6.2 COM–002 No string ending character

Alarm ID	COM–002	Alarm Title	No string ending character
Description	In MACRO the string in brackets of PRINT command has no " string ending character.		
Possible Cause	MACRO programming error.		
Solution	Check PRINT command in MACRO.		

6.3 COM–003 Syntax error

Alarm ID	COM–003	Alarm Title	Syntax error
Description	MACRO has syntax error when controller interprets it.		

Possible Cause	1. MACRO Programming error. 2. Use APP syntax incorrectly in G code macro, for example: G200 X10. APP "appname". 3. Use APP syntax in M/T code 4. The single line syntax in MACRO program is too complicated to parse
Solution	1. Check MACRO syntax according to line number shown in the alarm. 2. The APP argument must be the first of the G code argument, for example: G200 APP"appname" X10. 3. M/T code does not support using APP syntax, please rewrite the program. 4. Split calculations to simplify statements.

6.4 COM–004 Illegal variable

Alarm ID	**COM–004**	**Alarm Title**	**Illegal variable**
Description	To save in a variable which is not accessible in system.		
Possible Cause	Variable usage error.		
Solution	Check variable in MACRO and confirm the variable is accessible in system.		

6.5 COM–005 Expression too complex

Alarm ID	**COM–005**	**Alarm Title**	**Expression too complex**
Description	When MACRO is too complicated.		
Possible Cause	MACRO programming error.		
Solution	Check whether MACRO logic is clear and correct.		

6.6 COM–006 EXIT statement outside loop statement

Alarm ID	**COM–006**	**Alarm Title**	**EXIT statement outside loop statement**
Description	If EXIT command is not inside loop command, system will issue alarm.		
Possible Cause	MACRO Programming error.		

Solution	Check whether EXIT command in MACRO program is used correctly.

6.7 COM–007 Repeat loop too deep

Alarm ID	**COM–007**	**Alarm Title**	**Repeat loop too deep**
Description	IF Loop command in MACRO such as REPEAT loop, REPEAT loop, WHILE loop, FOR loop repeats more than 10 times, system will issue this alarm.		
Possible Cause	MACRO Programming error.		
Solution	Modify MACRO program to avoid too many loop commands.		

6.8 COM–008 absent statement ending character ';'

Alarm ID	**COM–008**	**Alarm Title**	**absent statement ending character ';'**
Description	Program doesn’t have ending character";" when MACRO command finishes.		
Possible Cause	MACRO Programming error.		
Solution	Check MACRO program to confirm whether it has the ending character.		

6.9 COM–009 Wrong assignment character ':='

Alarm ID	**COM–009**	**Alarm Title**	**wrong assignment character ':='**
Description	In MACRO program, if assigning value to variable doesn’t use the correct character ':= ' system will issue alarm.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, whether value assignment character is correctly used.		

6.10 COM–010 absent right ')'

Alarm ID	**COM–010**	**Alarm Title**	**absent right ')'**

Description	In program, symbol “(” and “)” must be used in pairs.
Possible Cause	MACRO programming error
Solution	Check MACRO program to confirm whether using “(” and “)” symbol is used in pairs.

6.11 COM–011 absent right ']'

Alarm ID	**COM–011**	**Alarm Title**	**absent right ']'**
Description	In MACRO program, symbol “[” and “]” must be used in pairs.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if “[” and “]” is used in pairs.		

6.12 COM–012 absent 'FOR' keyword in FOR statement

Alarm ID	**COM–012**	**Alarm Title**	**absent 'FOR' keyword in FOR statement**
Description	In MACRO, incorrectly uses TO to define loop condition in FOR loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm TO is correctly used in FOR loop.		

6.13 COM–013 absent 'DO' keyword in FOR statement

Alarm ID	**COM–013**	**Alarm Title**	**absent 'DO' keyword in FOR statement**
Description	In MACRO, FOR loop uses DO to define Implement task in loop incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if DO is used correctly in FOR loop.		

6.14 COM–014 absent 'END_FOR' keyword in FOR statement

Alarm ID	COM–014	Alarm Title	absent 'END_FOR' keyword in FOR statement
Description	In MACRO, END_FOR is not used correctly in FOR loop to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_FOR is correctly used in FOR loop.		

6.15 COM–015 absent 'UNTIL' keyword in REPEAT statement

Alarm ID	COM–015	Alarm Title	absent 'UNTIL' keyword in REPEAT statement
Description	In MACRO, REPEAT loop uses UNTIL to define loop condition incorrectly.		
Possible Cause	MACRO programming error		
Solution	Check MACRO program to confirm if UNTIL is used correctly in REPEAT loop.		

6.16 COM–016 absent 'END_REPEAT' keyword in REPEAT statement

Alarm ID	COM–016	Alarm Title	absent 'END_REPEAT' keyword in REPEAT statement
Description	In MACRO, REPEAT loop doesn’t have END_REPEAT to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_REPEAT is used correctly in REPEAT loop.		

6.17 COM–017 absent 'DO' keyword in WHILE statement

Alarm ID	COM–017	Alarm Title	absent 'DO' keyword in WHILE statement
Description	In MACRO, WHILE loop uses DO to define task incorrectly.		

Possible Cause	MACRO programming error.
Solution	Check MACRO program to confirm if DO is correctly used in WHILE loop.

6.18 COM–018 absent 'END_WHILE' keyword in WHILE statement

Alarm ID	**COM–018**	**Alarm Title**	**absent 'END_WHILE' keyword in WHILE statement**
Description	In MACRO, WHILE loop doesn’t have END_WHILE to finish loop.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END_WHILE is used correctly to end the WHILE loop.		

6.19 COM–019 absent 'THEN' keyword in IF statement

Alarm ID	**COM–019**	**Alarm Title**	**absent 'THEN' keyword in IF statement**
Description	In MACRO, IF uses THEN to define task incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program to confirm if END is correctly used in IF loop.		

6.20 COM–020 absent 'END_IF' or 'ELSE' keyword in IF statement

Alarm ID	**COM–020**	**Alarm Title**	**absent 'END_IF' or 'ELSE' keyword in IF statement**
Description	In MACRO, IF loop doesn’t use ELSE or END_IF correctly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if ELSE or END_IF is used correctly in IF loop.		

6.21 COM–021 absent 'END_IF' keyword in IF statement

Alarm ID	COM–021	Alarm Title	absent 'END_IF' keyword in IF statement
Description	In MACRO, IF loop uses END_IF to finish loop incorrectly.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, END_IF is correctly used in IF loop.		

6.22 COM–022 absent 'OF' keyword in CASE statement

Alarm ID	COM–022	Alarm Title	absent 'OF' keyword in CASE statement
Description	In MACRO, OF is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if OF is used correctly in CASE command.		

6.23 COM–023 absent 'END_CASE' or 'ELSE' keyword in CASE statement

Alarm ID	COM–023	Alarm Title	absent 'END_CASE' or 'ELSE' keyword in CASE statement
Description	In MACRO program, ELSE or END_CASE is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, if ELSE or END_CASE is used correctly in CASE command.		

6.24 COM–024 absent 'END_CASE' keyword in CASE statement

Alarm ID	COM–024	Alarm Title	absent 'END_CASE' keyword in CASE statement
Description	In MACRO, END_CASE keyword is used incorrectly in CASE command.		

Possible Cause	MACRO programming error.
Solution	Check MACRO program, ensure that END_CASE is used before CASE statement ending.

6.25 COM–025 absent ':' or ',' delimiter in CASE statement

Alarm ID	**COM–025**	**Alarm Title**	**absent ':' or ',' delimiter in CASE statement**
Description	In MACRO program, "; " or ", " is used incorrectly in CASE command.		
Possible Cause	MACRO programming error.		
Solution	Check MACRO program, "; " or ", " is used correctly in CASE command, In CASE statement ending use "; ", and in comment use ", ".		

6.26 COM-026 Illegal MOVJ instruction input

Alarm ID	**COM-026**	**Alarm Title**	**Illegal MOVJ instruction input**
Description	Illegal MOVJ command input.		
Possible Cause	MOVJ command format error.		
Solution	Check if MOVJ command format is correct.		

6.27 COM-027 Invalid character

Alarm ID	**COM-027**	**Alarm Title**	**Invalid Character**
Description	Only ASCII characters are acceptable in NC files. Using non-ASCII characters is thus unacceptable. Note: Listed below are the special cases in which non-ASCII characters are considered acceptable: 1. Comment. 2. Arguments of MACRO functions that are of string type.		

Alarm ID	COM-027	Alarm Title	Invalid Character
Possible Cause	There are non-ASCII characters in the NC file. Common cases include: 1. Using full width semicolons instead of half width ones. 2. Writing comments without using comment characters. 3. Using full-width spaces instead of half width ones.		
Solution	Please check the machining program to make sure that only ASCII is used in comments and macros. Recommended to check: 1. Make sure that full-width semicolons is not used. 2. Make sure that all the program comments are added with comment syntax. 3. Make sure that full-width space is not used. Suggested exclusions: • Confirm the line number of the stopped processing file, delete and re-enter the processing file of this line on the controller.		

6.28 COM-028 Illegal use of '/'

Alarm ID	COM-028	Alarm Title	Illegal use of ' / '
Description	Incorrectly use single block optional skip function " / ".		
Possible Cause	MACRO programming error, function syntax or variable calculation is written after " / ".		
Solution	Check the incorrect line to ensure no function syntax or variable operation is written after " / ".		

SYNTEC

7 Program Execute Error Alarm - COR

Alarm ID	**COR–001** **BGND-001**	**Alarm title**	**Array Variable is empty**
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

Alarm ID	**COR–002** **BGND-002**	**Alarm title**	**File not exist**
Description	If the file that the system wants to read does not exist i.e. use M98 (or G65, G66, and etc.) to call a non existing file.		
Reason	Programming error.		
Solution	Check the NC program to make sure the existence of the file.		

Alarm ID	**COR–003** **BGND-003**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of MACRO is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		
Reason	Programming error		
Solution	Check the NC program to ensure that the denominator is not equal to 0.		

Alarm ID	**COR–004** **BGND-004**	**Alarm title**	**Operation domain error**
Description	Operation domain error.		
Reason	Programming error		

Solution	Please check the NC program.		
Alarm ID	**COR–005** **BGND-005**	**Alarm title**	**Program loading failure**
Description	MACRO syntax error.		
Reason	Programming error.		
Solution	Please check the NC program.		
Alarm ID	**COR–006** **BGND-006**	**Alarm title**	**Arc not on work plane**
Description	1. In clockwise, counter-clockwise arc cutting (G02, G03) syntax, if the vector from center of circle to starting point doesn't exist on the arc working surface. i.e. execute G17 G02 I50. K10. , and system will issue this alarm. 2. In spiral interpolation (G02, G03) syntax, when corresponding to G17, G18, and G19 three surface conditions, if K, J , and I arguments are not zero, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that G02 and G03 are used correctly.		
Alarm ID	**COR–007** **BGND-007**	**Alarm title**	**Arc radius too short**
Description	In G02 and G03 syntax, if Arc radius is smaller than 10 to the power of minus 10 BLU (10^-10), system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.		
Alarm ID	**COR–008** **BGND-008**	**Alarm title**	**Arc destination not on arc**

Description	In G02 and G03 syntax, if the Arc end point coordinate is not on the circle, system will issue this alarm. From 8.31 version, adding Pr3807 setting arc final point is not on the check window, allowing to set window range's error in Pr3807. When arc end point position's error is smaller than Pr3807 setting window's range, system will auto correct the circle's center position, and make the end point position to locate on the circle correctly. When arc end point position error is bigger than Pr3807 window range setting, system will issue this alarm.
Reason	Programming error.
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.

Alarm ID	**COR–009** **BGND-009**	**Alarm title**	**G65 layers called by Macro too deep**
Description	Use G65 to call MACRO program layer more than 12 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 12 layers.		

Alarm ID	**COR–010** **BGND-010**	**Alarm title**	**G66 layers called by Macro too deep**
Description	Use G66 to call MACRO program layer more than 4 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 4 layers.		

Alarm ID	**COR–011** **BGND-011**	**Alarm title**	**Subprogram call too deep**
Description	Use M98 to call subprogram that has more than 16 layers.		
Reason	Programming error.		

Solution	Check NC program to ensure that M98 calls subprogram that has less than 16 layers.		
Alarm ID	**COR–012** **BGND-012**	**Alarm title**	**G66 mode not cancel by G67**
Description	G66 and G67 need to be used in pairs, when numbers of G67 is more than G66 in one NC program, this alarm will be issued.		
Reason	Programming error.		
Solution	Check NC program to ensure that G66 and G67 are used in pairs		
Alarm ID	**COR–013** **BGND-013**	**Alarm title**	**G65, G66 must be at the end of the block**
Description	G65 and G66 are MACRO, so in one same block the program on the right hand side of G65 and G66 will be processed as G65 and G66's arguments. So in one same block, if there is other G code command please write them in the left hand side of G65 and G66. In one same block, if the right hand side of G65 and G66 has G code or M code, system will issue this alarm.		
Reason	Programming error.		
Solution	Please check the NC program.		
Alarm ID	**COR–014** **BGND-014**	**Alarm title**	**Absent program number**
Description	The right hand side of G65 and G66 doesn’t have P argument to specify program number.		
Reason	Programming error.		
Solution	Please check the NC program to ensure G65 and G66 use P argument to specify program number.		
Alarm ID	**COR–015** **BGND-015**	**Alarm title**	**Too many M code**
Description	There are more than 3 M codes in a single block.		

Reason	Programming error.		
Solution	Please check the NC program to ensure that there are equal or less than 3 M codes in a single block		
Alarm ID	**COR–016** **BGND-016**	**Alarm title**	**Illegal variable access**
Description	Attempted to access a variable that does not exist.		
Reason	1. Programming error. 2. Illegal access to # or @ variable. 3. The parameter of SYSDATA function is out of system data range. 4. The controller does not support the status variable which DRVDATA is to read. 5. The program that attempts to access AR or MAR variables does NOT lie within the scope of App Macros. 6. The AR or MAR variables to access do NOT lie within the range defined by the App. 7. Non-integer numbers are used to specify the addresses of AR or MAR variables. ONLY integers are acceptable.		
Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable. 4. Make sure the variable which DRVDATA is to read can be found on the "Controller Axis Info." page. 5. Make sure the AR and the MAR variables are accessed ONLY in App Macros. 6. Make sure the AR and the MAR variables to access lie within the range defined by the App. 7. Make sure only integers are used to specify the addresses of AR or MAR variables.		

Alarm ID	**COR–017** **BGND-017**	**Alarm title**	**Label not found**
Description	Cannot find out corresponding serial number N in GOTO command.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	**COR–018** **BGND-018**	**Alarm title**	**Line number not found**
Description	Input line number is incorrect.		

Reason	1. When Pr3851 sets as 888800, input breakpoint's line number exceeds NC program's maximum line number. 2. When Pr3851 sets as 999900, input breakpoint's line number exceeds NC program's maximum line number. 3. When Pr3851 sets as 999901, input breakpoint's line number exceeds NC program's maximum line number or positioning command line number cannot be found after scanning through entire NC program. 4. When M99 Q_ sub-program returns to the line number which is assigned by main program, the line number Q_ assigned is over main program's maximum line number.
Solution	1. Input the correct program line number. 2. When Pr3851 sets as 999901, modify NC program by inserting positioning command after initial assigned breakpoint.

Alarm ID	**COR–019** **BGND-019**	**Alarm title**	**sub-program has no M99**
Description	When main program calls sub-procedure, if sub-program finish executing and need to return the main program, there is no M99.		
Reason	Programming error.		
Solution	Write in M99 when sub-program finish executing and return the main program.		

Alarm ID	**COR–020** **BGND-020**	**Alarm title**	**Too many G code**
Description	There are more than 10 G codes in a single block.		
Reason	Programming error.		
Solution	Breakdown that single block which has over 10 G codes into single blocks that has less than 10 G codes.		

Alarm ID	**COR–021** **BGND-021**	**Alarm title**	**Too many (I,J,K) triples**
Description	Repeat too much I, J, and K command in the same block.		
Reason	Programming error.		

Solution	Please check the NC program.		
Alarm ID	**COR–022** **BGND-022**	**Alarm title**	**Use undefined workpiece coordinate**
Description	Use an undefined G54 coordinate system.		
Reason	Programming error.		
Solution	Use the correct G54 coordinate system.		
Alarm ID	**COR-023** **BGND-023**	**Alarm Title**	**Semantic error**
Description	1. When using G code, different G code can input different argument (P_, L_, R_, and etc.), if argument setting is wrong, and system will issue this alarm. 2. The argument of macro function is wrong.		
Reason	Programming error.		
Solution	Check the NC program to ensure the argument is used correctly by referring the manual.		
Alarm ID	**COR–024** **BGND-024**	**Alarm title**	**Invalid arc radius value**
Description	When executing G02, G03, appointed Arc end point and given radius is contradicted, which the given radius cannot meet appointing Arc end point. i.e. G91 G03 X1500 Y4000 R2000.		
Reason	Programming error.		
Solution	Check the program and recalculate.		
Alarm ID	**COR–025** **BGND-025**	**Alarm title**	**Macro stack is overflow or STKTOP[ ] argument error**

Description	1. STACK can store maximum 4095 values. If stored value exceeding the maximum number, controller issue this alarm. 2. In STKTOP[n], n is started from 0, if the value of n is bigger than the value storing in stack-1, controller issue this alarm
Reason	1. Too much value store in STACK. 2. STKTOP[] arguments exceeds the value storing in STACK.
Solution	1.Stack is full, do not use store command anymore. 2. Input a reasonable argument in STKTOP[].

Alarm ID	**COR–026** **BGND-026**	**Alarm title**	**macro stack is empty**
Description	Empty stack still want to pop value, system issue this alarm.		
Reason	The numbers of Push commands and Pop commands are not the same.		
Solution	Check the program to ensure that the number of Push commands is the same with that of Pop commands.		

Alarm ID	**COR–027** **BGND-027**	**Alarm title**	**Invalid macro arguments**
Description	Macro issue self-defined alarm.		
Possible Cause	Once Macro determines the self-defined alarm error condition is meet, NC program will be stopped and alarm will appear.		
Solution	According to the display content of alarm to find out the error.		

Alarm ID	**COR–028** **BGND-028**	**Alarm title**	**System program error, can’t normally machining.**
Description	Use Quiet Mode in MACRO, but can't leave Quiet Mode when program finished.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–029 BGND-029	Alarm title	Tool length offset change at arc
Description	G43, G44, G49 only receive linear interpolation command in the next block.		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR-030 BGND-030	Alarm Title	Cutting speed command is 0
Description	When execute cutting command, given F code argument is zero.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the argument of F code shall not be 0.		

Alarm ID	COR–031 BGND-031	Alarm title	Radius compensation cancel at arc
Description	G40 only receives linear interpolation command in the next block		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–032 BGND-032	Alarm title	Radius compensation activate at arc
Description	G41,G42 only receive linear interpolation command in the next block.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–033 BGND-033	Alarm title	Improper use of A, R, or C command

Description	Timing of using A, R, or C command is incompatible with the specification.
Reason	Programming error.
Solution	Check NC program to confirm that if the blocks are compatible with specifications

Alarm ID	**COR–034** **BGND-034**	**Alarm title**	**Command syntax has conflict**
Description	No application currently.		
Reason			
Solution			

Alarm ID	**COR–035** **BGND-035**	**Alarm title**	**Corner is too small can’t insert round corner chamfer**
Description	Angel between blocks, which to be inserted round corner or chamfer is too small so system can’t calculate it.		
Reason	Programming error.		
Solution	1. Check NC program to confirm that whether the block is compatible with specifications. 2. Check and confirm the working plane and round corner plane are the same.		

Alarm ID	**COR–036** **BGND-036**	**Alarm title**	**Inappropriate A angle command**
Description	A is valid only in linear interpolation of single block.		
Reason	Programming error.		
Solution	Check NC program to confirm that whether A angle command is compatible with specifications		

Alarm ID	**COR–037** **BGND-037**	**Alarm title**	**Chamfer value bigger than displacement**

Description	Chamfer value is bigger than pre-block and post-block length, so system can't calculate it.		
Reason	Programming errors.		
Solution	1. Check NC program to confirm that whether inserted value of chamfer is compatible with specifications. 2. Check to confirm whether C value is compatible with specification.		
Alarm ID	**COR–038** **BGND-038**	**Alarm title**	**Incorrect block jump's switch number.**
Description	Incorrect block jump switch number. i.e. / 2 G00 X100. ; If assigned number is bigger than 9 or equals to 0, system will issue this alarm.		
Reason	Block jump switch number is 0 or bigger than 9.		
Solution	Check the NC program, and confirm block jump switch number is between 1 and 9.		

Alarm ID	**COR–039** **BGND-039**	**Alarm title**	**Measure function can't start tool radius compensation**
Description	Measure function can't start tool radius compensation.		
Reason	Measure function related instruction (i.e., G31, G31.10, G31.11) is executed after tool compensation command.		
Solution	Confirm there is no measure function related command is executed after tool compensation command.		

Alarm ID	**COR–040** **BGND-040**	**Alarm title**	**Block end point exceed software stroke limit**
Description	The machine coordinate in the program exceeds the software stroke limit.		
Reason	Programing error.		
Solution	Check the NC program, and correct coordinate position.		

Alarm ID	**COR–041** **BGND-041**	**Alarm title**	**GOTO label must be integer**
Description	The input GOTO label is not an integer. i.e. GOTO 1 Correct GOTO 1. Wrong N1; Correct N1.; Wrong		
Reason	Programming error.		
Solution	Check the NC program, and input integer in GOTO argument.		

Alarm ID	**COR–042** **BGND-042**	**Alarm title**	**Logic operand is non-integer or empty**
Description	Logic operand must be non-integer or empty.		
Reason	Logic operand has floating point. i.e. #1=1.5And3. System will issue this alarm.		
Solution	Please check NC program. Confirm logic operand is integer or empty.		

Alarm ID	**COR–043** **BGND-043**	**Alarm title**	**ASIN, ACOS operand must between ±1.0**
Description	ASIN() and ACOS() operand is not between -1.0 and 1.0.		
Reason	Programming error.		
Solution	Check the NC program.		

Alarm ID	**COR–044** **BGND-044**	**Alarm title**	**SQRT operand should not be negative**
Description	The square root of a negative value will be imaginary number, but the controller does not provide imaginary number function.		

Reason	Programming error.		
Solution	Check the NC program; enter a positive value in SQRT operand.		
Alarm ID	**COR–045** **BGND-045**	**Alarm title**	**L address should be integer**
Description	The L address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in L address.		
Alarm ID	**COR–046** **BGND-046**	**Alarm title**	**O address should be integer**
Description	The O address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in O address.		
Alarm ID	**COR–047** **BGND-047**	**Alarm title**	**M address should be integer**
Description	The M address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in M address.		
Alarm ID	**COR–048** **BGND-048**	**Alarm title**	**Spindle speed S should be integer**
Description	The spindle speed S is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in the spindle speed S.		

Alarm ID	**COR–049** **BGND-049**	**Alarm title**	**Tool length compensation H should be integer**
Description	Tool length compensation H is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool length compensation H.		
Alarm ID	**COR–050** **BGND-050**	**Alarm title**	**Tool radius compensation D should be integer**
Description	Tool radius compensation D is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool radius compensation.		
Alarm ID	**COR–051** **BGND-051**	**Alarm title**	**Tool number T should be integer**
Description	Tool number T is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use tool number T in integer.		
Alarm ID	**COR–052** **BGND-052**	**Alarm title**	**Sub-program number, P, should be integer**
Description	Sub-program number P is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use the sub-program number P in integer.		
Alarm ID	**COR–053** **BGND-053**	**Alarm title**	**Repeat count L should be integer**

Description	Repeat count L is not an integer.
Reason	Programming error.
Solution	Please check the NC program, and use the repeat count L in integer.

Alarm ID	**COR–054** **BGND-054**	**Alarm title**	**Incompatible data type**
Description	Data format is incompatible with controller specifications.		
Reason	NC program is not compatible with the SYNTEC controller.		
Solution	Make sure that the NC program data format is compatible with Syntec controller specifications.		

Alarm ID	**COR–055** **BGND-055**	**Alarm title**	**Tool length compensation H out of range**
Description	The tool length compensation H exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool length compensation H, is in the range of tool number.		

Alarm ID	**COR–056** **BGND-056**	**Alarm title**	**G10 table index P is out of range**
Description	G10's format is G10 L_ P_ R_; Different number L will correspond to different number P. i.e. L10 corresponding number P is tool number. Input P1000 means the 1000th tool. If that tool number doesn't exist, and controller will issue this alarm.		
Reason	Programming error.		
Solution	Confirm G10 data table address number P is in reasonable range.		

Alarm ID	**COR–057** **BGND-057**	**Alarm title**	**Tool radius D out of range**

Description	The tool radius number D exceeds the range of tool number.
Reason	Programming error.
Solution	Make sure that the tool radius number D is in the range of tool number.

Alarm ID	**COR–058** **BGND-058**	**Alarm title**	**Tool nose compensation D is out of range**
Description	The tool nose compensation D exceeds the range of tool number.		
Reason	Program error.		
Solution	Make sure that the tool radius compensation D is in the range of tool number.		

Alarm ID	**COR–059** **BGND-059**	**Alarm title**	**Subprogram Call H must be integer**
Description	Subprogram call H is not an integer.		
Reason	Program error.		
Solution	Confirm the subprogram call H is an integer.		

Alarm ID	**COR–060** **BGND-060**	**Alarm title**	**M99 return number P must integer**
Description	The M99 return sequence number P is not an integer.		
Reason	Program error.		
Solution	Confirm the M99 return sequence number P is an integer.		

Alarm ID	**COR–061** **BGND-061**	**Alarm title**	**Workpiece number is out of range**
Description	The number of work pieces exceeds the allowable range of the controller.		
Reason	Program error.		

Solution	Make sure the number of work pieces is in the allowed range of the controller.

Alarm ID	**COR–062** **BGND-062**	**Alarm title**	**Dwell skip source Q must be integer**
Description	Dwell skip source Q is not an integer.		
Reason	Program error.		
Solution	Change the dwell skip source Q into an integer.		

Alarm ID	**COR–063** **BGND-063**	**Alarm title**	**Dwell skip source Q out of range**
Description	Dwell skip source Q exceeds the allowed range.		
Reason	Program error.		
Solution	Make sure the dwell skip source Q is in the allowed range.		

Alarm ID	**COR–064** **BGND-064**	**Alarm title**	**Invalid P address**
Description	Invalid P address.		
Reason	1. Address P is less than 0. 2. Address P is not integer. 3. Address P is out of range.		
Solution	1. Make sure address P is greater or equal to 0. 2. Make sure address P is integer. 3. Make sure address P is within the range.		

Alarm ID	**COR–065** **BGND-065**	**Alarm title**	**,A command format error**
Description	In NC program ,A command's format is wrong.		
Reason	Programming error.		

Solution	Please confirm ,A command format is one of the followings: 1. In the same block command as ,A, Moving instruction only includes one of x axis or z axis. i.e. G01 X10. ,A30. 2. Use ,A advanced function in two separate line. The first line will assign ,A angle, and the second line will assign X, Z and ,A at the same time. i.e. G01 ,A30. G01 X0. Z50. ,A45

Alarm ID	**COR–066** **BGND-066**	**Alarm title**	**Inc. axis command and abs. axis command conflict**
Description	Both G91 and G90 are in the same line.		
Reason	Programming error.		
Solution	Decide to use incremental or absolute command in one line, and enter the correct command.		

Alarm ID	**COR–067** **BGND-067**	**Alarm title**	**Arc center vector and radius conflict**
Description	The arc end point is not on the arc created by the arc starting point and the specify center.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	**COR–068** **BGND-068**	**Alarm title**	**Quiet Mode not support imperial Metric switch command**
Description	The single block command in Quiet mode operation cannot switch Metric/ imperial unit.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–069 BGND-069	Alarm title	Round corner and chamfer cmd. conflict
Description	Chamfer command and round corner command are in the same line.		
Reason	Program error.		
Solution	Do not let chamfer command and round corner command exist in the same line.		

Alarm ID	COR–070 BGND-070	Alarm title	Invalid G Code
Description	Enter incorrect G code to controller.		
Reason	Program error.		
Solution	Enter the valid G-code.		

Alarm ID	COR–071 BGND-071	Alarm title	No main program name assigned
Description	The name of main program is not specified.		
Reason	The NC program is not loaded.		
Solution	Specify the name of main program.		

Alarm ID	COR–072 BGND-072	Alarm title	Threading command exceeds max. cutting speed
Description	Threading command exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of threading.		

Alarm ID	COR–073 BGND-073	Alarm title	Tapping command exceed max. cutting speed

Description	Tapping command speed exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of tapping.		
Alarm ID	**COR–074** **BGND-074**	**Alarm title**	**Tool radius too big, path overcut**
Description	1. During machining notch, the notch width is smaller than two times of tool radius. 2. During machining trapezoidal, the height of trapezoidal is less than two times of tool radius.		
Reason	Tool compensation causes path overcut.		
Solution	Please check the NC program, and decide whether this part of the machining should cancels tool radius compensation.		
Alarm ID	**COR–075** **BGND-075**	**Alarm title**	**Exact stop wait timeout**
Description	After 2 seconds after sending Exact stop (G09/G61) command, the difference between feedback and command exceeds allowable value (G01 is determined by Pr421~; G00 is determined by Pr481~).		
Reason	Servo vibration.		
Solution	1. Servo tuning. 2. Modify parameters Pr421~ or Pr481~.		
Alarm ID	**COR–076** **BGND-076**	**Alarm title**	**G04 dwell time cannot be negative**
Description	Input value of dwell time G04 is negative.		
Reason	Program error.		
Solution	Check the NC program, and enter a positive or zero value in G04 argument.		

Alarm ID	COR-077 BGND-077	Alarm Title	NURBS curve format is wrong
Description	G6.2 format is wrong.		
Reason	Input argument P, K, R is against allowable format.		
Solution	Refer the program manual to correct the command syntax.		

Alarm ID	COR-078 BGND-078	Alarm Title	NURBS curve system memory insufficient
Description	NURBS curve memory is insufficient.		
Reason	System processing defective.		
Solution	Contact OEM Syntec.		

Alarm ID	COR-079 BGND-079	Alarm Title	Number of Nurbs curve nodes more than the upper limit
Description	NURBS curve control point is over limit.		
Reason	NC file G6.2 curve is too long and cause using too many control point.		
Solution	Re-generate the CAM, and restrict the number of curve nodes less than 400.		

Alarm ID	COR-080 BGND-080	Alarm Title	Threading pitch is negative
Description	Threading pitch distance can’t be negative value.		
Reason	Threading pitch distance setting error.		
Solution	Check and correct the configured threading pitch to be positive value.		

Alarm ID	**COR-081** **BGND-081**	**Alarm Title**	**Use arc interpolation under tool offset function. Arc command doesn't use assigned angle.**
Description	In tool offset function (G45~G48), G02 and G03 can only be 90 degree or 270 degree.		
Reason	In NC program's tool offset function (G45~G48) block, all arc interpolation angle can only be 90 degree or 270 degree.		
Solution	Check if G45~G48 is 90 or 270 degree when using them in the arc interpolation (G02/G03). Note, IJK to specify circle center shall be used.		

Alarm ID	**COR-082** **BGND-082**	**Alarm Title**	**Tool compensation function and tool offset function can't use at the same time**
Description	Can't use tool offset function (G45~G47) and tool compensation function (G41, G42) or tool radius compensation at the same time.		
Reason	NC program edit error.		
Solution	Check NC program, and confirm tool offset function (G45~G47) and tool compensation function (G41, G42) are not used at the same time.		

Alarm ID	**COR-083** **BGND-083**	**Alarm Title**	**When pull tool back to pause point, and move axis position manually.**
Description	Before pull tool back to pause point, press stop and manually move tool.		
Reason	Operation error.		
Solution	Please wait until tool is pulled exactly back to pause point, then execute the manual position movement.		

Alarm ID	**COR-084** **BGND-084**	**Alarm Title**	**P Argument unassigned when using the rapid drilling**
Description	When using rapid drilling function, must set P argument (bottom hole dwell time), or system will issue this alarm.		

Alarm ID	**COR-084** **BGND-084**	**Alarm Title**	**P Argument unassigned when using the rapid drilling**
Reason	Programming error.		
Solution	Check the NC program to ensure the P Argument is assigned when using the G code of rapid drilling.		

Alarm ID	**COR-085** **BGND-085**	**Alarm Title**	**F Argument unassigned in Inverse Time Feed mode**
Description	Under Inverse Time Feed mode, if F argument is not assigned in command line and system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure there is F command is given following in each line after Inverse Time Feed mode.		

Alarm ID	**COR-086** **BGND-086**	**Alarm Title**	**R and I arguments unassigned in G10 L1501 mode**
Description	G10 L1501 argument only has argument R or argument I.		
Reason	Programming error.		
Solution	Check the NC program to ensure the R and I arguments are not used independently.		

Alarm ID	**COR-087** **BGND-087**	**Alarm Title**	**P and Q arguments must be different in G10 L1501 mode**
Description	P and Q argument in G10 L1501 mode setting are the same.		
Reason	Programming error.		
Solution	Please check program, and confirm P and Q argument in G10 L1501 mode aren’t the same.		

Alarm ID	COR-088 BGND-088	Alarm Title	No time-axis command after the I argument in G10 L1501 mode
Description	In spring machine dedicated machine's motion plan, if use I argument in G10 L1501, before the next G10 L1502 command, user can’t command P argument axis number (time axis) in G1 0L1501.		
Reason	Programming error.		
Solution	Check the NC program.		

Alarm ID	COR-089 BGND-089	Alarm Title	Main axis must be increment command in G10L1501 mode
Description	G10 L1501 mode spindle must be incremental command.		
Reason	Programming error.		
Solution	Under G10 L1501 mode, confirm the command which sends to spindle is incremental command.		

Alarm ID	COR-090 BGND-090	Alarm Title	C-axis zero-speed check failed in G12.1 mode
Description	When start to switch polar coordinate, C axis must be totally stop.		
Reason	When C axis uses as spindle, but spindle not yet stop rotating.		
Solution	Switch the spindle to C-axis mode before activating polar coordinate.		

Alarm ID	COR-091 BGND-091	Alarm Title	Main axis synchronization – basic main axis number error
Description	When using spindle synchronous function G114.1, Pr4021, Pr4023, Pr4025 can’t be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4021/Pr4023/Pr4025 are correct.		

Alarm ID	COR-092 BGND-092	Alarm Title	Main axis synchronization – synchronization main axis number error
Description	When using spindle synchronous function G114.1, Pr4022, Pr4024, Pr4026 can't be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4022/Pr4024/Pr4026 are correct.		

Alarm ID	COR-093 BGND-093	Alarm Title	Main axis synchronization – spindle type error
Description	When using spindle synchronous and Superimposition function, setting wrong spindle motor type.		
Reason	Parameter setting error.		
Solution	Check 1791~Pr1800 to set correct spindle configurations by referring to HELP.		

Alarm ID	COR-094 BGND-094	Alarm Title	Under spindle load rigid tapping spindle rotation speed exceed
Description	When using spindle load function to do rigid tapping, tool axis must synchronize with offset axis speed, and then do end surface taping according to user setting tapping speed. If tool axis speed is over spindle rotation speed maximum value, and system will issue this alarm.		
Reason	Command is over range that mechanical can bear.		
Solution	1. Reduce the rotation speed (tapping speed) of tool axis. 2. Reduce the rotation speed of workpiece axis.		

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (P argument) must be bigger than zero.		
Reason	Programming error.		

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Solution	Check the value of P argument in G51.2.		

Alarm ID	COR-096 BGND-096	Alarm Title	In polygon cutting function, synchronous spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (Q argument) must be bigger than zero.		
Reason	Programming error.		
Solution	Check the value of Q argument in G51.2.		

Alarm ID	COR-097 BGND-097	Alarm Title	Axis coupling function ON or OFF failed
Description	Axis coupling function ON or OFF failed.		
Reason	Too much axis coupling groups is used.		
Solution	Check if the number of axis coupling groups is more than 16 (including the number of axis coupling group configured in parameter).		

Alarm ID	COR-098 BGND-098	Alarm Title	Parameter learning argument error
Description	When using learning function, if argument (P, Q, R, K) setting is wrong, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure the arguments are in compliance with specifications.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

Alarm ID	COR-099 BGND-099	Alarm Title	Tapping learning condition is not matched
Description	Condition of tapping learning function is not matched		
Reason	When activating tapping learning function, if tapping condition (tapping depth, R point height, feedrate, rotation speed, and etc.) is different, it can't apply to same learning data. If force different tapping condition to use same learning data, it may misuse learning data, and cause tapping error or even tool break.		
Solution	Check tapping instruction in tapping learning function, and confirm all tapping instruction arguments are the same.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

Alarm ID	COR-100 BGND-100	Alarm Title	Unsupported G code command or option software is not activated
Description	Different controllers will have correspond G code, but not all G code can use.		
Reason	1. This controller type may not support this G code command. 2. This controller type will not support serial bus spindle (C-Type) to use lathe G21, G33, G34, G78 commands. 3. This controller type will not support serial bus spindle (A-Type) to use lathe G32, G73, G76, G92 commands. 4. This controller type can support this G code command, but the option software function has not been purchased, which makes the G code unusable. 5. Loader path and Woodworking label path only support part of G codes: G00, G01, G02, G03, G04, G04.1, G09, G10, G17, G18, G19, G22, G23, G31, G52, G53, G54, G55, G56, G57, G58, G59, G59.x, G90, G91, G92. 6. The setting of Pr3802 is incorrect. This controller type does not support the G62 command.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Description	When using spindle synchronize, if basic spindle and synchronous spindle pitch setting are different and will cause synchronous abnormal condition. And will issue this alarm for protection.		

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Possible Cause	Parameter setting error.		
Solution	Check the spindle used in Pr161~180 to ensure the parameter configurations are the same.		

Alarm ID	COR-102 BGND-102	Alarm Title	Under Spindle synchronization – synchronization command duplicated or conflicted
Description	When spindle synchronizing, repeat G51.2, G114.1 or G114.3 commands.		
Possible Cause	Haven't use G113 to cancel G114.1 and G114.3, or use G50.2 to cancel G51.2, and again set a repeated G114.1, G114.3 or G51.2 commands.		
Solution	Cancel the G114.1 and G114.3 mode with G113, or cancel G51.2 mode with G50.2. Then execute the G114.1/G114.3/G51.2 commands to avoid the angle difference for repeated machining.		

Alarm ID	COR-103 BGND-103	Alarm Title	Invalid precision channel setting
Description	Invalid precision channel setting.		
Possible Cause	Using G64, G62P_ or G120.1 P_Q_ to choose invalid precision channel. Currently can only choose: 1. G64、G62 P0~P9、G62/G64 P21 ~ P23 2. G120.1 P0、G120.1 P1 Q1 ~ G120.1 P3 Q3		
Solution	To select precision channel with the G64/G62 P_ or G120.1 P_ Q_. Check the NC program to ensure if any non-existing precision channel is selected.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Description	The virtual axis function failed to be functioned.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Possible Cause	1. The P and Q arguments in G10 L800 P_ Q_ or G10 L801 P_ Q_ are set incorrectly. a. No P, Q arguments. b. The P argument is set incorrectly and the value is invalid. c. The Q argument is set incorrectly, the value is invalid; or the corresponding axis is not set to the axis of the parameter table that has been turned on. 2. In the same machining, G10 L800 and G10 L801 appear at the same time. 3. The robot does not support the virtual axis function.		
Solution	1. Check the virtual axis commands to ensure the P and Q Arguments are configured. 2. Check the virtual axis commands to ensure the P Argument shall be within 100~999 and the Q Argument shall be within 1~16 (axis number) or 100~999 (axis name). 3. Check the axial direction (Q Argument) mapped with the virtual axis to ensure the parameter table is configured to the enabled axial direction. 4. Check if the G10 L800 and G10 L801 shall not be existent in the same NC program.		

Alarm ID	COR-105 BGND-105	Alarm Title	Axial coupling configuration error
Description	The axis coupling function is set incorrectly.		
Possible Cause	In G10 L900 P_ Q_ R_, the P and Q arguments are set incorrectly.		
Solution	1. Check the name or number of coupling axis is mapped to ensure the axial direction is enabled. 2. Chedk the specified coupling axis is not assigned to inclined axes control. 3. Check the specified coupling axis is not assigned to indexing axis.		

SYNTEC

Alarm ID	COR-106 BGND-106	Alarm Title	Prohibit to use G5.1 in G61/G63/ G63.2 mode
Description	1. G5.1 path smoothing. In G61, G63, G63.2 mode, it is forbidden to use G5.1 smoothing function, otherwise the system will issue an alarm; If G61, G63, G63.2 are enabled in G5.1 mode, the system will stop the smoothing function until it jumps out of G61, G63, G63.2 mode and then activate again. 2. G05 High Precision Contour Control. a. 10.116.16J previous version, in G61, G63, G63.2 mode, it is forbidden to use G05 High Precision Contour Control function, otherwise the system will issue an alarm; If G61, G63, G63.2 is enabled in G05 mode, no alarm will be issued, but the High Precision Contour Control function is disabled, It is necessary to command G05 again to activate. b. above 10.116.16K versions, no alarm will be issued, and the G05 High Precision Contour Control function will be disabled, and can be valid after G61, G63 and G63.2.		
Possible Cause	The cutting mode setting in the NC program is incorrect.		
Solution	1. Do not mix use G5.1 path smoothing with G61/G63/G63.2 functions. 2. In the version before 10.116.16J (included), do not mix use the G05 High Precision Contour Control with the G61/G63/G63.2 functions.		

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Description	The G5.1 and G05 commands are in the wrong format.		
Possible Cause	1. The format of the G5.1 path smoothing command in the NC program is incorrect. 2. The G05 high-precision cutting mode command format in the NC program is incorrect.		

SYNTEC

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Solution	Confirm the following command formats are correct not have these error: 1. G5.1 a. Q argument: None, more than 2, or less than 0. b. E argument: None or less than 0. 2. G05 a. System issue alarm when using G05 in following cases for each version: i. G05 P argument is not 10000 nor 0. ii. G05 E argument is not positive. b. Activate command G05 P10000 X0 Y0 Z0 α_ β_ in 10.116.36 or above versions: i. More than 5 axial directions are assigned. ii. The geometry axis argument not 0. iii. The rotary axis argument is configured to 0. iv. The axial direction of geometry axis is configured but this of rotary axis is not. v. The axis of the rotation axis is not set when the axis of the geometry axis is not set. vi. More than 2 axial directions of rotation axes are configured. vii. Any axial arguments is negative. c. In the version before 10.116.16B, there is the 4th axis command in addition to the block movement commands of X, Y, or Z axes after G05 is executed.		

Alarm ID	COR-108 BGND-108	Alarm Title	G10 L1501/L1502 command format error
Description	There is no NC program between G10 L1501 and G10 L1502.		
Possible Cause	The NC program was written incorrectly.		
Solution	Check the NC program to ensure the programming between G10 L1501 and G10 L1502 is correct.		

Alarm ID	COR-109 BGND-109	Alarm Title	The second software stroke limit command error, activating failure
Description	The second software stroke limit (G22) command is written incorrectly, causing the second software stroke limit fail to activate.		
Possible Cause	The arguments written after the second software stroke limit (G22) command are wrong, and the same group parameters should exist at the same time.		
Solution	Check if the arguments in the same set after G22 are defined completely.		

Alarm ID	COR-110 BGND-110	Alarm Title	NC program of spring machine motion plan more than 500 lines
Description	The number of machining lines between G10 L1501 and G10 L1502 exceeds 500 lines.		
Possible Cause	There are too many single blocks in the NC program.		
Solution	Check and refine the NC program.		

Alarm ID	COR-111 BGND-111	Alarm Title	Axis exchange function – axial configuration error
Description	Under the axis exchange function, the axial setting is incorrect.		
Possible Cause	The two axial's axis exchange parameters are set incorrectly.		
Solution	Check if Pr3721 and Pr3722 are mapped to physical axial directions (Pr21~) and to 2 different axis.		

Alarm ID	COR-112 BGND-112	Alarm Title	Axis exchange function – diameter/radius axis configuration error
Description	Under the axis exchange function, the diameter axis and radius axis are set incorrectly.		
Possible Cause	The parameters of the diameter and radius axis of the two axis exchange are set incorrectly.		
Solution	Check if the diameter and radius axis configurations (Pr281~) of the 2 axis to be exchanged is consistent.		

Alarm ID	COR-113 BGND-113	Alarm Title	Axis exchange function – ON or OFF timing error
Description	When the axis exchange function is enabled, the enabled or disabled timing error.		
Possible Cause	When the cross-path axis is exchanged, the G04.1 waiting action is not performed.		
Solution	Check PLC to ensure no G04.1 waiting command is executed in more than 1 path.		

Alarm ID	COR-114 BGND-114	Alarm Title	Axis exchange function – path configuration error
Description	When using the axis exchange function, the path setting is incorrect.		
Possible Cause	The axis used for axis exchange belong to multi-paths.		
Solution	Check path parameter (Pr701~) if the two axis to be exchanged are mapped to the multiple path.		

Alarm ID	COR-115 BGND-115	Alarm Title	G92.1 axial configuration error
Description	When using the G92.1 rotation function, the axial setting is incorrect.		
Possible Cause	The parameter sets the first 3 axis of the path contains the rotary axis.		
Solution	Check the parameter to ensure the first 3 axis are linear axis.		

Alarm ID	COR-116 BGND-116	Alarm Title	Absolute coordinate system synchronization failed
Description	Absolute coordinate system failed to synchronize when using C35.		
Possible Cause	Machine is not stationary or the program is continuously interpreted.		
Solution	To execute the synchronization of coordinate system in C35 mode shall ensure the machine is still and the program stops to interpret (i.e. Using M code shall turn C38 off).		

Alarm ID	COR-117 BGND-117	Alarm Title	M code not disabled in the interrupt subprogram
Description	If Pr3600 is set to n, when the interrupt type subprogram is used, it starts with M(n) and ends with M(n+1). If M(n) is used and M(n+1) is not used to end the function, it will cause an action error; therefore, this alarm is issued for protection.		
Possible Cause	The NC programming error.		
Solution	Check the NC program to ensure the M(n+1) command is given.		

Alarm ID	COR-118 BGND-118	Alarm Title	Prohibit G53 commands in tool tip control mode
Description	G53 command cannot be used in the tool point control mode.		
Possible Cause	1. The NC programming error. 2. The machine type is the tool point control mode.		
Solution	1. Please check the NC program, make sure that the G53 command is not within the validity range of G43.4 or G43.5. 2. Please check the NC program, make sure that the G53 command is not within the validity of G12.1. 3. If the machine configuration used is the tool point control mode, the G53 command cannot be used.		

Alarm ID	COR-119 BGND-119	Alarm Title	G10 L16 command format error
Description	Since the command G10 L16 (virtual circle radius) does not support functions such as G05, G7.1, G12.1, G93, G95, and G05, therefore, an alarm will be issued if G10 L16 is used when the above command is issued.		
Possible Cause	1. The axis type is linear axis (i.e. G10 L16 X50., and X is set to a linear axis, this alarm is issued). 2. When using G10 L16 while command G05, G7.1, G12.1, G93, G95 and etc. functions are also given.		
Solution	1. Check Pr221~ to ensure the correct axis type. 2. G10 L16 and G7.1/G12.1/G93/G95/G05 shall not be executed at the same time.		

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Description	Different controller products can correspond to different maximum number of simultaneous cutting axis, and if this limit is exceeded, an alarm will be issued. (i.e. G01 X10. Y10 .Z10. Represents the number of simultaneous axes is 3).		
Possible Cause	Please refer to "System Information" under the controller HMI screen to check "Machine Properties" and "Machine Code". Please refer to the catalogue for each product specification. Following examples: 1. If "Machine Properties" is Mill, "Machine Code" is 200A-5, and the maximum number of axis is 9. 2. If the "Machine Properties" is Lathe, the "Machine Code" is 6B, and the maximum number of axis is 4.		

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Solution	The number of simultaneous axis in the NC program shall be according to the product specifications.		

Alarm ID	COR-121 BGND-121	Alarm Title	LN operator shall be positive
Description	The operands after the LN function cannot be negative.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Description	The base operand of the POW function must not be a negative number.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-123 BGND-123	Alarm Title	Illegal STR2INT input or too long string
Description	The syntax of the STR2INT function is incorrect or the string is too long.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Description	This model does not support S code commands.		
Possible Cause	This model does not support S code commands.		

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Solution	Replace a controller model that supports S codes.		

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Description	T code command is illegal.		
Possible Cause	1. This model does not support T code commands. 2. The T code command is out of the T code range supported by the product.		
Solution	Replace a controller model that supports T codes.		

Alarm ID	COR-126 BGND-126	Alarm Title	H code commands unsupported in the machine
Description	This model does not support H code commands.		
Possible Cause	This model does not support H code commands.		
Solution	Replace a controller model that supports H codes.		

Alarm ID	COR-127 BGND-127	Alarm Title	D code commands unsupported in the machine
Description	This model does not support D code commands		
Possible Cause	This model does not support D code commands		
Solution	Replace a controller model that supports D codes.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Description	The total number of M codes and T codes in the same block exceeds 20.		
Possible Cause	NC programming error.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Solution	Check the NC program to ensure the total number of M and T codes are not more than 20 in one single block.		

Alarm ID	COR-132 BGND-132	Alarm Title	Illegal character in the program name
Description	When open a file using Macro, the specified file name contains illegal characters.		
Possible Cause	NC programming error.		
Solution	Check the specified file names.		

Alarm ID	COR-133 BGND-133	Alarm Title	Command unsupported in Three-points arc interpolation mode
Description	This command is not supported in the three-point arc interpolation mode (G02.4, G03.4)		
Possible Cause	1. The tool radius compensation function is not turned off before using this function. 2. It is not supported in G62 cutting mode. 3. In the three-point arc interpolation mode (G02.4, G03.4), A, C, and R commands are not supported.		
Solution	Check the NC program to ensure there is no command described in the Possible Cause.		

Alarm ID	COR-134 BGND-134	Alarm Title	Three-points arc interpolation command format error
Description	The three-point arc interpolation (G02.4, G03.4) command format is wrong.		
Possible Cause	The three-point arc interpolation (G02.4, G03.4) command is regarded as a group of two lines, which can be specified continuously. The end point of the previous arc is the starting point of the next arc, but the F command can only be in the odd line. This alarm is issued if the total number of lines in the command is odd or the number of lines in the F command is even.		
Solution	Check the three-point arc interpolation G02.4/G03.4 command format in NC program.		

Alarm ID	COR-135 BGND-135	Alarm Title	Read/write command format error for R value
Description	SETRREGBIT, READRREGBIT bring in an argument with type error or range error.		
Possible Cause	1. SETRREGBIT (R value number, specify BIT, on or off) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31. c. If the third argument is not 0 (off) or 1 (on). 2. READRREGBIT (R value number, specify BIT) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31.		
Solution	Correct the argument type or range in the commands, SETRREGBIT and READRREGBIT		

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Description	After a certain serial bus axial tuning fails, without rebooting, cycle start is triggered.		
Possible Cause	The serial bus axial tuning failed.		
Solution	Reboot controller.		

Alarm ID	COR–137 BGND-137	Alarm title	Path synchronization waiting's P-argument sequence error
Description	The machining multi-path program uses the path synchronization waiting (G04.1) to wait for the synchronization action: 1. When the Q argument is not selected, when two programs go to the position of the path synchronization waiting (G04.1) at the same time, if the P arguments substituted are different, this alarm is issued for reminder. 2. When the next Q argument (decimal) specifies the path waiting for each other, when the two programs go to the position of the path synchronization waiting (G04.1) at the same time, and wait for each other, but the P arguments substituted are different, this alarm is issued for reminder.		

Possible Cause	1. In the NC program of multiple paths, the order of P arguments is abnormal or the number is incorrect. 2. Machining is performed using M99, but in each program, the path without Q argument is synchronously waiting (G04.1), and the number is different. 3. Machining is performed using M99, but in each program, the path with the same Q argument is synchronously waiting (G04.1), and the number is different. 4. The two paths “wait for each other”, but the P argument substituted are not the same. i.e. G04.1 P1 Q123 under the first path and G04.1 P2 Q124 under the second path.
Solution	Please check the number of Synchronization wait between paths (G04.1) and the order of the assigned value for argument P.

Alarm ID	**COR-138** **BGND-138**	**Alarm Title**	**Read/write command format error at the I/O/A point**
Description	The system provides 512 I/O/A points, the input I/O/A point number shall be within 0~511 when using the commands, SETDO, SETABIT, READDI, READDO, and READABIT.		
Possible Cause	NC Programming error.		
Solution	Check if there is any of I/O/A point number in the commands, SETDO, SETABIT, READDI, READDO, and READABIT in the NC program is over the range of 0 ~ 511.		

Alarm ID	**COR-139** **BGND-139**	**Alarm Title**	**Polynomial solution error**
Description	Determine the polynomial cannot be derived to solutions in case of the alarm.		
Possible Cause	Numerical processing calculation errors cause root failure.		
Solution	Check the value of system data no. 321, and provide it to the controller OEM Syntec for solutions.		

Alarm ID	**COR-140** **BGND-140**	**Alarm Title**	**Invalid high-precision contour control mode using**
Description	1. Turn on G05 high-speed high-precision mode in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Possible Cause	1. In the RTCP/STCP mode, turn on the G05 high-precision contour control mode with commands, such as G05 P10000. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		
Solution	1. Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-precision contour control mode. If (1), remove the command to turn on the G05 high-precision contour control mode in the RTCP/STCP mode. If (2), turn off the RTCP/STCP mode before turning on the G05 high-precision contour control mode. 2. When the high-precision contour control mode is enabled during processing, do not use the single block stop C40 at the same time. 3. When the STCP mode is enabled during processing, do not use the single block stop C40 at the same time.		

Alarm ID	COR-141 BGND-141	Alarm Title	G68.3 command format error
Description	[command format] G68.3 X_ Y_ Z_ R_; // The origin and z-axis rotation angle in the characteristic coordinate system. G68.3 P1 X_ Y_ Z_; // The origin of the characteristic coordinate system, and the coordinate system is determined with the tool rotation angle.		
Possible Cause	G68.3 command format, X, Y and Z are all exist or non-exist at the same time.		
Solution	Check if G68.3 command format is correct.		

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Description	K Argument error while using Spindle synchronization function.		
Possible Cause	The inputted K Argument is not within the valid range.		

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Solution	Input a K Argument within the valid range to enable the Spindle synchronization.		

Alarm ID	COR-143 BGND-143	Alarm Title	Programmable data input specified axis does not exist
Description	In G10 L1501 mode, the P argument is set incorrectly.		
Possible Cause	The axial direction substituted by the P Argument is inexistent.		
Solution	Check Pr21~ to ensure the axial direction is ON.		

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Description	When the machining multiple path program applies the G04.1 to wait for synchronization, the Q Argument assigns the paths to be waited for each other (decimally). The alarm is used to remind the users if the Q Argument type error or the assigned path is inexistent.		
Possible Cause	1. The Q Argument is not a positive integer (negative number, decimal point and zero are all illegal). 2. The path assigned by the Q Argument is inexistent. i.e. Q24 is used to assign the 2nd and 4th path to wait for each other. However, the Pr731=3 and there are only 3 CNC main system paths. 3. The Q Argument includes 0 when assigning the paths. i.e. Q103. 4. The paths assigned by the Q Argument do not include this in which the command is. ie. In the 2nd path, the G04.1 P1 Q13 assigns the 1st and 3rd paths to wait for each other, but it does not assign the path in which it is.		
Solution	Check the Path synchronization waiting (G04.1) in the program to ensure the type and the assigned path are correct for a given Q Argument.		

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Description	Spindle positioning failed.		

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Possible Cause	1. Incorrect Spindle ID is assigned. 2. The Spindle is operating in the tapping mode when the Spindle positioning function is ON. 3. After starting the positioning, C61 is OFF so that the positioning is broken.		
Solution	1. Check the Spindle parameter configuration. 2. Check the NC program to ensure the execution of Spindle positioning in non-tapping mode. 3. Check the PLC to ensure C61 is ON.		

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Description	Argument type error in a block.		
Possible Cause	The two following situations may trigger this alarm: 1. Input float value to an argument required integer value. 2. Input integer value to an argument required float value.		
Solution	Correct the argument type in the block.		

Alarm ID	COR-147 BGND-147	Alarm Title	Spindle is not enabled when path machining
Description	The machining Spindle assigned by the path is not enabled when executing the commands related to the Spindle.		
Possible Cause	When executing the G33/G34/G63/G74/G84, the Spindle assigned by the R791~R794 is not enabled.		
Solution	1. Check if the PLC switches R791~R794 values correctly in the block and whole NC program about the alarm. 2. Check if the Pr1621~Pr1628 configurations are correct (cooperate with R791~R794).		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Description	When using the tool retract function, a given improper argument results in the function cannot be executed.		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Possible Cause	1. In the Tilted working plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned axial position retraction. 2. In the non-Tilted working plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned retraction along the tool vector.		
Solution	1. In the slopping plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, apply G10.6 R_ to assign the tool retraction function. 2. In the non-slopping plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, apply G10.6 X_ to assign the tool retraction function.		

Alarm ID	COR-149 BGND-149	Alarm Title	Tilted working plane machining tool alignment P Argument over range
Description	P Argument is over range in Tilted working plane machining tool alignment mode.		
Possible Cause	P Argument is not within 0~2.		
Solution	Do not input the P Argument or input the correction configuration for the P Argument in Tilted working plane machining tool alignment mode.		

Alarm ID	COR-151 BGND-151	Alarm Title	1st rotation axis entering illegal range
Description	1st rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1st rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3009 and Pr3010 are configured correctly. The determination of such two configurations is related to Pr3007. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	**COR-152** **BGND-152**	**Alarm Title**	**2nd rotation axis entering illegal range**
Description	2nd rotation axis entering illegal range		
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3011 and Pr3012 are configured correctly. The determination of such two configurations is related to Pr3008. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	**COR-153** **BGND-153**	**Alarm Title**	**Tool direction unknown**
Description	Tool direction unknown.		
Possible Cause	5-axis configurations and machine mechanism is incompatible.		
Solution	The tool cannot reach the destination. It may be caused by the incompatible 5-axis configurations and machine mechanism. Please check all 5-axis configurations.		

Alarm ID	**COR-154** **BGND-154**	**Alarm Title**	**No 5-axis function**
Description	No 5-axis function.		
Possible Cause	Pr3001 is not configured when executing G53.1 tool alignment command.		
Solution	Check if Pr3001 is configured to 0. If yes, configure the other non-zero values based on the 5-axis mechanism type and reboot.		

Alarm ID	**COR-155** **BGND-155**	**Alarm Title**	**5-axis tool direction error**
Description	5-axis tool direction error.		

Alarm ID	COR-155 BGND-155	Alarm Title	5-axis tool direction error
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd rotation axis (Pr3005 and Pr3006) configuration error.		
Solution	Check if the Pr3002 is configured correctly, or if the Pr3005 or Pr3006 is configured correctly. The alarm will be triggered in case the 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

Alarm ID	COR-156 BGND-156	Alarm Title	5-axis axial direction error
Description	5-axis axial direction error.		
Possible Cause	Incorrect configurations are mapped to the axial direction parameters of 5 axis.		
Solution	Check if each axial direction is configured completely (Pr21~), if Pr3005, Pr3006, Pr3007 and Pr3008 is configured correctly, or if the axis name (Pr321~) is mapped to Pr3005 and Pr3006.		

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Description	Incompatible direction of 5-axis tool direction and this of rotation axis.		
Possible Cause	1. The 2nd axial direction and the tool direction are the same in the Spindle type. 2. The 1st axial direction and the tool direction are the same in the table type. 3. The 1st axial direction and the tool direction are the same in the hybrid type.		
Solution	Check if the tool direction and the rotation axial direction are the same depended on the used 5-axis mechanism type.		

P.S. Valid version of COR-157 : before 10.118.41M, 10.118.47 (included).

Alarm ID	COR-158 BGND-158	Alarm Title	Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode
Description	Since the G43.5 mode specifies the tool attitude based on the tool vector I, J and K, it shall not be executed for the 1st and 2nd rotation axis commands which can also specify the tool attitude.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the movement commands of the 1st and 2nd rotation axis are over the valid range in the G43.5 mode.		

Alarm ID	COR-159 BGND-159	Alarm Title	Illegal tool vector
Description	In NC program, a movement block assigns an incorrect tool vector.		
Possible Cause	Programming error. i.e. Execute G01 X_ Y_ Z_ I0 J0 K0 in the G43.5 mode, and the I0 J0 K0 refers to the 0 vector, 0 vector is illegal.		
Solution	Check the NC program to ensure the assigned tool vector is correct in the block indicated by the alarm line number		

Alarm ID	COR-160 BGND-160	Alarm Title	5-axis mechanism chain switched when the 5-axis function is ON
Description	In NC program, the command G10 L5000 [P_] is executed to switch the 5-axis mechanism when turning on the 5-axis function (RTCP or the tool vector alignment on the characteristic coordinate system is completed).		
Possible Cause	Programming error. i.e. Execute G10 L5000 [P_] in the G43.4 mode.		
Solution	Check the NC program to ensure the 5-axis function is not turned on in the block indicated by the alarm line number.		

Alarm ID	COR-161 BGND-161	Alarm Title	Selected 5-axis mechanism chain is not ON
Description	In NC program, the 5-axis mechanism chain parameters assigned by the command G10 L5000 [P_] is not ON.		
Possible Cause	Programming error. i.e. The assigned 5-axis mechanism chain is not configured correctly when executing a command G10 L5000 [P_].		
Solution	Check the 5-axis mechanism chain parameters to ensure the assigned 5-axis mechanism chain function is turned on correctly: 1. 1st set: Pr3001 2. 2nd set: Pr3101 3. 3rd set: Pr5501 4. 4th set: Pr5601		

Alarm ID	COR-162 BGND-162	Alarm Title	4-axis RTCP configuration error
Description	The controller will issue the alarm in case the 4-axis RTCP configuration is incorrect.		
Possible Cause	The 5-axis mechanism parameters, Pr3001, Pr3101, Pr5501 and Pr5601 are configured to 1~3 in case the specific 4-axis tool tip control function (option-29) is turned on but the tool tip control function (option-12) is not.		
Solution	1. Configure the 5-axis mechanism parameters to 4 or 5 correctly. 2. Turn on the tool tip control function (option-12).		

Alarm ID	COR-163 BGND-163	Alarm Title	Multi-kinematic chain command Q Argument setting error.
Description	Command G10 L5000P_ Q_, Q argument range error.		
Possible Cause	Command G10 L5000P_ Q_, Q argument range error.		
Solution	While using G10 L5000P_ Q_, check Q argument to be within 0~4, and is a integer.		

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Description	Command G10 L5000P_ Q_ specified the 5-Axis kinematic chain, and the 5-Axis mechanism parameter setting error.		
Possible Cause	While executing G10 L5000P_ Q_, Q argument is given, but the 5-Axis mechanism parameter of the designated 5-Axis kinematic-chain is not a spindle-type 5-Axis machine.		
Solution	Please check the designated 5-Axis kinematic-chain. The 5-Axis mechanism configuration parameter must be a spindle-type 5-Axis machine. 1. The first group : Pr3001 is 1. 2. The second group : Pr3101 is 1. 3. The third group : Pr5501 is 1. 4. The fourth group : Pr5601 is 1.		

Alarm ID	COR-165 BGND-165	Alarm Title	Multi-kinematic chain command not illegal.
Description	Command G10 L5000P_ Q_ is used for switching 5-Axis kinematic chain, and only provides partial 5-Axis mechanism function command.		
Possible Cause	G10 L5000 P_ Q_ command, the Q argument is set to 2~4 (not the first group of sub-kinematic chain), and only supports the following 5-Axis machine function command. 1. RTCP: G43.4. 2. RTCP: G43.5. 3. Tilted working plane : G68.2 + Tool alignment functions. 4. Tilted working plane : G68.3. Notice: Tool alignment functions include G53.1, G53.3, G53.6, ...		
Solution	When using the G10 L5000 P_ Q_ command to switch multi-kinematic chains, please use the supported 5-Axis machine function command.		

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Description	Option13 (Characteristic Coordinate System Option) was not active, therefore the CNC couldn't execute relevant commands.		

Alarm ID	COR-166 BGND-166	Alarm Title	**Characteristic Coordinate System Option not supported**
Possible Cause	One or more commands below were given while the Option13 (Characteristic Coordinate System Option) was inactive: 1. G68.2, G68.3 2. G53.1, G53.3, G53.6 3. Other commands that are relevant to Option13 (Characteristic Coordinate System Option)		
Solution	1. Activate Option13 (Characteristic Coordinate System Option) 2. Avoid using the commands listed above		

Alarm ID	COR-167 BGND-167	Alarm Title	Program File name conflicts in NcFiles
Description	Program File name conflicts in NcFiles folder.		
Possible Cause	When Pr3220 is set to 1, "non-main system", "PLC axis", "pre- and post-processing auxiliary programs" restrict calling files from the Macro folder, and prohibit the NcFiles folder from having the same file name.		
Solution	1. Set Pr3220 to 0. 2. Remove the conflicting files in the NcFiles folder or change the file name.		

Alarm ID	COR-168 BGND-168	Alarm Title	**Illegal tool compensation.**
Description	Illegal tool compensation		
Possible Cause	1. Trying to use non-zero tool length or tool radius compensation value on a machine type which doesn't support tool compensation		
Solution	1. Set tool compensation value as zero 2. Turn off tool compensation related functions		

Alarm ID	COR-169 BGND-169	Alarm Title	Overlap on the same axis
Description	Overlap on the same axis.		
Possible Cause	Using G1.10 with argument Q on the same axis with two consecutive blocks.		
Solution	Please check NC Program and avoid overlapping on the same axis with two consecutive blocks.		

Alarm ID	COR-170 BGND-170	Alarm 标题	【G43.4 L2 illegal argument value 】
说明	When using G43.4 L2. E_ R_, Illegal argument value is assigned		
可能原因	Illegal argument value 1. R value is negative 2. E value is not in the range of 0.001 to 179.999		
排除方法	Please check the E_ R_ argument values		

Alarm ID	COR-171 BGND-171	Alarm title	【Non-linear kinematic transform is not allowed in inclined axes control】
Description	When inclined axes control is enabled, it can't work with RTCP related function.		
Possible Cause	using following function, when inclined axes control is enabled. 1. G43.4 Rotate tool center point Type 1. 2. G43.5 Rotate tool center point Type 2. 3. G68.2,G68.3 Titled working plane machining with G43,G44 Tool length compensation. 4. G41.1, G42.1 tangential control.		
Solution	Don't use these function at program when inclined axes control is enabled.		

Alarm ID	COR-180	Alarm Title	【Invalid parameters setting for tangential control】

Description	Invalid command or parameters setting for tangential control.
Reason	1. Feed axes for tangential control do not exist. 2. Specified axis for tangential control does not exist. 3. Do not support specified axis type for tangential control. 4. Using spindle as rotary axis for tangential control but axis is not in C-axis mode. 5. Rotation direction of the tangential control axis is not yet set. 6. Location of the tangential control axis is not yet set. 7. G41.1, G42.1 command: a. Value of the Q argument is out of range. b. Value of the E argument is out of range. c. Value of the P argument is neither 0 or 1.
Solution	1. X or Y or Z axis do not exist. Please ensure axis exist(Pr21~) and axis name(Pr321~) correct. 2. Axis for tangential control(Pr5801) should be the existing axis(Pr21~). 3. Set axis type(Pr221~) of the tangential control axis to 2 or 5. 4. Switch spindle mode to C-axis mode(C63) before start tangential control. 5. Set rotation direction of the tangential control axis(Pr5802) to 1, 2, or 3. 6. Set location of the tangential control axis(Pr5803) to 1 or 2. 7. Please refer to the G41.1/G42.1 command description.

Alarm ID	**COR-181**	**Alarm Title**	**【Tangential control plane conflict】**
Description	Work plane is conflict with the tangential control plane.		
Reson	After tangential control function enabled, user has specified conflict work plane.		
Solution	Reassign tangential control axis and rotation direction of the tangential control axis or remove the conflict workplane specified.		

Alarm ID	**COR-182**	**Alarm Title**	**【Unsupport functions under tangential control】**
Description	When tangential control function is enabled, do not support some other functions.		
Reason	1. Put movement command to tangential control rotary axis. 2. Using tool compensation command such as G41, G42, G43.4, G43.5. 3. Using G02/G03 L_ command to do multiple turns of circle cutting. 4. Using G10L1150 to write parameters of the tangential control mechanism. 5. Put chopping command to axes under tangential control. 6. Turn on the G05 high-speed high-precision mode with command, such as G05 P10000. 7. Using G53 to do machine coordinate orientation. 8. Using G10L1500 to do rotary axis auxiliary brake.		

Solution	Don't use these functions until reset or tangential control function disabled(G40.1 or G49).

Alarm ID	**COR–201** **BGND-201**	**Alarm title**	**Program file not exist**
Description	The specified program does not exist.		
Possible Cause	The specified program does not exist.		
Solution	Ensure that program file exists.		

Alarm ID	**COR–202** **BGND-202**	**Alarm title**	**Communication link error**
Description	Communication link is dropped.		
Possible Cause	The transmission communication link is dropped.		
Solution	Reconnect a good transmission communication link.		

Alarm ID	**COR–203** **BGND-203**	**Alarm title**	**Illegal NC file format**
Description	NC program format not valid, as a result, the system cannot fully interpret the NC program.		
Possible Cause	1. The NC file is in macro format. 2. Use M98 to call a multi-path subprogram (including $1 and $2), and the size of the subprogram is greater than 60KB(60000bytes).		
Solution	1. Update the controller to 10.114.50I or the later version. 2. For M98 application, reduce the size of subprogram. Or, split it into individual single-path subprograms, then call each of them by coordinate respectively.		

Alarm ID	**COR–204** **BGND-204**	**Alarm title**	**File size too large**
Description	Program file is too large.		

Possible Cause	If the size of MACRO program is larger than 60KB(60000bytes) and use the syntax with scope statement. The syntax with scope statement contains: IF, CASE, REPEAT, FOR, WHILE		
Solution	1. Reduce the program size, or split program into two subprograms. 2. Remove all the syntax with scope statement.		
Alarm ID	**COR–205** **BGND-205**	**Alarm title**	**File content is empty**
Description	After controller loads the program, the file content found to be null.		
Possible Cause	Program loading error or CF card damaged.		
Solution	Reload program or replace CF card.		
Alarm ID	**COR–206** **BGND-206**	**Alarm title**	**Loading page lock failure**
Description	New NC program fails to required the system to distribute loading page.		
Possible Cause	Lack of memory when multi-system executes large-size program.		
Solution	Please upgrade to 9.240.27/ 9.242.5/10.104.36/10.111.41/10.113.5 or contact OEM Syntec.		
Alarm ID	**COR–207** **BGND-207**	**Alarm title**	**Specified sequence number not found**
Description	Specified sequence number is not found.		
Possible Cause	Programming error.		
Solution	Use a sequence number within the NC program range.		
Alarm ID	**COR–208** **BGND-208**	**Alarm title**	**Cannot use jump statement in sequential file**
Description	Using jump command to execute sequential file.		

Possible Cause	Use the jump command when executing the sequence file.
Solution	Do not use jump command to execute sequential file.

Alarm ID	**COR–209** **BGND-209**	**Alarm title**	**【File format error】**
Description	Invalid File Format.		
Possible Cause	1. It is not a MACRO format file. For example, using APP syntax to call MARCO G200, G200 APP "appname", but this G200 file is not declared as a MACRO format file. 2. It is not a ISO format file.		
Solution	Please check the NC program, which should be written according to the file opening requirements.		

Alarm ID	**COR-250** **BGND-250**	**Alarm title**	**Wrong mechanism axial setting**
Description	The axial setting of the mechanism is wrong.		
Possible Cause	The number of axis and the axis name required for the corresponding mechanism conversion are not set.		
Solution	Set sufficient number of axis (Pr21~), and required axis names (Pr321~).		

Alarm ID	**COR-251** **BGND-251**	**Alarm title**	**【The joint is not set to rotate in the right/left-hand direction】**
Description	The joint is not set to the direction of rotation as the right hand, left hand rule.		
Possible Cause	Pr4141~Pr4150 are not set to the right-hand rule or the left-hand rule.		
Solution	Set the rotation direction of each joint, 0 is not allowed.		

Alarm ID	COR-252 BGND-252	Alarm title	【The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position】
Description	The 2D mechanism is converted and the target position cannot be reached. Please enter a reasonable target position.		
Possible Cause	The program coordinate position exceeds the limit of the machine.		
Solution	The position is not reasonable. Please re-check the NC program.		

Alarm ID	COR -253 BGND-253	Alarm title	【Unworkable parallel mechanism posture】
Description	Parallel mechanism posture that is not feasible.		
Possible Cause	Currently at a parallel mechanism posture that is not feasible.		
Solution	After Reset, use the axial movement mode to leave this posture, or modify the target point.		

Alarm ID	COR-301 BGND-301	Alarm title	【OPEN command format error】
Description	OPEN command format error [command format] OPEN ("file name", "write file mode") (i) Designated as '**a**': Represent **retains** the original file content and **continuously** outputs the new data in the text file. (ii) Not specified or designated as '**w**': Represents the **emptying** of the original content and **re-outputs** the new material in the text file.		
Possible Cause	"Write mode" in the OPEN command format specifies an error. This alarm is issued if the string is specified as a string other than 'a' or 'w'.		
Solution	Set "FileWrite mode" to "a" or "w" according to the requirement of FileWrite.		

Alarm ID	COR-302 BGND-302	Alarm title	【G10 L1150 command format error】
Description	G10 L1150 command format error.		
Possible Cause	G10 L1150 parameter numbering (P argument) is a non-integer or the input is out of range or not entered. G10 L1150 parameter value (R argument) is out of range or not entered.		
Solution	Set the G10 L1150 parameter numbering and values (P and R arguments) correctly.		

Alarm ID	COR-303 BGND-303	Alarm title	【The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)】
Description	The basic spindle uses the spindle synchronization (G114.1, G51.2) commands in the position control mode.		
Possible Cause	When using this function, the basic spindle cannot be in position control mode.		
Solution	Write the NC program or PLC correctly so that the principal axis leaves the position control mode before the same period (C63).		

Alarm ID	COR-304 BGND-304	Alarm title	【Thread cutting geometrical axis name error】
Description	Thread cutting (G33, G34) command format is incorrect. Only the geometric axis names such as X, Y, Z, X1, Y1, Z1, etc. are supported.		
Possible Cause	When the thread cutting (G33, G34) does not have X, Y, Z, X1, Y1, Z1... and other geometric axis arguments, this alarm is issued.		
Solution	Adjust the axis names (Pr321~) of the geometrical axis to XYZ, X1/Y1/Z1...		

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Description	The mode in which the system is currently running cannot be used with the G91 incremental command.		

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Possible Cause	G43.5 cannot be used with the G91 incremental command function: G43.5 determines the tool attitude through the tool vectors I, J, and K. The tool attitude is expressed only in absolute quantities.		
Solution	Check the NC program to confirm that G91 was not executed in G43.5 mode and G43.5 was not executed in G91 mode		

Alarm ID	COR-306 BGND-306	Alarm title	【Advanced look-ahead function without the specified M code】
Description	The advanced look-ahead function is enabled, and the specified M code is not found after the complete NC program is interpreted.		
Possible Cause	1. Pr3599 is not set. 2. The NC programming error.		
Solution	1. Confirm whether the setting of Pr3599 is correct. 2. Check the NC program to confirm that an command has been given to speedily predecode M code (Pr3599).		

Alarm ID	COR-307 BGND-307	Alarm title	【Advanced look-ahead of M code, wrong designation of argument P】
Description	The advanced look-ahead M code (Pr3599) must specify the P argument.		
Possible Cause	When using the advanced look-ahead function, the P argument is not specified or the P argument is not an integer		
Solution	Check the NC program and confirm that there is a P argument and specify a 4-digit code subprogram.		

Alarm ID	COR-308 BGND-308	Alarm title	【Insufficient system memory, increase a single step displacement for advanced look-ahead function】
Description	The system look-ahead memory is insufficient, and the subsequent move command cannot be interpreted and is stuck.		

Alarm ID	COR-308 BGND-308	Alarm title	【Insufficient system memory, increase a single step displacement for advanced look-ahead function】
Possible Cause	There are too many very short blocks in the program, so the interpretation conditions cannot be met.		
Solution	Check the NC program and increase a single step displacement amount appropriately.		

Alarm ID	COR-309 BGND-309	Alarm title	【Advanced look-ahead function failed】
Description	The advanced look-ahead function cannot be executed correctly.		
Possible Cause	1. The interpolation mode is incorrect during advanced look-ahead. 2. The contents of the look-ahead NC program cannot be stored correctly.		
Solution	1. Check if the subprogram of the look-ahead NC programs contains commands other than G01, G02, and G03. 2. Check if disk space is adequate.		

Alarm ID	COR-310 BGND-310	Alarm title	【Look-ahead NC programs failed】
Description	Look-ahead NC program cannot be executed correctly.		
Possible Cause	1. The condition of the look-ahead NC program does not match: a. The version of the advanced look-ahead, the interpolation time, the number of axes, do not match. b. There is an abnormal interruption in the look-ahead, resulting in incomplete content of the look-ahead. c. To look-ahead the axis of movement in the subprogram, enter the advanced look-ahead M code in the main program. The coordinates before (i.e. M298) should be the same. (i.e. if the subprogram has X, Y, and Z three-axis movement commands, if the X, Y, and Z coordinates before look-ahead M code are modified after the look-ahead is completed, the look-ahead NC program will be executed. Issue this alarm.) 2. The contents of the look-ahead NC program cannot be read correctly.		
Solution	After advanced look-ahead under correct conditions, execute the look-ahead NC programs.		

Alarm ID	COR-311 BGND-311	Alarm title	【Hard disk storage space is insufficient to place look-ahead NC programs】
Description	The hard disk storage space is not enough to place the look-ahead NC program.		
Possible Cause	There is not enough storage space on the hard drive.		
Solution	Release hard disk storage space.		

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Description	Friction compensation adjustment setting is illegal.		
Possible Cause	1. Selected adjustment mode is not supported. 2. No axis is assigned for adjustment. 3. Radius or feedrate is not configured in Comp ON/Comp OFF mode . 4. No. of circular setting is not configured in Comp Learning mode.		
Solution	1. Select other compensation mode and try again. 2. Please set a set of available axes. 3. Please set correct radius or feedrate. 4. Please set correct No. of circular setting. If the alarm still exists, please contact Syntec OEM.		

Alarm ID	COR-321 BGND-321	Alarm title	【The arguments (P,Q,R) retrieved from the I point position must be integers】
Description	The I value position taken argument (G10 L1010 P_ Q_ R_) is not an integer.		
Possible Cause	NC programming error.		
Solution	Modify the NC program to specify correct P, Q, and R arguments.		

Alarm ID	COR-322 BGND-322	Alarm title	【I point position capture failed】
Description	I value position capture function (G10 L1010 P_ Q_ R_), startup failed.		

Alarm ID	**COR-322** **BGND-322**	**Alarm title**	**【I point position capture failed】**
Possible Cause	1. The specified P argument (axis number) does not exist. 2. The specified Q argument (I value) is not supported. 3. Exceeding the limit of the number of used groups, currently support to start three I value location captures at the same time.		
Solution	1. Modify the program to specify correct P, Q, and R arguments. 2. Turn off settings that are not in use.		

Alarm ID	**COR-323** **BGND-323**	**Alarm title**	**【The arguments (P,R,I,J,K) captured from the driver signal position must be integers】**
Description	The argument of the drive signal position (G10 L1011 P_ R_ I_ J_ K_) is not an integer.		
Possible Cause	The NC programming error.		
Solution	Modify the program to specify correct P, R, I, J, and K arguments.		

Alarm ID	**COR-324** **BGND-324**	**Alarm title**	**【Driver signal position captured failed to activate】**
Description	The drive signal position capture function (G10 L1011 P_ R_ I_ J_ K_) failed to activate.		
Possible Cause	1. The range of R values set by the specified P and R arguments is invalid (occupied by the system or out of range). 2. The specified I argument (axis number) does not exist. 3. The specified J argument (which set of latches to use) is out of range. 4. The specified K argument (select signal source) is out of range.		
Solution	Modify the NC program to specify correct arguments.		

Alarm ID	**COR-325** **BGND-325**	**Alarm title**	**【In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited】**
Description	In the polar coordinate interpolation (G12.1) mode, the diameter & radius axis programming switching command (G10.9) cannot be used.		

Alarm ID	COR-325 BGND-325	Alarm title	【In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited】
Possible Cause	In polar coordinate interpolation (G12.1) mode, if switch the diameter & radius axis program with the diameter & radius axis programming switching command (G10.9), this alarm is issued.		
Solution	In the polar coordinate interpolation (G12.1) mode, do not use the diameter & radius axis to program the switching command (G10.9). Please follow Pr281~Axis radius axis or diameter axis set value to move the command. [Remarks] For milling machine G12.1, the X axis is programmed in radius axis. For lathe machine G12.1, and the X axis programming mode can be set using Pr4020 (G12.1 X axis programming).		

Alarm ID	COR-326 BGND-326	Alarm title	【Diameter & radius axis programming command argument error】
Description	Diameter and Radius axis programming (G10.9) switching, command arguments are written incorrectly.		
Possible Cause	No axial programming is specified, or programming is specified as a value other than 0 and 1.		
Solution	1. Specify any axial direction after diameter & radius axis programming (G10.9). 2. Specify the argument value (programming mode) as 0 or 1.		

Alarm ID	COR-327 BGND-327	Alarm title	【Skip function argument input error】
Description	Skip function (G31) input argument is incorrect.		
Possible Cause	Skip function (G31) specifies both P and R arguments.		
Solution	Please modify the Skip function (G31), no specified both P and R.		

Alarm ID	COR-328 BGND-328	Alarm title	【G10 L1800 command argument number, out of specification limit】
Description	The command argument number is outside the specification limit. i.e. G10 L1800 I514 P10 R1, I argument is out of range.		

Alarm ID	COR-328 BGND-328	Alarm title	【G10 L1800 command argument number, out of specification limit】
Possible Cause	The NC programming error.		
Solution	Please confirm the argument value of G10 L1800.		

P.S. Valid version of COR-328 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

Alarm ID	COR-329 BGND-329	Alarm title	【G10 L1800 command number has exceeded the single block limit】
Description	The number of G10 L1800 commands with different arguments exceeds the upper limit allowed by a single block, up to 5, or too many G10 L1800 commands.		
Possible Cause	The NC programming error.		
Solution	Please reduce the number of G10 L1800 commands with different arguments to less than 5 in a block, or reduce the number of G10 L1800s as a whole.		

P.S. Valid version of COR-329 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

Alarm ID	COR-330 BGND-330	Alarm title	【Illegal interrupt signal format】
Description	The command argument number is outside the specification limit. i.e. M96 P5566 I4 Q100 R1 L1000; I argument specifies the error signal source.		
Possible Cause	The NC programming error..		
Solution	Please check the NC program to ensure that the command arguments are correct.		

Alarm ID	COR-331 BGND-331	Alarm title	【This product limits T Code function】
Description	This product only supports T0~T4.		
Possible Cause	This product only supports T0~T4.		
Solution	Use a products that support the full T-code function.		

Alarm ID	COR-332 BGND-332	Alarm title	【Interrupt type subprogram (M96, M97) execution failure】
Description	When the interrupt type subprogram function is used, the execution fails when the interrupt signal is triggered. Note: Due to the pre-solved relationship when issuing this alarm, there is no guarantee that it will stop at the problem line.		
Possible Cause	Only support to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). Triggering interrupt signals in other subprograms is not supported.		
Solution	1. Make sure to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). 2. Set #1510 to 4 (the second bit is set to on) and only display the line number of the main program. This can be regarded as triggering the interrupt in the main program, but this setting will re-execute the subprogram when it returns.		

Alarm ID	COR-333 BGND-333	Alarm title	【Single end point exceeds hardware stroke limit】
Description	The coordinate position in the program exceeds the hardware stroke limit set by the machine.		
Possible Cause	1. The NC program is wrong. 2. The hardware stroke limit signal is abnormal.		
Solution	1. Check the NC program and correct the coordinate position. 2. First confirm whether it is occasionally happen during machining, and MOT-25, 26 is no longer exists after the system enters the feedhold. If so, it may be that an axis hardware stroke limit signal is triggered, but then disappears; please check the limit switch wiring or the hardware itself for abnormality.		

Alarm ID	COR-334 BGND-334	Alarm title	【G10 L1810 illegal signal condition command format】
Description	G10 L1810 command format error.		
Possible Cause	1. One of the I, Q, and R arguments is not set. 2. The I, Q, R, and J arguments are out of range, set to a negative, or a decimal value.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-334 : before 10.118.34 (included).

Alarm ID	COR-335 BGND-335	Alarm title	【G10 L1810 signal waiting condition is too many or repeat the same signal】
Description	The G10 L1810 signal waiting condition exceeds the allowable number, or the G10 L1810 repeatedly sets the same signal.		
Possible Cause	1. G10 L1810 continuously sets more than 5 commands. 2. The same G10 L1810 signal command has been set.		
Solution	Please combine multiple signals into one signal to wait, or reduce the signal waiting condition.		

Alarm ID	COR-336 BGND-336	Alarm title	【G10 L1820 illegal waiting signal condition command format】
Description	G10 L1820 command format error.		
Possible Cause	P, K arguments are out of range, set negative or decimal.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-336 : before 10.118.34 (included).

Alarm ID	COR-337 BGND-337	Alarm title	【G10 L1820 without G10 L1810 command in the front】
Description	The G10 L1820 command needs to wait for the G10 L1810 signal wait condition. At least one G10 L1810 command must be programmed first.		
Possible Cause	The G10 L1820 command was not written before the G10 L1820 command.		
Solution	Modify the NC program and write the G10 L1810 command before the G10 L1820 command.		

Alarm ID	COR-338 BGND-338	Alarm title	【G10 L1820 signal condition waiting timeout】
Description	G10 L1820 waiting signal condition exceeds the waiting time.		
Possible Cause	The signal status condition was not reached within the waiting time.		

Alarm ID	COR-338 BGND-338	Alarm title	【G10 L1820 signal condition waiting timeout】
Solution	1. Check if the G10 L1810 setting signal condition is correct. 2. Check if the PLC wait signal is processed correctly. 3. Check the hardware device to confirm why the signal status is not reached.		

Alarm ID	COR-339 BGND-339	Alarm title	【Chopping axis prohibition movement command】
Description	The chopping axial direction does not accept any movement commands.		
Possible Cause	After using the chopping function (G81.1, G81.2, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.		
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.		

Alarm ID	COR-340 BGND-340	Alarm title	【Chopping axis prohibits changing coordinate system】
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.		
Possible Cause	1. After using the chopping function (G81.1, G81.2, C86), switch the coordinate system before closing and affect the chopping axis. 2. Simultaneously use of chopping function (G81.1, G81.2, C86) and tilted work plane machining function (G68.2, G68.3). 3. Simultaneously use the chopping function (G81.1, G81.2, C86) and the axis exchange function (C133~C136). Note: C86 support version: 10.118.19 and earlier.		

Alarm ID	COR-340 BGND-340	Alarm title	【Chopping axis prohibits changing coordinate system】
Solution	1. Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. 2. Check if the NC program uses the chopping function (G81.1, G81.2,C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). 3. Check if the NC program has the chopping function (G81.1, G81.2, C86) and the shaft exchange of the chopping shaft. (C133~C136). Note 1: C86 support version: 10.118.19 and earlier. Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658). Note 3: Coordinate system related operations: external coordinate offset, MPG offset.		

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Description	The specified axis cannot be switched to the chopping axis.		
Possible Cause	1. This axis has been designated as the PLC axis. 2. The axis has been designated as the spindle. 3. This axis has been designated as a chopping axis by other paths.		
Solution	1. Do not specify the PLC axis as the chopping axis. 2. Do not specify the spindle as the chopping axis. 3. Check if the multi-path repeats the chopping function for the same axis (G81.1, G81.2, C86). Note: C86 support version: 10.118.19 and earlier version.		

Alarm ID	COR-342 BGND-342	Alarm title	【Chopping axis prohibits non-linear kinematic transform】
Description	Non-linear kinematic transform is prohibited for the chopping axis.		
Possible Cause	1. Using non-linear kinematic transform when chopping function is enabled. 2. The machine type used is two-dimensional kinemetic (special model).		

Alarm ID	COR-342 BGND-342	Alarm title	【Chopping axis prohibits non-linear kinematic transform】
Solution	1. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the tangential control (G41.1, G42.1). 3. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 4. The machine configuration used is the two-dimensional kinemetic (special model), and the chopping function (G81.1, G81.2, C86) cannot be used. Note: C86 support version: 10.118.19 and earlier.		

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Description	M02/M30/M99 is not used at the end of the main program.		
Possible Cause	1. Programming error. 2. NC program is damaged. 3. Some machining information was lost due to file transmission errors.		
Solution	1. Check the NC program to reset the alarm and clear it. 2. Close the Pr3853-NC program error check function.		

Alarm ID	COR-344 BGND-344	Alarm title	【The NC program under invalid paragraph】
Description	1. There is continuous blank/annotation/garbled in the NC program, and the size is more than 8KB. 2. There are no valid commands in the NC program.		
Possible Cause	1. Programmning error. 2. NC program is damaged.		
Solution	1. Avoid writing lots of consecutive blanks or annotations. 2. Please check if the NC program is damaged. After the problem is solved, reset to remove the alarm. 3. Close the Pr3853 - NC program error check function.		

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Description	Programmable data input argument format error.		
Possible Cause	1. Wobble command a. One of the I, Q, R arguments is not set. b. I, Q, R arguments out of range. c. I argument not integer. 2. G10 L1803 command a. One of the I, Q, P, R arguments is not set. b. I, Q, P, R, J, E arguments out of range. c. I, Q, R, J, K, E arguments not integer. 3. G10 L1805 、G10 L1810 command a. One of the I, Q, R arguments is not set. b. I, Q, R, J arguments out of range. c. I, Q, R, J arguments not integer. 4. G10 L1820 command a. P, K arguments out of range. b. P, K arguments not positive integer. 5. G10 L4500 command a. One of the P, Q, R arguments is not set. b. P argument is not a string. c. Q, R arguments out of range. d. Q, R arguments not integer. 6. G10 L1030 command a. P、Q、R、X、Y、Z、I、J、K、U、V arguments out of range. b. Q、X、Y、Z、I、J、K、U、V arguments not integer. 7. G10 L1031 command a. R argument is not set. b. R, V arguments out of range. c. R, V arguments not integer. 8. G10 L1500 command a. One of the P、Q arguments is not set. b. One of the P、Q arguments is out of range. c. One of the P、Q arguments is not an integer. 9. G10 L1350 command a. P, E argument is not set. b. P, E argument is out of range. c. P argument not integer.		
Solution	Please refer to command description to troubleshoot.		

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Description	Programmable data input conflicts with specifications.		

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Possible Cause	1. Between wobble command, only one block is allowed to wobble. 2. G10 L1803 commands with different arguments exceeds the upper limit allowed by a single block, up to 12. 3. Too many G10 L1803 and G10 L1805 commands. 4. The feature as L-argument specified is unsupported for current version (from 10.118.41P, 10.118.49). 5. Any axis belongs to multiple coordinates.		
Solution	1. Remove the exceeding number of wobble moving blocks. 2. Reduce the number of G10 L1803 commands in a single movement block to less than 12. 3. Reduce G10 L1803 and G10 L1805 commands. 4. Update to the version which supports the feature as L-argument specified. 5. Please set Pr701~ of the axis to only belong to a single axis group.		

Alarm ID	COR-347 BGND-347	Alarm title	【This function only supports Syntec axis】
Description	Only Syntec axis can use this function.		
Possible Cause	The weave function only supports Syntec drives.		
Solution	Please use Syntec driver.		

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error 】
Description	`There is an error in the APP command.`		
Possible Cause	1. APP name is not a string. 2. The length of APP name is over 20 characters. 3. No APP name is specified.		
Solution	Please make sure that APP commands are used correctly in the NC program.		

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Description	Limited to use with Syntec magnetic gear or ring gear encoders.		

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Possible Cause	When complex threading cycle(G78 or G78.2) enables chip removal function, the active spindle must use Syntec magnetic gear or ring gear encoders. In dual-feedback system, it is allowed to use chip removal function when either of the encoders is Syntec magnetic gear or ring gear encoder.		
Solution	1. Set the argument of D to 0. 2. Please use Syntec magnetic gear or ring gear encoders.		

Alarm ID	COR-350 BGND-350	Alarm title	【Invalid EnIP Macro】
Description	EnIP macro syntax error.		
Possible Cause	The format or range of the input macro arguments is incorrect.		
Solution	Refer to the EnIP macro syntax description.		

Alarm ID	COR-351 BGND-351	Alarm title	【Communication command abnormal】
Description	1. The execution of the macro communication command failed. 2. The macro communication to driver failed.		
Possible Cause	1. The controller does not support EnIP communication. 2. The EnIP function input request data value range is incorrect. 3. An error occurred while sending or receiving a communication command.		
Solution	1. Please contact the Syntec OEM. 2. Each request data of EnIP represents a 1 value ranging from 0 to 255. 3. This alarm is accompanied by a more detailed error alarm, refer to the alarm description for troubleshooting.		

Alarm ID	COR-352 BGND-352	Alarm title	【Communication command response data is too long】
Description	The device return data length is too long.		
Possible Cause	The storage space provided by the user for returning data is insufficient.		

Alarm ID	**COR-352** **BGND-352**	**Alarm title**	**【Communication command response data is too long】**
Solution	Make sure that the storage space for the returned data is sufficient.		

Alarm ID	**COR–353** **BGND-353**	**Alarm title**	**【Invalid argument of CHKINF】**
Description	The type of argument of CHKINF() is incorrect, or the category number is out of range.		
Reason	Programming error.		
Solution	1. Please check the NC program. 2. Check the type of first argument is integer. 3. Check the type of second argument is string. 4. Ensure the category number is in the range, please refer to OpenCNC_Macro Development Manual.		

Alarm ID	**COR-354** **BGND-354**	**Alarm title**	**【With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.】**
Description	With Rotate Tool Center Point function(G43.4, G43.5) enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.		
Reason	The NC file was written incorrectly.		
Solution	Please modify the NC file, cancel Rotate Tool Center Point function(G49), and then use G10 L1150 to set the Five-Axis mechanism parameter.		

Alarm ID	**COR–355** **BGND-355**	**Alarm title**	**Need geometric axes positioning after tool length compensation changed**
Description	Positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) is required after executing tool length compensation commands, if tool radius compensation commands needed.		
Reason	No positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) between executing tool length compensation commands and tool radius compensation commands. Tool length compensation commands: H, T(Lathe), G43, G44, G43.4, G43.5, G49, G10 L1050, G10 L1051 Tool radius compensation commands: G41, G42		

Solution	Please modify the NC file. Add positioning command of the axes on work plane(G17: XY, G18: ZX, G19: YZ) after tool length compensation commands and before tool radius compensation commands. Positioning commands: G90 G00, G90 G01, G92

Alarm ID	**COR–356** **BGND-356**	**Alarm title**	**【Invalid APP command】**
Description	There is a syntax error on the Modbus command(G10 L1900/1901/1910/1911).		
Possible Cause	The format or range of the input argument is incorrect.		
Solution	Please refer to the Industry Machine Application Manual.		

Alarm ID	**COR–357** **BGND-357**	**Alarm title**	**【Illegal Modbus packet contents】**
Description	The contents of the Modbus command packet is illegal.		
Possible Cause	The contents of the customized package is incorrect.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

Alarm ID	**COR–358** **BGND-358**	**Alarm title**	**【Modbus communication timeout】**
Description	The Modbus communication experienced a timeout.		
Possible Cause	Abnormal communication or unestablished communication causes the connection timeout.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

Alarm ID	**COR–359** **BGND-359**	**Alarm title**	**【Modbus communication failure】**
Description	The Modbus communication failed.		
Possible Cause	1. There is a communication error or device setting error.		

Alarm ID	**COR–359** **BGND-359**	**Alarm title**	**【Modbus communication failure】**
Solution	1. Please refer to R5039, as this register provides the error code for troubleshooting		

Alarm ID	**COR-361** **BGND-361**	**Alarm title**	**【The number of non-moving blocks exceeds the permissible value under tool radius compensation】**
Description	The machining program has programmed too many non-moving blocks when tool radius compensation is enable.		
Possible Cause	Programming error.		
Solution	1. Modify the machining program and issue the non-moving blocks when tool radius compensation is disable. 2. Modify the machining program and reduce the number of non-moving blocks when tool radius compensation is enable.		

Alarm ID	**COR-362** **BGND-362**	**Alarm 标题**	**【Multi-axis multi-signal skip function (G31.10, G31.11) command error】**
说明	Multi-axis multi-signal skip function (G31.10, G31.11) command error		
可能原因	1. Setting command (G31.10) or execution command (G31.11) cannot be issued alone, and other commands cannot be issued between them. Always set first then execute. 2. A set of multi-axis multi-signal skip function is composed of one G31.10 command at least and one G31.11 command. Set first then execute. Both commands should be issued in a set of multi-axis multi-signal skip function, and it is not allowed to insert other commands in between. 3. Supports up to six G31.10 commands in a set of multi-axis multi-signal skip function. 4. Same axis is used in different G31.10 commands in a set of multi-axis multi-signal skip function. E.g., a. Same axis name is used in different G31.10 commands. b. Assign same axis by using virtual axis function in different G31.10 commands. (E.g., Virtual axis Z is corresponding to axis Z1 and Z2. Using axis Z in one G31.10 command, and Z1 in another G31.10 command.) c. Assign same axis by misusing alias of axis. (E.g., There are axis X, Y, Z1, Z2, Z3, Z4, the alias of Z1 is Z.)		

排除方法	1. Make sure there is always an execution command (G31.11) behind setting command (G31.10), and there is no execution command (G31.11) left alone. 2. Make sure there are six G31.10 commands at most in each set of multi-axis multi-signal skip function. 3. Make sure there is no axis assigned repeatedly in each set of multi-axis multi-signal skip function.		
Alarm ID	**COR-363**	**Alarm title**	**【Invalid axis removal/axis borrowing function】**
Description	Axis removal (G52.1) or axis borrowing (G52.2) function is used incorrectly.		
Reason	1. Without any argument of P, Q, R in the G52.1 and G52.2 commands. 2. The axes corresponding to the P, Q, R arguments does not exist in the system. 3. I, J, K arguments are integers but out of the settable range. 4. Issue the G52.1 or G52.2 command for non-roaming axis. 5. When using the axis removal command, the axis to be removed hasn't been borrowed by the axis group. 6. When using the axis borrowing command, the axis to be borrowed has been borrowed by the axis group. 7. It is not allowed to set the related axes of nonlinear mechanism transform as roaming axes, including: • RTCP (G43.4, G43.5); • Tilted plane function (G68.2, G68.3 and G53.1, G53.3, G53.6); • Tangential control function (G41.1, G42.1); • Polar coordinate conversion (G12.1); • Two-Dimensional Kinemetic (special model). 8. Issue the G52.1 or G52.2 command for Indexing axis.		
Solution	1. In the G52.1 or G52.2 command, at least one argument among P, Q, R is required. 2. Please refer to Pr321~ setting value, modify the NC program, and specify the correct axes. 3. Please refer to the command format, modify the NC program, and set the parameters within the settable range. 4. Please set Pr701~ to belong to multiple axis groups, and set Pr742 to 1(i.e., set the axis to be borrowed as the roaming axis.); Or please modify the processing program and delete the G52.1 or G52.2 command. 5. Check the NC program, fix the P, Q, R argument assignment error. 6. Check the NC program, fix the P, Q, R argument assignment error. 7. Please set Pr742 to 0 to avoid the disallowed usages in No.7 of the possible reason. 8. Check and modify the NC program, don't remove or borrow indexing axis.		
Alarm ID	**COR-364**	**Alarm title**	**【Axis borrowing function failed to borrow】**
Description	Use axis borrowing (G52.2) function, unable to borrow all specified axes successfully.		

Reason	Because the axis to be borrowed is being borrowed by other axis groups, it cannot be borrowed successfully.
Solution	1. Check the NC program and confirm that the axis to be borrowed has been removed by other axis groups before issuing the borrow instruction. 2. Enlarge the K argument setting value, so that extends the waiting time. 3. Set the I parameter to 0, and continue to wait for other axis groups to remove the axis.

Alarm ID	**COR-365**	**Alarm title**	**【Issue a movement command to the unborrowed roaming axis】**
Description	Issue a movement command to the unborrowed roaming axis.		
Reason	1. When Pr742 is set to 1, the axis group or PLC Rn sub-program component issue a movement command to the unborrowed roaming axis. 2. The tool break retract is performed on the path with the roaming axis command, and the roaming axis that has not been borrowed successfully, so an alarm is issued.		
Solution	1. For the axis group or PLC Rn sub-program component, before issuing a movement command to the roaming axis, use the G52.2 command to successfully borrow the axis. 2. Modify the tool break retract setting line number or serial number before G52.2 command.		

Alarm ID	**COR-366** **BGND-366**	**Alarm Title**	**【The parameters of rotary axis auxiliary brake are set incorrectly】**
Description	Using the rotary axis auxiliary brake, the parameters mapped to the enable group are set incorrectly.		
Possible Cause	Parameters are incorrect. Enable G10 L1500 rotary axis auxiliary brake: 1. The axis ID of the rotary axis auxiliary brake is not specified. 2. The axis type of the specified axis is linear. 3. The M code of the rotary axis auxiliary brake is not specified.		
Solution	Modify parameters Pr3741~Pr3744.		

Alarm ID	**COR-367** **BGND-367**	**Alarm Title**	**【Prohibited commands in rotary axis auxiliary brake mode】**

Description	When the rotary axis auxiliary brake is enabled, prohibited commands are used in the nc program.
Possible Cause	1. The axis, which is in rotary axis auxiliary brake mode, is specified by skip function(G28.1, G31, G31.11). 2. The axis, which is in rotary axis auxiliary brake mode, is specified by chopping function(G81.1, G81.2). 3. Enable the rotary axis auxiliary brake in RTCP mode(G43.4, G43.5). 4. Enable both the rotary axis auxiliary brake mode and the polar coordinate mode(G12.1).
Solution	Check the NC program.

Alarm ID	**COR-368** **BGND-368**	**Alarm title**	**【The position command of the indexing axis is inaccurate】**
Description	The index position command isn't an integral multiple of the minimum precision angle.		
Possible Cause	The index position or movement command at current program isn't an integral multiple of the minimum precision angle.		
Solution	Please check the program, correct the position or movement command of the index table to an integer multiple of the minimum precision angle.		

Alarm ID	**COR-369** **BGND-369**	**Alarm title**	**【Illegal indexing axis command 】**
Description	The command of the indexing axis in the program is illegal.		
Possible Cause	1. There are both indexing axis and normal axes in the same G01、G02、G03、G02.4、G03.4、G33、G34 command. 2. G12.1 polar coordinate interpolation specifies the indexing axis as the rotation axis. 3. G28.1, G31, G31.10 skip function specifies the indexing axis. 4. G80.1, G80.2, G81.1, G81.2 command the indexing axis to chopping.		
Solution	1. Please check and modify the program, and split the indexing axis and normal axis into different G01、G02、G03、G02.4、G03.4、G33、G34 command. 2. Don't specify an indexing axis as a rotation axis for polar coordinate interpolation. 3. Don't use the indexing axis for the skip function. 4. Don't use indexing axis for chopping.		

Alarm ID	COR-370 BGND-370	Alarm title	【Failed to enable chopping function】
Description	Failed to enable the chopping function .		
Possible Cause	1. G81.2 or G81.1 command repeatedly a. G81.1 command repeatedly. b. G81.2 and G81.1 command simultaneously. 2. Using G81.2 to make 2 axis in the same axis group do advanced chopping simultaneously.		
Solution	Please activate or deactivate the chopping function correctly.		

Alarm ID	COR–401 BGND-401	Alarm title	Path planning plug-in error
Description	Path planning plug-in error.		
Reason	Path planning plug-in error leads to system abnormality.		
Solution	Please contact OEM Syntec.		

7.1 COR-001 Array Variable is empty

Alarm ID	COR–001 BGND-001	Alarm title	Array Variable is empty
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

7.2 COR-002 File not exist

Alarm ID	COR–002 BGND-002	Alarm title	File not exist

Description	If the file that the system wants to read does not exist i.e. use M98 (or G65, G66, and etc.) to call a non existing file.
Reason	Programming error.
Solution	Check the NC program to make sure the existence of the file.

7.3 COR-003 Devide by zero error

Alarm ID	**COR–003** **BGND-003**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of MACRO is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		
Reason	Programming error		
Solution	Check the NC program to ensure that the denominator is not equal to 0.		

7.4 COR-004 Operation domain error

Alarm ID	**COR–004** **BGND-004**	**Alarm title**	**Operation domain error**
Description	Operation domain error.		
Reason	Programming error		
Solution	Please check the NC program.		

7.5 COR-005 Program loading failure

Alarm ID	**COR–005** **BGND-005**	**Alarm title**	**Program loading failure**
Description	MACRO syntax error.		

Reason	Programming error.
Solution	Please check the NC program.

7.6 COR-006 Arc not on work plane

Alarm ID	**COR–006** **BGND-006**	**Alarm title**	**Arc not on work plane**
Description	1. In clockwise, counter-clockwise arc cutting (G02, G03) syntax, if the vector from center of circle to starting point doesn't exist on the arc working surface. i.e. execute G17 G02 I50. K10. , and system will issue this alarm. 2. In spiral interpolation (G02, G03) syntax, when corresponding to G17, G18, and G19 three surface conditions, if K, J , and I arguments are not zero, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that G02 and G03 are used correctly.		

7.7 COR-007 Arc radius too short

Alarm ID	**COR–007** **BGND-007**	**Alarm title**	**Arc radius too short**
Description	In G02 and G03 syntax, if Arc radius is smaller than 10 to the power of minus 10 BLU (10^-10), system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.		

7.8 COR-008 Arc destination not on arc

Alarm ID	**COR–008** **BGND-008**	**Alarm title**	**Arc destination not on arc**

Description	In G02 and G03 syntax, if the Arc end point coordinate is not on the circle, system will issue this alarm. From 8.31 version, adding Pr3807 setting arc final point is not on the check window, allowing to set window range's error in Pr3807. When arc end point position's error is smaller than Pr3807 setting window's range, system will auto correct the circle's center position, and make the end point position to locate on the circle correctly. When arc end point position error is bigger than Pr3807 window range setting, system will issue this alarm.
Reason	Programming error.
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.

7.9 COR-009 G65 layers called by Macro too deep

Alarm ID	**COR–009** **BGND-009**	**Alarm title**	**G65 layers called by Macro too deep**
Description	Use G65 to call MACRO program layer more than 12 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 12 layers.		

7.10 COR-010 G66 layers called by Macro too deep

Alarm ID	**COR–010** **BGND-010**	**Alarm title**	**G66 layers called by Macro too deep**
Description	Use G66 to call MACRO program layer more than 4 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 4 layers.		

7.11 COR-011 Subprogram layer call too deep

Alarm ID	COR–011 BGND-011	Alarm title	Subprogram call too deep
Description	Use M98 to call subprogram that has more than 16 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that M98 calls subprogram that has less than 16 layers.		

7.12 COR-012 G66 mode not cancel by G67

Alarm ID	COR–012 BGND-012	Alarm title	G66 mode not cancel by G67
Description	G66 and G67 need to be used in pairs, when numbers of G67 is more than G66 in one NC program, this alarm will be issued.		
Reason	Programming error.		
Solution	Check NC program to ensure that G66 and G67 are used in pairs		

7.13 COR-013 G65, G66 must be at the end of the block

Alarm ID	COR–013 BGND-013	Alarm title	G65, G66 must be at the end of the block
Description	G65 and G66 are MACRO, so in one same block the program on the right hand side of G65 and G66 will be processed as G65 and G66's arguments. So in one same block, if there is other G code command please write them in the left hand side of G65 and G66. In one same block, if the right hand side of G65 and G66 has G code or M code, system will issue this alarm.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.14 COR-014 Absent program number

Alarm ID	COR–014 BGND-014	Alarm title	Absent program number
Description	The right hand side of G65 and G66 doesn't have P argument to specify program number.		
Reason	Programming error.		
Solution	Please check the NC program to ensure G65 and G66 use P argument to specify program number.		

7.15 COR-015 Too many M code

Alarm ID	COR–015 BGND-015	Alarm title	Too many M code
Description	There are more than 3 M codes in a single block.		
Reason	Programming error.		
Solution	Please check the NC program to ensure that there are equal or less than 3 M codes in a single block		

7.16 COR-016 Illegal variable access

Alarm ID	COR–016 BGND-016	Alarm title	Illegal variable access
Description	Attempted to access a variable that does not exist.		
Reason	1. Programming error. 2. Illegal access to # or @ variable. 3. The parameter of SYSDATA function is out of system data range. 4. The controller does not support the status variable which DRVDATA is to read. 5. The program that attempts to access AR or MAR variables does NOT lie within the scope of App Macros. 6. The AR or MAR variables to access do NOT lie within the range defined by the App. 7. Non-integer numbers are used to specify the addresses of AR or MAR variables. ONLY integers are acceptable.		

Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable. 4. Make sure the variable which DRVDATA is to read can be found on the "Controller Axis Info." page. 5. Make sure the AR and the MAR variables are accessed ONLY in App Macros. 6. Make sure the AR and the MAR variables to access lie within the range defined by the App. 7. Make sure only integers are used to specify the addresses of AR or MAR variables.

7.17 COR-017 Label not found

Alarm ID	**COR–017** **BGND-017**	**Alarm title**	**Label not found**
Description	Cannot find out corresponding serial number N in GOTO command.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.18 COR-018 Line number not found

Alarm ID	**COR–018** **BGND-018**	**Alarm title**	**Line number not found**
Description	Input line number is incorrect.		
Reason	1. When Pr3851 sets as 888800, input breakpoint’s line number exceeds NC program’s maximum line number. 2. When Pr3851 sets as 999900, input breakpoint’s line number exceeds NC program’s maximum line number. 3. When Pr3851 sets as 999901, input breakpoint’s line number exceeds NC program’s maximum line number or positioning command line number cannot be found after scanning through entire NC program. 4. When M99 Q_ sub-program returns to the line number which is assigned by main program, the line number Q_ assigned is over main program’s maximum line number.		
Solution	1. Input the correct program line number. 2. When Pr3851 sets as 999901, modify NC program by inserting positioning command after initial assigned breakpoint.		

7.19 COR-019 sub-program has no M99

Alarm ID	**COR–019** **BGND-019**	**Alarm title**	**sub-program has no M99**
Description	When main program calls sub-procedure, if sub-program finish executing and need to return the main program, there is no M99.		
Reason	Programming error.		
Solution	Write in M99 when sub-program finish executing and return the main program.		

7.20 COR-020 Too many G code

Alarm ID	**COR–020** **BGND-020**	**Alarm title**	**Too many G code**
Description	There are more than 10 G codes in a single block.		
Reason	Programming error.		
Solution	Breakdown that single block which has over 10 G codes into single blocks that has less than 10 G codes.		

7.21 COR-021 Too many (I,J,K) parameters

Alarm ID	**COR–021** **BGND-021**	**Alarm title**	**Too many (I,J,K) triples**
Description	Repeat too much I, J, and K command in the same block.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.22 COR-022 Use undefined G54 workpiece coordinate

Alarm ID	**COR–022** **BGND-022**	**Alarm title**	**Use undefined workpiece coordinate**
Description	Use an undefined G54 coordinate system.		
Reason	Programming error.		
Solution	Use the correct G54 coordinate system.		

7.23 COR-023 Semantic error

Alarm ID	**COR-023** **BGND-023**	**Alarm Title**	**Semantic error**
Description	1. When using G code, different G code can input different argument (P_, L_, R_, and etc.), if argument setting is wrong, and system will issue this alarm. 2. The argument of macro function is wrong.		
Reason	Programming error.		
Solution	Check the NC program to ensure the argument is used correctly by referring the manual.		

7.24 COR-024 Invalid arc radius value

Alarm ID	**COR–024** **BGND-024**	**Alarm title**	**Invalid arc radius value**
Description	When executing G02, G03, appointed Arc end point and given radius is contradicted, which the given radius cannot meet appointing Arc end point. i.e. G91 G03 X1500 Y4000 R2000.		
Reason	Programming error.		
Solution	Check the program and recalculate.		

7.25 COR-025 Macro stack is overflow or STKTOP[ ] argument error

Alarm ID	COR–025 BGND-025	Alarm title	Macro stack is overflow or STKTOP[ ] argument error
Description	1. STACK can store maximum 4095 values. If stored value exceeding the maximum number, controller issue this alarm. 2. In STKTOP[n], n is started from 0, if the value of n is bigger than the value storing in stack-1, controller issue this alarm		
Reason	1. Too much value store in STACK. 2. STKTOP[] arguments exceeds the value storing in STACK.		
Solution	1.Stack is full, do not use store command anymore. 2. Input a reasonable argument in STKTOP[].		

7.26 COR-026 Macro stack is empty

Alarm ID	COR–026 BGND-026	Alarm title	macro stack is empty
Description	Empty stack still want to pop value, system issue this alarm.		
Reason	The numbers of Push commands and Pop commands are not the same.		
Solution	Check the program to ensure that the number of Push commands is the same with that of Pop commands.		

7.27 COR-027 Invalid macro arguments

Alarm ID	COR–027 BGND-027	Alarm title	Invalid macro arguments
Description	Macro issue self-defined alarm.		
Possible Cause	Once Macro determines the self-defined alarm error condition is meet, NC program will be stopped and alarm will appear.		
Solution	According to the display content of alarm to find out the error.		

7.28 COR-028 System program error, can’t normally machining.

Alarm ID	COR–028 BGND-028	Alarm title	System program error, can’t normally machining.
Description	Use Quiet Mode in MACRO, but can't leave Quiet Mode when program finished.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.29 COR-029 Tool length offset change at arc

Alarm ID	COR–029 BGND-029	Alarm title	Tool length offset change at arc
Description	G43, G44, G49 only receive linear interpolation command in the next block.		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

7.30 COR-030 Cutting speed command is 0

Alarm ID	COR-030 BGND-030	Alarm Title	Cutting speed command is 0
Description	When execute cutting command, given F code argument is zero.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the argument of F code shall not be 0.		

7.31 COR-031 Radius compensation cancel at arc

Alarm ID	COR–031 BGND-031	Alarm title	Radius compensation cancel at arc

Description	G40 only receives linear interpolation command in the next block
Possible Cause	Programming error.
Solution	Please check the NC program.

7.32 COR-032 Radius compensation activate at arc

Alarm ID	**COR–032** **BGND-032**	**Alarm title**	**Radius compensation activate at arc**
Description	G41,G42 only receive linear interpolation command in the next block.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.33 COR-033 Improper use of A, R, or C command

Alarm ID	**COR–033** **BGND-033**	**Alarm title**	**Improper use of A, R, or C command**
Description	Timing of using A, R, or C command is incompatible with the specification.		
Reason	Programming error.		
Solution	Check NC program to confirm that if the blocks are compatible with specifications		

7.34 COR-034 Command syntax has conflict

Alarm ID	**COR–034** **BGND-034**	**Alarm title**	**Command syntax has conflict**
Description	No application currently.		
Reason			

Alarm ID	COR–034 BGND-034	Alarm title	Command syntax has conflict
Solution			

7.35 COR-035 Corner is too small can’t insert round corner chamfer

Alarm ID	COR–035 BGND-035	Alarm title	Corner is too small can’t insert round corner chamfer
Description	Angel between blocks, which to be inserted round corner or chamfer is too small so system can’t calculate it.		
Reason	Programming error.		
Solution	1. Check NC program to confirm that whether the block is compatible with specifications. 2. Check and confirm the working plane and round corner plane are the same.		

7.36 COR-036 Inappropriate A angle command

Alarm ID	COR–036 BGND-036	Alarm title	Inappropriate A angle command
Description	A is valid only in linear interpolation of single block.		
Reason	Programming error.		
Solution	Check NC program to confirm that whether A angle command is compatible with specifications		

7.37 COR-037 Chamfer value bigger than displacement

Alarm ID	COR–037 BGND-037	Alarm title	Chamfer value bigger than displacement
Description	Chamfer value is bigger than pre-block and post-block length, so system can’t calculate it.		

Reason	Programming errors.
Solution	1. Check NC program to confirm that whether inserted value of chamfer is compatible with specifications. 2. Check to confirm whether C value is compatible with specification.

7.38 COR-038 Incorrect block jump's switch number.

Alarm ID	**COR–038** **BGND-038**	**Alarm title**	**Incorrect block jump's switch number.**
Description	Incorrect block jump switch number. i.e. / 2 G00 X100. ; If assigned number is bigger than 9 or equals to 0, system will issue this alarm.		
Reason	Block jump switch number is 0 or bigger than 9.		
Solution	Check the NC program, and confirm block jump switch number is between 1 and 9.		

7.39 COR-039 Measure function can't start tool radius compensation

Alarm ID	**COR–039** **BGND-039**	**Alarm title**	**Measure function can't start tool radius compensation**
Description	Measure function can't start tool radius compensation.		
Reason	Measure function related instruction (i.e., G31, G31.10, G31.11) is executed after tool compensation command.		
Solution	Confirm there is no measure function related command is executed after tool compensation command.		

7.40 COR-040 Block end point exceed software stroke limit

Alarm ID	**COR–040** **BGND-040**	**Alarm title**	**Block end point exceed software stroke limit**
Description	The machine coordinate in the program exceeds the software stroke limit.		

Reason	Programing error.
Solution	Check the NC program, and correct coordinate position.

7.41 COR-041 GOTO label must be integer

Alarm ID	**COR–041** **BGND-041**	**Alarm title**	**GOTO label must be integer**
Description	The input GOTO label is not an integer. i.e. GOTO 1 Correct GOTO 1. Wrong N1; Correct N1.; Wrong		
Reason	Programming error.		
Solution	Check the NC program, and input integer in GOTO argument.		

7.42 COR-042 Logic operand is non-integer or empty

Alarm ID	**COR–042** **BGND-042**	**Alarm title**	**Logic operand is non-integer or empty**
Description	Logic operand must be non-integer or empty.		
Reason	Logic operand has floating point. i.e. #1=1.5And3. System will issue this alarm.		
Solution	Please check NC program. Confirm logic operand is integer or empty.		

7.43 COR-043 ASIN, ACOS operand must between ±1.0

Alarm ID	**COR–043** **BGND-043**	**Alarm title**	**ASIN, ACOS operand must between ±1.0**

Description	ASIN() and ACOS() operand is not between -1.0 and 1.0.
Reason	Programming error.
Solution	Check the NC program.

7.44 COR-044 SQRT operand should not be negative

Alarm ID	**COR–044** **BGND-044**	**Alarm title**	**SQRT operand should not be negative**
Description	The square root of a negative value will be imaginary number, but the controller does not provide imaginary number function.		
Reason	Programming error.		
Solution	Check the NC program; enter a positive value in SQRT operand.		

7.45 COR-045 L address should be integer

Alarm ID	**COR–045** **BGND-045**	**Alarm title**	**L address should be integer**
Description	The L address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in L address.		

7.46 COR-046 O address should be integer

Alarm ID	**COR–046** **BGND-046**	**Alarm title**	**O address should be integer**
Description	The O address is not an integer.		
Reason	Programming error.		

Solution	Check the NC program, and use integer in O address.

7.47 COR-047 M address should be integer

Alarm ID	**COR–047** **BGND-047**	**Alarm title**	**M address should be integer**
Description	The M address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in M address.		

7.48 COR-048 Spindle speed S should be integer

Alarm ID	**COR–048** **BGND-048**	**Alarm title**	**Spindle speed S should be integer**
Description	The spindle speed S is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in the spindle speed S.		

7.49 COR-049 Tool length compensation H should be integer

Alarm ID	**COR–049** **BGND-049**	**Alarm title**	**Tool length compensation H should be integer**
Description	Tool length compensation H is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool length compensation H.		

7.50 COR-050 Tool radius compensation D should be integer

Alarm ID	COR–050 BGND-050	Alarm title	Tool radius compensation D should be integer
Description	Tool radius compensation D is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool radius compensation.		

7.51 COR-051 Tool number T should be integer

Alarm ID	COR–051 BGND-051	Alarm title	Tool number T should be integer
Description	Tool number T is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use tool number T in integer.		

7.52 COR-052 Sub-program number P should be integer

Alarm ID	COR–052 BGND-052	Alarm title	Sub-program number, P, should be integer
Description	Sub-program number P is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use the sub-program number P in integer.		

7.53 COR-053 Repeat count L should be integer

Alarm ID	COR–053 BGND-053	Alarm title	Repeat count L should be integer

Description	Repeat count L is not an integer.
Reason	Programming error.
Solution	Please check the NC program, and use the repeat count L in integer.

7.54 COR-054 Incompatible data type

Alarm ID	**COR–054** **BGND-054**	**Alarm title**	**Incompatible data type**
Description	Data format is incompatible with controller specifications.		
Reason	NC program is not compatible with the SYNTEC controller.		
Solution	Make sure that the NC program data format is compatible with Syntec controller specifications.		

7.55 COR-055 Tool length compensation H out of range

Alarm ID	**COR–055** **BGND-055**	**Alarm title**	**Tool length compensation H out of range**
Description	The tool length compensation H exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool length compensation H, is in the range of tool number.		

7.56 COR-056 G10 table index P is out of range

Alarm ID	**COR–056** **BGND-056**	**Alarm title**	**G10 table index P is out of range**
Description	G10's format is G10 L_ P_ R_; Different number L will correspond to different number P. i.e. L10 corresponding number P is tool number. Input P1000 means the 1000th tool. If that tool number doesn't exist, and controller will issue this alarm.		

Reason	Programming error.
Solution	Confirm G10 data table address number P is in reasonable range.

7.57 COR-057 Tool radius D out of range

Alarm ID	**COR–057** **BGND-057**	**Alarm title**	**Tool radius D out of range**
Description	The tool radius number D exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool radius number D is in the range of tool number.		

7.58 COR-058 Tool nose compensation D is out of range

Alarm ID	**COR–058** **BGND-058**	**Alarm title**	**Tool nose compensation D is out of range**
Description	The tool nose compensation D exceeds the range of tool number.		
Reason	Program error.		
Solution	Make sure that the tool radius compensation D is in the range of tool number.		

7.59 COR-059 Subprogram call H must be integer

Alarm ID	**COR–059** **BGND-059**	**Alarm title**	**Subprogram Call H must be integer**
Description	Subprogram call H is not an integer.		
Reason	Program error.		
Solution	Confirm the subprogram call H is an integer.		

7.60 COR-060 M99 return number P must integer

Alarm ID	COR–060 BGND-060	Alarm title	M99 return number P must integer
Description	The M99 return sequence number P is not an integer.		
Reason	Program error.		
Solution	Confirm the M99 return sequence number P is an integer.		

7.61 COR-061 Workpiece number is out of range

Alarm ID	COR–061 BGND-061	Alarm title	Workpiece number is out of range
Description	The number of work pieces exceeds the allowable range of the controller.		
Reason	Program error.		
Solution	Make sure the number of work pieces is in the allowed range of the controller.		

7.62 COR-062 Dwell skip source Q must be integer

Alarm ID	COR–062 BGND-062	Alarm title	Dwell skip source Q must be integer
Description	Dwell skip source Q is not an integer.		
Reason	Program error.		
Solution	Change the dwell skip source Q into an integer.		

7.63 COR-063 Dwell skip source Q out of range

Alarm ID	COR–063 BGND-063	Alarm title	Dwell skip source Q out of range

Description	Dwell skip source Q exceeds the allowed range.
Reason	Program error.
Solution	Make sure the dwell skip source Q is in the allowed range.

7.64 COR-064 Invalid P address

Alarm ID	**COR–064** **BGND-064**	**Alarm title**	**Invalid P address**
Description	Invalid P address.		
Reason	1. Address P is less than 0. 2. Address P is not integer. 3. Address P is out of range.		
Solution	1. Make sure address P is greater or equal to 0. 2. Make sure address P is integer. 3. Make sure address P is within the range.		

7.65 COR-065 A command format error

Alarm ID	**COR–065** **BGND-065**	**Alarm title**	**,A command format error**
Description	In NC program ,A command's format is wrong.		
Reason	Programming error.		
Solution	Please confirm ,A command format is one of the followings: 1. In the same block command as ,A, Moving instruction only includes one of x axis or z axis. i.e. G01 X10. ,A30. 2. Use ,A advanced function in two separate line. The first line will assign ,A angle, and the second line will assign X, Z and ,A at the same time. i.e. G01 ,A30. G01 X0. Z50. ,A45		

7.66 COR-066 Inc. axis command and abs. axis command conflict

Alarm ID	COR–066 BGND-066	Alarm title	Inc. axis command and abs. axis command conflict
Description	Both G91 and G90 are in the same line.		
Reason	Programming error.		
Solution	Decide to use incremental or absolute command in one line, and enter the correct command.		

7.67 COR-067 Arc center vector and radius conflict

Alarm ID	COR–067 BGND-067	Alarm title	Arc center vector and radius conflict
Description	The arc end point is not on the arc created by the arc starting point and the specify center.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.68 COR-068 Quiet Mode not support imperial Metric switch command

Alarm ID	COR–068 BGND-068	Alarm title	Quiet Mode not support imperial Metric switch command
Description	The single block command in Quiet mode operation cannot switch Metric/ imperial unit.		
Reason	Programming error.		
Solution	Please check the NC program.		

7.69 COR-069 Round corner and chamfer cmd. conflict

Alarm ID	COR–069 BGND-069	Alarm title	Round corner and chamfer cmd. conflict
Description	Chamfer command and round corner command are in the same line.		
Reason	Program error.		
Solution	Do not let chamfer command and round corner command exist in the same line.		

7.70 COR-070 Invalid G Code

Alarm ID	COR–070 BGND-070	Alarm title	Invalid G Code
Description	Enter incorrect G code to controller.		
Reason	Program error.		
Solution	Enter the valid G-code.		

7.71 COR-071 No main program name assigned

Alarm ID	COR–071 BGND-071	Alarm title	No main program name assigned
Description	The name of main program is not specified.		
Reason	The NC program is not loaded.		
Solution	Specify the name of main program.		

7.72 COR-072 Threading command exceeds max. cutting speed

Alarm ID	COR–072 BGND-072	Alarm title	Threading command exceeds max. cutting speed

Description	Threading command exceeds the maximum cutting speed.
Reason	Program error.
Solution	Decrease the cutting speed of threading.

7.73 COR-073 Tapping command exceed max. cutting speed

Alarm ID	**COR–073** **BGND-073**	**Alarm title**	**Tapping command exceed max. cutting speed**
Description	Tapping command speed exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of tapping.		

7.74 COR-074 Tool radius too big, path overcut

Alarm ID	**COR–074** **BGND-074**	**Alarm title**	**Tool radius too big, path overcut**
Description	1. During machining notch, the notch width is smaller than two times of tool radius. 2. During machining trapezoidal, the height of trapezoidal is less than two times of tool radius.		
Reason	Tool compensation causes path overcut.		
Solution	Please check the NC program, and decide whether this part of the machining should cancels tool radius compensation.		

7.75 COR-075 Exact stop wait timeout

Alarm ID	**COR–075** **BGND-075**	**Alarm title**	**Exact stop wait timeout**

Description	After 2 seconds after sending Exact stop (G09/G61) command, the difference between feedback and command exceeds allowable value (G01 is determined by Pr421~; G00 is determined by Pr481~).
Reason	Servo vibration.
Solution	1. Servo tuning. 2. Modify parameters Pr421~ or Pr481~.

7.76 COR-076 G04 dwell time cannot be negative

Alarm ID	**COR–076** **BGND-076**	**Alarm title**	**G04 dwell time cannot be negative**
Description	Input value of dwell time G04 is negative.		
Reason	Program error.		
Solution	Check the NC program, and enter a positive or zero value in G04 argument.		

7.77 COR-077 NURBS curve format is wrong

Alarm ID	**COR-077** **BGND-077**	**Alarm Title**	**NURBS curve format is wrong**
Description	G6.2 format is wrong.		
Reason	Input argument P, K, R is against allowable format.		
Solution	Refer the program manual to correct the command syntax.		

7.78 COR-078 NURBS curve system memory insufficient

Alarm ID	**COR-078** **BGND-078**	**Alarm Title**	**NURBS curve system memory insufficient**
Description	NURBS curve memory is insufficient.		

Alarm ID	COR-078 BGND-078	Alarm Title	NURBS curve system memory insufficient
Reason	System processing defective.		
Solution	Contact OEM Syntec.		

7.79 COR-079 Number of NURBS curve nodes more than the upper limit

Alarm ID	COR-079 BGND-079	Alarm Title	Number of Nurbs curve nodes more than the upper limit
Description	NURBS curve control point is over limit.		
Reason	NC file G6.2 curve is too long and cause using too many control point.		
Solution	Re-generate the CAM, and restrict the number of curve nodes less than 400.		

7.80 COR-080 Threading pitch is negative

Alarm ID	COR-080 BGND-080	Alarm Title	Threading pitch is negative
Description	Threading pitch distance can’t be negative value.		
Reason	Threading pitch distance setting error.		
Solution	Check and correct the configured threading pitch to be positive value.		

SYNTEC

7.81 COR-081 Use arc interpolation under tool offset function. Arc command doesn’t use assigned angle.

Alarm ID	COR-081 BGND-081	Alarm Title	Use arc interpolation under tool offset function. Arc command doesn’t use assigned angle.
Description	In tool offset function (G45~G48), G02 and G03 can only be 90 degree or 270 degree.		
Reason	In NC program’s tool offset function (G45~G48) block, all arc interpolation angle can only be 90 degree or 270 degree.		
Solution	Check if G45~G48 is 90 or 270 degree when using them in the arc interpolation (G02/G03). Note, IJK to specify circle center shall be used.		

7.82 COR-082 Tool compensation function and tool offset function can’t use at the same time

Alarm ID	COR-082 BGND-082	Alarm Title	Tool compensation function and tool offset function can’t use at the same time
Description	Can’t use tool offset function (G45~G47) and tool compensation function (G41, G42) or tool radius compensation at the same time.		
Reason	NC program edit error.		
Solution	Check NC program, and confirm tool offset function (G45~G47) and tool compensation function (G41, G42) are not used at the same time.		

7.83 COR-083 When pull tool back to pause point, and move axis position manually.

Alarm ID	COR-083 BGND-083	Alarm Title	When pull tool back to pause point, and move axis position manually.
Description	Before pull tool back to pause point, press stop and manually move tool.		

Alarm ID	COR-083 BGND-083	Alarm Title	When pull tool back to pause point, and move axis position manually.
Reason	Operation error.		
Solution	Please wait until tool is pulled exactly back to pause point, then execute the manual position movement.		

7.84 COR-084 P Argument unassigned when using the rapid drilling

Alarm ID	COR-084 BGND-084	Alarm Title	P Argument unassigned when using the rapid drilling
Description	When using rapid drilling function, must set P argument (bottom hole dwell time), or system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure the P Argument is assigned when using the G code of rapid drilling.		

7.85 COR-085 F Argument unassigned in Inverse Time Feed mode

Alarm ID	COR-085 BGND-085	Alarm Title	F Argument unassigned in Inverse Time Feed mode
Description	Under Inverse Time Feed mode, if F argument is not assigned in command line and system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure there is F command is given following in each line after Inverse Time Feed mode.		

7.86 COR-086 R and I arguments unassigned in G10 L1501 mode

Alarm ID	COR-086 BGND-086	Alarm Title	R and I arguments unassigned in G10 L1501 mode
Description	G10 L1501 argument only has argument R or argument I.		
Reason	Programming error.		
Solution	Check the NC program to ensure the R and I arguments are not used independently.		

7.87 COR-087 P and Q arguments must be different in G10 L1501 mode

Alarm ID	COR-087 BGND-087	Alarm Title	P and Q arguments must be different in G10 L1501 mode
Description	P and Q argument in G10 L1501 mode setting are the same.		
Reason	Programming error.		
Solution	Please check program, and confirm P and Q argument in G10 L1501 mode aren’t the same.		

7.88 COR-088 No time-axis command after the I argument in G10 L1501 mode

Alarm ID	COR-088 BGND-088	Alarm Title	No time-axis command after the I argument in G10 L1501 mode
Description	In spring machine dedicated machine's motion plan, if use I argument in G10 L1501, before the next G10 L1502 command, user can’t command P argument axis number (time axis) in G1 0L1501.		
Reason	Programming error.		
Solution	Check the NC program.		

7.89 COR-089 Main axis must be increment command in G10 L1501 mode

Alarm ID	COR-089 BGND-089	Alarm Title	Main axis must be increment command in G10L1501 mode
Description	G10 L1501 mode spindle must be incremental command.		
Reason	Programming error.		
Solution	Under G10 L1501 mode, confirm the command which sends to spindle is incremental command.		

7.90 COR-090 C-axis zero-speed check failed in G12.1 mode

Alarm ID	COR-090 BGND-090	Alarm Title	C-axis zero-speed check failed in G12.1 mode
Description	When start to switch polar coordinate, C axis must be totally stop.		
Reason	When C axis uses as spindle, but spindle not yet stop rotating.		
Solution	Switch the spindle to C-axis mode before activating polar coordinate.		

7.91 COR-091 Main axis synchronization – basic main axis number error

Alarm ID	COR-091 BGND-091	Alarm Title	Main axis synchronization – basic main axis number error
Description	When using spindle synchronous function G114.1, Pr4021, Pr4023, Pr4025 can’t be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4021/Pr4023/Pr4025 are correct.		

7.92 COR-092 Main axis synchronization – synchronization main axis number error

Alarm ID	COR-092 BGND-092	Alarm Title	Main axis synchronization – synchronization main axis number error
Description	When using spindle synchronous function G114.1, Pr4022, Pr4024, Pr4026 can't be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4022/Pr4024/Pr4026 are correct.		

7.93 COR-093 Main axis synchronization – spindle type error

Alarm ID	COR-093 BGND-093	Alarm Title	Main axis synchronization – spindle type error
Description	When using spindle synchronous and Superimposition function, setting wrong spindle motor type.		
Reason	Parameter setting error.		
Solution	Check 1791~Pr1800 to set correct spindle configurations by referring to HELP.		

7.94 COR-094 Under spindle load rigid tapping spindle rotation speed exceed

Alarm ID	COR-094 BGND-094	Alarm Title	Under spindle load rigid tapping spindle rotation speed exceed
Description	When using spindle load function to do rigid tapping, tool axis must synchronize with offset axis speed, and then do end surface taping according to user setting tapping speed. If tool axis speed is over spindle rotation speed maximum value, and system will issue this alarm.		
Reason	Command is over range that mechanical can bear.		

Alarm ID	COR-094 BGND-094	Alarm Title	Under spindle load rigid tapping spindle rotation speed exceed
Solution	1. Reduce the rotation speed (tapping speed) of tool axis. 2. Reduce the rotation speed of workpiece axis.		

7.95 COR-095 In polygon cutting function, basic spindle rotation speed ratio is wrong

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (P argument) must be bigger than zero.		
Reason	Programming error.		
Solution	Check the value of P argument in G51.2.		

7.96 COR-096 In polygon cutting function, synchronous spindle rotation speed ratio is wrong

Alarm ID	COR-096 BGND-096	Alarm Title	In polygon cutting function, synchronous spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (Q argument) must be bigger than zero.		
Reason	Programming error.		
Solution	Check the value of Q argument in G51.2.		

7.97 COR-097 Axis coupling function ON or OFF failed

Alarm ID	COR-097 BGND-097	Alarm Title	Axis coupling function ON or OFF failed
Description	Axis coupling function ON or OFF failed.		
Reason	Too much axis coupling groups is used.		
Solution	Check if the number of axis coupling groups is more than 16 (including the number of axis coupling group configured in parameter).		

7.98 COR-098 Parameter learning argument error

Alarm ID	COR-098 BGND-098	Alarm Title	Parameter learning argument error
Description	When using learning function, if argument (P, Q, R, K) setting is wrong, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure the arguments are in compliance with specifications.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

7.99 COR-099 Tapping learning condition is not matched

Alarm ID	COR-099 BGND-099	Alarm Title	Tapping learning condition is not matched
Description	Condition of tapping learning function is not matched		
Reason	When activating tapping learning function, if tapping condition (tapping depth, R point height, feedrate, rotation speed, and etc.) is different, it can't apply to same learning data. If force different tapping condition to use same learning data, it may misuse learning data, and cause tapping error or even tool break.		
Solution	Check tapping instruction in tapping learning function, and confirm all tapping instruction arguments are the same.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

7.100 COR-100 Unsupported G code command or option software is not activated

Alarm ID	COR-100 BGND-100	Alarm Title	Unsupported G code command or option software is not activated
Description	Different controllers will have correspond G code, but not all G code can use.		
Reason	1. This controller type may not support this G code command. 2. This controller type will not support serial bus spindle (C-Type) to use lathe G21, G33, G34, G78 commands. 3. This controller type will not support serial bus spindle (A-Type) to use lathe G32, G73, G76, G92 commands. 4. This controller type can support this G code command, but the option software function has not been purchased, which makes the G code unusable. 5. Loader path and Woodworking label path only support part of G codes: G00, G01, G02, G03, G04, G04.1, G09, G10, G17, G18, G19, G22, G23, G31, G52, G53, G54, G55, G56, G57, G58, G59, G59.x, G90, G91, G92. 6. The setting of Pr3802 is incorrect. This controller type does not support the G62 command.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

7.101 COR-101 Under Spindle synchronization – thread pitches of two spindle is inconsistent

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Description	When using spindle synchronize, if basic spindle and synchronous spindle pitch setting are different and will cause synchronous abnormal condition. And will issue this alarm for protection.		
Possible Cause	Parameter setting error.		
Solution	Check the spindle used in Pr161~180 to ensure the parameter configurations are the same.		

7.102 COR-102 Under Spindle synchronization – synchronization command duplicated or conflicted

Alarm ID	**COR-102** **BGND-102**	**Alarm Title**	**Under Spindle synchronization – synchronization command duplicated or conflicted**
Description	When spindle synchronizing, repeat G51.2, G114.1 or G114.3 commands.		
Possible Cause	Haven't use G113 to cancel G114.1 and G114.3, or use G50.2 to cancel G51.2, and again set a repeated G114.1, G114.3 or G51.2 commands.		
Solution	Cancel the G114.1 and G114.3 mode with G113, or cancel G51.2 mode with G50.2. Then execute the G114.1/G114.3/G51.2 commands to avoid the angle difference for repeated machining.		

7.103 COR-103 Invalid precision channel setting

Alarm ID	**COR-103** **BGND-103**	**Alarm Title**	**Invalid precision channel setting**
Description	Invalid precision channel setting.		
Possible Cause	Using G64, G62P_ or G120.1 P_Q_ to choose invalid precision channel. Currently can only choose: 1. G64、G62 P0~P9、G62/G64 P21 ~ P23 2. G120.1 P0、G120.1 P1 Q1 ~ G120.1 P3 Q3		
Solution	To select precision channel with the G64/G62 P_ or G120.1 P_ Q_. Check the NC program to ensure if any non-existing precision channel is selected.		

7.104 COR-104 Virtual axis function enabling failed

Alarm ID	**COR-104** **BGND-104**	**Alarm Title**	**Virtual axis function enabling failed**
Description	The virtual axis function failed to be functioned.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Possible Cause	1. The P and Q arguments in G10 L800 P_ Q_ or G10 L801 P_ Q_ are set incorrectly. a. No P, Q arguments. b. The P argument is set incorrectly and the value is invalid. c. The Q argument is set incorrectly, the value is invalid; or the corresponding axis is not set to the axis of the parameter table that has been turned on. 2. In the same machining, G10 L800 and G10 L801 appear at the same time. 3. The robot does not support the virtual axis function.		
Solution	1. Check the virtual axis commands to ensure the P and Q Arguments are configured. 2. Check the virtual axis commands to ensure the P Argument shall be within 100~999 and the Q Argument shall be within 1~16 (axis number) or 100~999 (axis name). 3. Check the axial direction (Q Argument) mapped with the virtual axis to ensure the parameter table is configured to the enabled axial direction. 4. Check if the G10 L800 and G10 L801 shall not be existent in the same NC program.		

7.105 COR-105 Axial coupling configuration error

Alarm ID	COR-105 BGND-105	Alarm Title	Axial coupling configuration error
Description	The axis coupling function is set incorrectly.		
Possible Cause	In G10 L900 P_ Q_ R_, the P and Q arguments are set incorrectly.		
Solution	1. Check the name or number of coupling axis is mapped to ensure the axial direction is enabled. 2. Chedk the specified coupling axis is not assigned to inclined axes control. 3. Check the specified coupling axis is not assigned to indexing axis.		

SYNTEC

7.106 COR-106 Prohibit to use G5.1 in G61/G63/G63.2 mode

Alarm ID	COR-106 BGND-106	Alarm Title	Prohibit to use G5.1 in G61/G63/ G63.2 mode
Description	1. G5.1 path smoothing. In G61, G63, G63.2 mode, it is forbidden to use G5.1 smoothing function, otherwise the system will issue an alarm; If G61, G63, G63.2 are enabled in G5.1 mode, the system will stop the smoothing function until it jumps out of G61, G63, G63.2 mode and then activate again. 2. G05 High Precision Contour Control. a. 10.116.16J previous version, in G61, G63, G63.2 mode, it is forbidden to use G05 High Precision Contour Control function, otherwise the system will issue an alarm; If G61, G63, G63.2 is enabled in G05 mode, no alarm will be issued, but the High Precision Contour Control function is disabled, It is necessary to command G05 again to activate. b. above 10.116.16K versions, no alarm will be issued, and the G05 High Precision Contour Control function will be disabled, and can be valid after G61, G63 and G63.2.		
Possible Cause	The cutting mode setting in the NC program is incorrect.		
Solution	1. Do not mix use G5.1 path smoothing with G61/G63/G63.2 functions. 2. In the version before 10.116.16J (included), do not mix use the G05 High Precision Contour Control with the G61/G63/G63.2 functions.		

7.107 COR-107 G5.1/G05 command format error

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Description	The G5.1 and G05 commands are in the wrong format.		
Possible Cause	1. The format of the G5.1 path smoothing command in the NC program is incorrect. 2. The G05 high-precision cutting mode command format in the NC program is incorrect.		

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Solution	Confirm the following command formats are correct not have these error: 1. G5.1 a. Q argument: None, more than 2, or less than 0. b. E argument: None or less than 0. 2. G05 a. System issue alarm when using G05 in following cases for each version: i. G05 P argument is not 10000 nor 0. ii. G05 E argument is not positive. b. Activate command G05 P10000 X0 Y0 Z0 α_ β_ in 10.116.36 or above versions: i. More than 5 axial directions are assigned. ii. The geometry axis argument not 0. iii. The rotary axis argument is configured to 0. iv. The axial direction of geometry axis is configured but this of rotary axis is not. v. The axis of the rotation axis is not set when the axis of the geometry axis is not set. vi. More than 2 axial directions of rotation axes are configured. vii. Any axial arguments is negative. c. In the version before 10.116.16B, there is the 4th axis command in addition to the block movement commands of X, Y, or Z axes after G05 is executed.		

7.108 COR-108 G10 L1501/L1502 command format error

Alarm ID	COR-108 BGND-108	Alarm Title	G10 L1501/L1502 command format error
Description	There is no NC program between G10 L1501 and G10 L1502.		
Possible Cause	The NC program was written incorrectly.		
Solution	Check the NC program to ensure the programming between G10 L1501 and G10 L1502 is correct.		

7.109 COR-109 The second software stroke limit command error, activating failure

Alarm ID	COR-109 BGND-109	Alarm Title	The second software stroke limit command error, activating failure
Description	The second software stroke limit (G22) command is written incorrectly, causing the second software stroke limit fail to activate.		
Possible Cause	The arguments written after the second software stroke limit (G22) command are wrong, and the same group parameters should exist at the same time.		
Solution	Check if the arguments in the same set after G22 are defined completely.		

7.110 COR-110 NC program of spring machine motion plan more than 500 lines

Alarm ID	COR-110 BGND-110	Alarm Title	NC program of spring machine motion plan more than 500 lines
Description	The number of machining lines between G10 L1501 and G10 L1502 exceeds 500 lines.		
Possible Cause	There are too many single blocks in the NC program.		
Solution	Check and refine the NC program.		

7.111 COR-111 Axis exchange function – axial configuration error

Alarm ID	COR-111 BGND-111	Alarm Title	Axis exchange function – axial configuration error
Description	Under the axis exchange function, the axial setting is incorrect.		
Possible Cause	The two axial's axis exchange parameters are set incorrectly.		
Solution	Check if Pr3721 and Pr3722 are mapped to physical axial directions (Pr21~) and to 2 different axis.		

7.112 COR-112 Axis exchange function – diameter/radius axis configuration error

Alarm ID	COR-112 BGND-112	Alarm Title	Axis exchange function – diameter/radius axis configuration error
Description	Under the axis exchange function, the diameter axis and radius axis are set incorrectly.		
Possible Cause	The parameters of the diameter and radius axis of the two axis exchange are set incorrectly.		
Solution	Check if the diameter and radius axis configurations (Pr281~) of the 2 axis to be exchanged is consistent.		

7.113 COR-113 Axis exchange function – ON or OFF timing error

Alarm ID	COR-113 BGND-113	Alarm Title	Axis exchange function – ON or OFF timing error
Description	When the axis exchange function is enabled, the enabled or disabled timing error.		
Possible Cause	When the cross-path axis is exchanged, the G04.1 waiting action is not performed.		
Solution	Check PLC to ensure no G04.1 waiting command is executed in more than 1 path.		

7.114 COR-114 Axis exchange function – path configuration error

Alarm ID	COR-114 BGND-114	Alarm Title	Axis exchange function – path configuration error
Description	When using the axis exchange function, the path setting is incorrect.		
Possible Cause	The axis used for axis exchange belong to multi-paths.		
Solution	Check path parameter (Pr701~) if the two axis to be exchanged are mapped to the multiple path.		

7.115 COR-115 G92.1 axial configuration error

Alarm ID	COR-115 BGND-115	Alarm Title	G92.1 axial configuration error
Description	When using the G92.1 rotation function, the axial setting is incorrect.		
Possible Cause	The parameter sets the first 3 axis of the path contains the rotary axis.		
Solution	Check the parameter to ensure the first 3 axis are linear axis.		

7.116 COR-116 Absolute coordinate system synchronization failed

Alarm ID	COR-116 BGND-116	Alarm Title	Absolute coordinate system synchronization failed
Description	Absolute coordinate system failed to synchronize when using C35.		
Possible Cause	Machine is not stationary or the program is continuously interpreted.		
Solution	To execute the synchronization of coordinate system in C35 mode shall ensure the machine is still and the program stops to interpret (i.e. Using M code shall turn C38 off).		

7.117 COR-117 M code not disabled in the interrupt subprogram

Alarm ID	COR-117 BGND-117	Alarm Title	M code not disabled in the interrupt subprogram
Description	If Pr3600 is set to n, when the interrupt type subprogram is used, it starts with M(n) and ends with M(n+1). If M(n) is used and M(n+1) is not used to end the function, it will cause an action error; therefore, this alarm is issued for protection.		
Possible Cause	The NC programming error.		
Solution	Check the NC program to ensure the M(n+1) command is given.		

7.118 COR-118 Prohibit G53 commands in tool tip control mode

Alarm ID	COR-118 BGND-118	Alarm Title	Prohibit G53 commands in tool tip control mode
Description	G53 command cannot be used in the tool point control mode.		
Possible Cause	1. The NC programming error. 2. The machine type is the tool point control mode.		
Solution	1. Please check the NC program, make sure that the G53 command is not within the validity range of G43.4 or G43.5. 2. Please check the NC program, make sure that the G53 command is not within the validity of G12.1. 3. If the machine configuration used is the tool point control mode, the G53 command cannot be used.		

7.119 COR-119 G10 L16 command format error

Alarm ID	COR-119 BGND-119	Alarm Title	G10 L16 command format error
Description	Since the command G10 L16 (virtual circle radius) does not support functions such as G05, G7.1, G12.1, G93, G95, and G05, therefore, an alarm will be issued if G10 L16 is used when the above command is issued.		
Possible Cause	1. The axis type is linear axis (i.e. G10 L16 X50., and X is set to a linear axis, this alarm is issued). 2. When using G10 L16 while command G05, G7.1, G12.1, G93, G95 and etc. functions are also given.		
Solution	1. Check Pr221~ to ensure the correct axis type. 2. G10 L16 and G7.1/G12.1/G93/G95/G05 shall not be executed at the same time.		

SYNTEC

7.120 COR-120 More than the maximum number of cutting synchronization axis

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Description	Different controller products can correspond to different maximum number of simultaneous cutting axis, and if this limit is exceeded, an alarm will be issued. (i.e. G01 X10. Y10 .Z10. Represents the number of simultaneous axes is 3).		
Possible Cause	Please refer to "System Information" under the controller HMI screen to check "Machine Properties" and "Machine Code". Please refer to the catalogue for each product specification. Following examples: 1. If "Machine Properties" is Mill, "Machine Code" is 200A-5, and the maximum number of axis is 9. 2. If the "Machine Properties" is Lathe, the "Machine Code" is 6B, and the maximum number of axis is 4.		
Solution	The number of simultaneous axis in the NC program shall be according to the product specifications.		

7.121 COR-121 LN operator shall be positive

Alarm ID	COR-121 BGND-121	Alarm Title	LN operator shall be positive
Description	The operands after the LN function cannot be negative.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

7.122 COR-122 The 1st operator of POW shall not be negative

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Description	The base operand of the POW function must not be a negative number.		

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

7.123 COR-123 Illegal STR2INT input or too long string

Alarm ID	COR-123 BGND-123	Alarm Title	Illegal STR2INT input or too long string
Description	The syntax of the STR2INT function is incorrect or the string is too long.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

7.124 COR-124 S code commands unsupported in the machine

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Description	This model does not support S code commands.		
Possible Cause	This model does not support S code commands.		
Solution	Replace a controller model that supports S codes.		

7.125 COR-125 T code commands unsupported in the machine

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Description	T code command is illegal.		

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Possible Cause	1. This model does not support T code commands. 2. The T code command is out of the T code range supported by the product.		
Solution	Replace a controller model that supports T codes.		

7.126 COR-126 H code commands unsupported in the machine

Alarm ID	COR-126 BGND-126	Alarm Title	H code commands unsupported in the machine
Description	This model does not support H code commands.		
Possible Cause	This model does not support H code commands.		
Solution	Replace a controller model that supports H codes.		

7.127 COR-127 D code commands unsupported in the machine

Alarm ID	COR-127 BGND-127	Alarm Title	D code commands unsupported in the machine
Description	This model does not support D code commands		
Possible Cause	This model does not support D code commands		
Solution	Replace a controller model that supports D codes.		

7.128 COR-131 Too much M/T code in a block

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Description	The total number of M codes and T codes in the same block exceeds 20.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Possible Cause	NC programming error.		
Solution	Check the NC program to ensure the total number of M and T codes are not more than 20 in one single block.		

7.129 COR-132 Illegal character in the program name

Alarm ID	COR-132 BGND-132	Alarm Title	Illegal character in the program name
Description	When open a file using Macro, the specified file name contains illegal characters.		
Possible Cause	NC programming error.		
Solution	Check the specified file names.		

7.130 COR-133 Command unsupported in Three-points arc interpolation mode

Alarm ID	COR-133 BGND-133	Alarm Title	Command unsupported in Three-points arc interpolation mode
Description	This command is not supported in the three-point arc interpolation mode (G02.4, G03.4)		
Possible Cause	1. The tool radius compensation function is not turned off before using this function. 2. It is not supported in G62 cutting mode. 3. In the three-point arc interpolation mode (G02.4, G03.4), A, C, and R commands are not supported.		
Solution	Check the NC program to ensure there is no command described in the Possible Cause.		

7.131 COR-134 Three-points arc interpolation command format error

Alarm ID	COR-134 BGND-134	Alarm Title	Three-points arc interpolation command format error
Description	The three-point arc interpolation (G02.4, G03.4) command format is wrong.		
Possible Cause	The three-point arc interpolation (G02.4, G03.4) command is regarded as a group of two lines, which can be specified continuously. The end point of the previous arc is the starting point of the next arc, but the F command can only be in the odd line. This alarm is issued if the total number of lines in the command is odd or the number of lines in the F command is even.		
Solution	Check the three-point arc interpolation G02.4/G03.4 command format in NC program.		

7.132 COR-135 Read/write command format error for R value

Alarm ID	COR-135 BGND-135	Alarm Title	Read/write command format error for R value
Description	SETRREGBIT, READRREGBIT bring in an argument with type error or range error.		
Possible Cause	1. SETRREGBIT (R value number, specify BIT, on or off) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31. c. If the third argument is not 0 (off) or 1 (on). 2. READRREGBIT (R value number, specify BIT) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31.		
Solution	Correct the argument type or range in the commands, SETRREGBIT and READRREGBIT		

7.133 COR-136 Please reboot the controller when axis tuning failed

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Description	After a certain serial bus axial tuning fails, without rebooting, cycle start is triggered.		

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Possible Cause	The serial bus axial tuning failed.		
Solution	Reboot controller.		

7.134 COR-137 Path synchronization waiting's P-argument sequence error

Alarm ID	COR–137 BGND-137	Alarm title	Path synchronization waiting's P-argument sequence error
Description	The machining multi-path program uses the path synchronization waiting (G04.1) to wait for the synchronization action: 1. When the Q argument is not selected, when two programs go to the position of the path synchronization waiting (G04.1) at the same time, if the P arguments substituted are different, this alarm is issued for reminder. 2. When the next Q argument (decimal) specifies the path waiting for each other, when the two programs go to the position of the path synchronization waiting (G04.1) at the same time, and wait for each other, but the P arguments substituted are different, this alarm is issued for reminder.		
Possible Cause	1. In the NC program of multiple paths, the order of P arguments is abnormal or the number is incorrect. 2. Machining is performed using M99, but in each program, the path without Q argument is synchronously waiting (G04.1), and the number is different. 3. Machining is performed using M99, but in each program, the path with the same Q argument is synchronously waiting (G04.1), and the number is different. 4. The two paths “wait for each other”, but the P argument substituted are not the same. i.e. G04.1 P1 Q123 under the first path and G04.1 P2 Q124 under the second path.		
Solution	Please check the number of Synchronization wait between paths (G04.1) and the order of the assigned value for argument P.		

7.135 COR-138 Read/write command format error at the I/O/A point

Alarm ID	COR-138 BGND-138	Alarm Title	Read/write command format error at the I/O/A point
Description	The system provides 512 I/O/A points, the input I/O/A point number shall be within 0~511 when using the commands, SETDO, SETABIT, READDI, READDO, and READABIT.		
Possible Cause	NC Programming error.		
Solution	Check if there is any of I/O/A point number in the commands, SETDO, SETABIT, READDI, READDO, and READABIT in the NC program is over the range of 0 ~ 511.		

7.136 COR-139 Polynomial solution error

Alarm ID	COR-139 BGND-139	Alarm Title	Polynomial solution error
Description	Determine the polynomial cannot be derived to solutions in case of the alarm.		
Possible Cause	Numerical processing calculation errors cause root failure.		
Solution	Check the value of system data no. 321, and provide it to the controller OEM Syntec for solutions.		

7.137 COR-140 Invalid high-precision contour control mode using

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Description	1. Turn on G05 high-speed high-precision mode in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Possible Cause	1. In the RTCP/STCP mode, turn on the G05 high-precision contour control mode with commands, such as G05 P10000. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		
Solution	1. Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-precision contour control mode. If (1), remove the command to turn on the G05 high-precision contour control mode in the RTCP/STCP mode. If (2), turn off the RTCP/STCP mode before turning on the G05 high-precision contour control mode. 2. When the high-precision contour control mode is enabled during processing, do not use the single block stop C40 at the same time. 3. When the STCP mode is enabled during processing, do not use the single block stop C40 at the same time.		

7.138 COR-141 G68.3 command format error

Alarm ID	COR-141 BGND-141	Alarm Title	G68.3 command format error
Description	[command format] G68.3 X_ Y_ Z_ R_; // The origin and z-axis rotation angle in the characteristic coordinate system. G68.3 P1 X_ Y_ Z_; // The origin of the characteristic coordinate system, and the coordinate system is determined with the tool rotation angle.		
Possible Cause	G68.3 command format, X, Y and Z are all exist or non-exist at the same time.		
Solution	Check if G68.3 command format is correct.		

7.139 COR-142 Spindle synchronization – K Argument error

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Description	K Argument error while using Spindle synchronization function.		
Possible Cause	The inputted K Argument is not within the valid range.		
Solution	Input a K Argument within the valid range to enable the Spindle synchronization.		

7.140 COR-143 Programmable data input specified axis does not exist

Alarm ID	COR-143 BGND-143	Alarm Title	Programmable data input specified axis does not exist
Description	In G10 L1501 mode, the P argument is set incorrectly.		
Possible Cause	The axial direction substituted by the P Argument is inexistent.		
Solution	Check Pr21~ to ensure the axial direction is ON.		

7.141 COR-144 Path synchronization waiting Q Argument content error

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Description	When the machining multiple path program applies the G04.1 to wait for synchronization, the Q Argument assigns the paths to be waited for each other (decimally). The alarm is used to remind the users if the Q Argument type error or the assigned path is inexistent.		

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Possible Cause	1. The Q Argument is not a positive integer (negative number, decimal point and zero are all illegal). 2. The path assigned by the Q Argument is inexistent. i.e. Q24 is used to assign the 2nd and 4th path to wait for each other. However, the Pr731=3 and there are only 3 CNC main system paths. 3. The Q Argument includes 0 when assigning the paths. i.e. Q103. 4. The paths assigned by the Q Argument do not include this in which the command is. ie. In the 2nd path, the G04.1 P1 Q13 assigns the 1st and 3rd paths to wait for each other, but it does not assign the path in which it is.		
Solution	Check the Path synchronization waiting (G04.1) in the program to ensure the type and the assigned path are correct for a given Q Argument.		

7.142 COR-145 Failed to activate Spindle positioning

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Description	Spindle positioning failed.		
Possible Cause	1. Incorrect Spindle ID is assigned. 2. The Spindle is operating in the tapping mode when the Spindle positioning function is ON. 3. After starting the positioning, C61 is OFF so that the positioning is broken.		
Solution	1. Check the Spindle parameter configuration. 2. Check the NC program to ensure the execution of Spindle positioning in non-tapping mode. 3. Check the PLC to ensure C61 is ON.		

7.143 COR-146 Single block argument type error

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Description	Argument type error in a block.		

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Possible Cause	The two following situations may trigger this alarm: 1. Input float value to an argument required integer value. 2. Input integer value to an argument required float value.		
Solution	Correct the argument type in the block.		

7.144 COR-147 Spindle is not enabled when path machining

Alarm ID	COR-147 BGND-147	Alarm Title	Spindle is not enabled when path machining
Description	The machining Spindle assigned by the path is not enabled when executing the commands related to the Spindle.		
Possible Cause	When executing the G33/G34/G63/G74/G84, the Spindle assigned by the R791~R794 is not enabled.		
Solution	1. Check if the PLC switches R791~R794 values correctly in the block and whole NC program about the alarm. 2. Check if the Pr1621~Pr1628 configurations are correct (cooperate with R791~R794).		

7.145 COR-148 The use of tool retract function error

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Description	When using the tool retract function, a given improper argument results in the function cannot be executed.		
Possible Cause	1. In the Tilted working plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned axial position retraction. 2. In the non-Tilted working plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned retraction along the tool vector.		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Solution	1. In the slopping plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, apply G10.6 R_ to assign the tool retraction function. 2. In the non-slopping plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, apply G10.6 X_ to assign the tool retraction function.		

7.146 COR-149 Tilted working plane machining tool alignment P Argument over range

Alarm ID	COR-149 BGND-149	Alarm Title	Tilted working plane machining tool alignment P Argument over range
Description	P Argument is over range in Tilted working plane machining tool alignment mode.		
Possible Cause	P Argument is not within 0~2.		
Solution	Do not input the P Argument or input the correction configuration for the P Argument in Tilted working plane machining tool alignment mode.		

7.147 COR-151 1st rotation axis entering illegal range

Alarm ID	COR-151 BGND-151	Alarm Title	1st rotation axis entering illegal range
Description	1st rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1st rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3009 and Pr3010 are configured correctly. The determination of such two configurations is related to Pr3007. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

7.148 COR-152 2nd rotation axis entering illegal range

Alarm ID	COR-152 BGND-152	Alarm Title	2nd rotation axis entering illegal range
Description	2nd rotation axis entering illegal range		
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3011 and Pr3012 are configured correctly. The determination of such two configurations is related to Pr3008. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

7.149 COR-153 Tool direction unknown

Alarm ID	COR-153 BGND-153	Alarm Title	Tool direction unknown
Description	Tool direction unknown.		
Possible Cause	5-axis configurations and machine mechanism is incompatible.		
Solution	The tool cannot reach the destination. It may be caused by the incompatible 5-axis configurations and machine mechanism. Please check all 5-axis configurations.		

7.150 COR-154 No 5-axis function

Alarm ID	COR-154 BGND-154	Alarm Title	No 5-axis function
Description	No 5-axis function.		
Possible Cause	Pr3001 is not configured when executing G53.1 tool alignment command.		
Solution	Check if Pr3001 is configured to 0. If yes, configure the other non-zero values based on the 5-axis mechanism type and reboot.		

7.151 COR-155 5-axis tool direction error

Alarm ID	COR-155 BGND-155	Alarm Title	5-axis tool direction error
Description	5-axis tool direction error.		
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd rotation axis (Pr3005 and Pr3006) configuration error.		
Solution	Check if the Pr3002 is configured correctly, or if the Pr3005 or Pr3006 is configured correctly. The alarm will be triggered in case the 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

7.152 COR-156 5-axis axial direction error

Alarm ID	COR-156 BGND-156	Alarm Title	5-axis axial direction error
Description	5-axis axial direction error.		
Possible Cause	Incorrect configurations are mapped to the axial direction parameters of 5 axis.		
Solution	Check if each axial direction is configured completely (Pr21~), if Pr3005, Pr3006, Pr3007 and Pr3008 is configured correctly, or if the axis name (Pr321~) is mapped to Pr3005 and Pr3006.		

7.153 COR-157 Incompatible direction of 5-axis tool direction and this of rotation axis

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Description	Incompatible direction of 5-axis tool direction and this of rotation axis.		
Possible Cause	1. The 2nd axial direction and the tool direction are the same in the Spindle type. 2. The 1st axial direction and the tool direction are the same in the table type. 3. The 1st axial direction and the tool direction are the same in the hybrid type.		

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Solution	Check if the tool direction and the rotation axial direction are the same depended on the used 5-axis mechanism type.		

P.S. Valid version of COR-157 : before 10.118.41M, 10.118.47 (included).

7.154 COR-158 Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode

Alarm ID	COR-158 BGND-158	Alarm Title	Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode
Description	Since the G43.5 mode specifies the tool attitude based on the tool vector I, J and K, it shall not be executed for the 1st and 2nd rotation axis commands which can also specify the tool attitude.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the movement commands of the 1st and 2nd rotation axis are over the valid range in the G43.5 mode.		

7.155 COR-159 Illegal tool vector

Alarm ID	COR-159 BGND-159	Alarm Title	Illegal tool vector
Description	In NC program, a movement block assigns an incorrect tool vector.		
Possible Cause	Programming error. i.e. Execute G01 X_ Y_ Z_ I0 J0 K0 in the G43.5 mode, and the I0 J0 K0 refers to the 0 vector, 0 vector is illegal.		
Solution	Check the NC program to ensure the assigned tool vector is correct in the block indicated by the alarm line number		

7.156 COR-160 5-axis mechanism chain switched when the 5-axis function is ON

Alarm ID	COR-160 BGND-160	Alarm Title	5-axis mechanism chain switched when the 5-axis function is ON
Description	In NC program, the command G10 L5000 [P_] is executed to switch the 5-axis mechanism when turning on the 5-axis function (RTCP or the tool vector alignment on the characteristic coordinate system is completed).		
Possible Cause	Programming error. i.e. Execute G10 L5000 [P_] in the G43.4 mode.		
Solution	Check the NC program to ensure the 5-axis function is not turned on in the block indicated by the alarm line number.		

7.157 COR-161 Selected 5-axis mechanism chain is not ON

Alarm ID	COR-161 BGND-161	Alarm Title	Selected 5-axis mechanism chain is not ON
Description	In NC program, the 5-axis mechanism chain parameters assigned by the command G10 L5000 [P_] is not ON.		
Possible Cause	Programming error. i.e. The assigned 5-axis mechanism chain is not configured correctly when executing a command G10 L5000 [P_].		
Solution	Check the 5-axis mechanism chain parameters to ensure the assigned 5-axis mechanism chain function is turned on correctly: 1. 1st set: Pr3001 2. 2nd set: Pr3101 3. 3rd set: Pr5501 4. 4th set: Pr5601		

7.158 COR-162 4-axis RTCP configuration error

Alarm ID	COR-162 BGND-162	Alarm Title	4-axis RTCP configuration error
Description	The controller will issue the alarm in case the 4-axis RTCP configuration is incorrect.		
Possible Cause	The 5-axis mechanism parameters, Pr3001, Pr3101, Pr5501 and Pr5601 are configured to 1~3 in case the specific 4-axis tool tip control function (option-29) is turned on but the tool tip control function (option-12) is not.		
Solution	1. Configure the 5-axis mechanism parameters to 4 or 5 correctly. 2. Turn on the tool tip control function (option-12).		

7.159 COR-163 Multi-kinematic chain command Q Argument setting error.

Alarm ID	COR-163 BGND-163	Alarm Title	Multi-kinematic chain command Q Argument setting error.
Description	Command G10 L5000P_ Q_, Q argument range error.		
Possible Cause	Command G10 L5000P_ Q_, Q argument range error.		
Solution	While using G10 L5000P_ Q_, check Q argument to be within 0~4, and is a integer.		

7.160 COR-164 Multi-kinematic chain command related 5-Axis mechanism setting error.

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Description	Command G10 L5000P_ Q_ specified the 5-Axis kinematic chain, and the 5-Axis mechanism parameter setting error.		
Possible Cause	While executing G10 L5000P_ Q_, Q argument is given, but the 5-Axis mechanism parameter of the designated 5-Axis kinematic-chain is not a spindle-type 5-Axis machine.		

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Solution	Please check the designated 5-Axis kinematic-chain. The 5-Axis mechanism configuration parameter must be a spindle-type 5-Axis machine. 1. The first group : Pr3001 is 1. 2. The second group : Pr3101 is 1. 3. The third group : Pr5501 is 1. 4. The fourth group : Pr5601 is 1.		

7.161 COR-165 Multi-kinematic chain command not illegal.

Alarm ID	COR-165 BGND-165	Alarm Title	Multi-kinematic chain command not illegal.
Description	Command G10 L5000P_ Q_ is used for switching 5-Axis kinematic chain, and only provides partial 5-Axis mechanism function command.		
Possible Cause	G10 L5000 P_ Q_ command, the Q argument is set to 2~4 (not the first group of sub-kinematic chain), and only supports the following 5-Axis machine function command. 1. RTCP: G43.4. 2. RTCP: G43.5. 3. Tilted working plane : G68.2 + Tool alignment functions. 4. Tilted working plane : G68.3. Notice: Tool alignment functions include G53.1, G53.3, G53.6, ...		
Solution	When using the G10 L5000 P_ Q_ command to switch multi-kinematic chains, please use the supported 5-Axis machine function command.		

7.162 COR-166 Characteristic Coordinate System Option not supported

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Description	Option13 (Characteristic Coordinate System Option) was not active, therefore the CNC couldn't execute relevant commands.		

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Possible Cause	One or more commands below were given while the Option13 (Characteristic Coordinate System Option) was inactive: 1. G68.2, G68.3 2. G53.1, G53.3, G53.6 3. Other commands that are relevant to Option13 (Characteristic Coordinate System Option)		
Solution	1. Activate Option13 (Characteristic Coordinate System Option) 2. Avoid using the commands listed above		

7.163 COR-167 File name conflicts in NcFiles

Alarm ID	COR-167 BGND-167	Alarm Title	Program File name conflicts in NcFiles
Description	Program File name conflicts in NcFiles folder.		
Possible Cause	When Pr3220 is set to 1, "non-main system", "PLC axis", "pre- and post-processing auxiliary programs" restrict calling files from the Macro folder, and prohibit the NcFiles folder from having the same file name.		
Solution	1. Set Pr3220 to 0. 2. Remove the conflicting files in the NcFiles folder or change the file name.		

7.164 COR-168 Illegal tool compensation

Alarm ID	COR-168 BGND-168	Alarm Title	Illegal tool compensation.
Description	Illegal tool compensation		
Possible Cause	1. Trying to use non-zero tool length or tool radius compensation value on a machine type which doesn't support tool compensation		

Alarm ID	COR-168 BGND-168	Alarm Title	Illegal tool compensation.
Solution	1. Set tool compensation value as zero 2. Turn off tool compensation related functions		

7.165 COR-169 Overlap on the same axis

Alarm ID	COR-169 BGND-169	Alarm Title	Overlap on the same axis
Description	Overlap on the same axis.		
Possible Cause	Using G1.10 with argument Q on the same axis with two consecutive blocks.		
Solution	Please check NC Program and avoid overlapping on the same axis with two consecutive blocks.		

7.166 COR-170 G43.4 L2 illegal argument value

Alarm ID	COR-170 BGND-170	Alarm 标题	【G43.4 L2 illegal argument value】
说明	When using G43.4 L2. E_ R_, Illegal argument value is assigned		
可能原因	Illegal argument value 1. R value is negative 2. E value is not in the range of 0.001 to 179.999		
排除方法	Please check the E_ R_ argument values		

7.167 COR-171 Non-linear kinematic transform is not allowed in inclined axes control

Alarm ID	COR-171 BGND-171	Alarm title	【Non-linear kinematic transform is not allowed in inclined axes control】
Description	When inclined axes control is enabled, it can't work with RTCP related function.		
Possible Cause	using following function, when inclined axes control is enabled. 1. G43.4 Rotate tool center point Type 1. 2. G43.5 Rotate tool center point Type 2. 3. G68.2,G68.3 Titled working plane machining with G43,G44 Tool length compensation. 4. G41.1, G42.1 tangential control.		
Solution	Don't use these function at program when inclined axes control is enabled.		

7.168 COR-180 Invalid parameters setting for tangential control

Alarm ID	COR-180	Alarm Title	【Invalid parameters setting for tangential control】
Description	Invalid command or parameters setting for tangential control.		
Reason	1. Feed axes for tangential control do not exist. 2. Specified axis for tangential control does not exist. 3. Do not support specified axis type for tangential control. 4. Using spindle as rotary axis for tangential control but axis is not in C-axis mode. 5. Rotation direction of the tangential control axis is not yet set. 6. Location of the tangential control axis is not yet set. 7. G41.1, G42.1 command: a. Value of the Q argument is out of range. b. Value of the E argument is out of range. c. Value of the P argument is neither 0 or 1.		
Solution	1. X or Y or Z axis do not exist. Please ensure axis exist(Pr21~) and axis name(Pr321~) correct. 2. Axis for tangential control(Pr5801) should be the existing axis(Pr21~). 3. Set axis type(Pr221~) of the tangential control axis to 2 or 5. 4. Switch spindle mode to C-axis mode(C63) before start tangential control. 5. Set rotation direction of the tangential control axis(Pr5802) to 1, 2, or 3. 6. Set location of the tangential control axis(Pr5803) to 1 or 2. 7. Please refer to the G41.1/G42.1 command description.		

7.169 COR-181 Tangential control plane conflict

Alarm ID	COR-181	Alarm Title	【Tangential control plane conflict】
Description	Work plane is conflict with the tangential control plane.		
Reson	After tangential control function enabled, user has specified conflict work plane.		
Solution	Reassign tangential control axis and rotation direction of the tangential control axis or remove the conflict workplane specified.		

7.170 COR-182 Unsupport functions under tangential control

Alarm ID	COR-182	Alarm Title	【Unsupport functions under tangential control】
Description	When tangential control function is enabled, do not support some other functions.		
Reason	1. Put movement command to tangential control rotary axis. 2. Using tool compensation command such as G41, G42, G43.4, G43.5. 3. Using G02/G03 L_ command to do multiple turns of circle cutting. 4. Using G10L1150 to write parameters of the tangential control mechanism. 5. Put chopping command to axes under tangential control. 6. Turn on the G05 high-speed high-precision mode with command, such as G05 P10000. 7. Using G53 to do machine coordinate orientation. 8. Using G10L1500 to do rotary axis auxiliary brake.		
Solution	Don't use these functions until reset or tangential control function disabled(G40.1 or G49).		

7.171 COR-201 Program file not exist

Alarm ID	COR–201 BGND-201	Alarm title	Program file not exist
Description	The specified program does not exist.		
Possible Cause	The specified program does not exist.		
Solution	Ensure that program file exists.		

7.172 COR-202 Communication link error

Alarm ID	COR–202 BGND-202	Alarm title	Communication link error
Description	Communication link is dropped.		
Possible Cause	The transmission communication link is dropped.		
Solution	Reconnect a good transmission communication link.		

7.173 COR-203 Illegal NC file format

Alarm ID	COR–203 BGND-203	Alarm title	Illegal NC file format
Description	NC program format not valid, as a result, the system cannot fully interpret the NC program.		
Possible Cause	1. The NC file is in macro format. 2. Use M98 to call a multi-path subprogram (including $1 and $2), and the size of the subprogram is greater than 60KB(60000bytes).		
Solution	1. Update the controller to 10.114.50I or the later version. 2. For M98 application, reduce the size of subprogram. Or, split it into individual single-path subprograms, then call each of them by coordinate respectively.		

7.174 COR-204 File size too large

Alarm ID	COR–204 BGND-204	Alarm title	File size too large
Description	Program file is too large.		
Possible Cause	If the size of MACRO program is larger than 60KB(60000bytes) and use the syntax with scope statement. The syntax with scope statement contains: IF, CASE, REPEAT, FOR, WHILE		

Solution	1. Reduce the program size, or split program into two subprograms. 2. Remove all the syntax with scope statement.

7.175 COR-205 File content is empty

Alarm ID	**COR–205** **BGND-205**	**Alarm title**	**File content is empty**
Description	After controller loads the program, the file content found to be null.		
Possible Cause	Program loading error or CF card damaged.		
Solution	Reload program or replace CF card.		

7.176 COR-206 Loading page lock failure

Alarm ID	**COR–206** **BGND-206**	**Alarm title**	**Loading page lock failure**
Description	New NC program fails to required the system to distribute loading page.		
Possible Cause	Lack of memory when multi-system executes large-size program.		
Solution	Please upgrade to 9.240.27/ 9.242.5/10.104.36/10.111.41/10.113.5 or contact OEM Syntec.		

7.177 COR-207 Specified sequence number not found

Alarm ID	**COR–207** **BGND-207**	**Alarm title**	**Specified sequence number not found**
Description	Specified sequence number is not found.		
Possible Cause	Programming error.		
Solution	Use a sequence number within the NC program range.		

7.178 COR-208 Cannot use jump statement in sequential file

Alarm ID	COR–208 BGND-208	Alarm title	Cannot use jump statement in sequential file
Description	Using jump command to execute sequential file.		
Possible Cause	Use the jump command when executing the sequence file.		
Solution	Do not use jump command to execute sequential file.		

7.179 COR-209 File format error

Alarm ID	COR–209 BGND-209	Alarm title	【File format error】
Description	Invalid File Format.		
Possible Cause	1. It is not a MACRO format file. For example, using APP syntax to call MARCO G200, G200 APP "appname", but this G200 file is not declared as a MACRO format file. 2. It is not a ISO format file.		
Solution	Please check the NC program, which should be written according to the file opening requirements.		

7.180 COR-250 Wrong mechanism axial setting

Alarm ID	COR-250 BGND-250	Alarm title	Wrong mechanism axial setting
Description	The axial setting of the mechanism is wrong.		
Possible Cause	The number of axis and the axis name required for the corresponding mechanism conversion are not set.		
Solution	Set sufficient number of axis (Pr21~), and required axis names (Pr321~).		

7.181 COR-251 The joint is not set to rotate in the right/left-hand direction

Alarm ID	**COR-251** **BGND-251**	**Alarm title**	**【The joint is not set to rotate in the right/left-hand direction】**
Description	The joint is not set to the direction of rotation as the right hand, left hand rule.		
Possible Cause	Pr4141~Pr4150 are not set to the right-hand rule or the left-hand rule.		
Solution	Set the rotation direction of each joint, 0 is not allowed.		

7.182 COR-252 The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position

Alarm ID	**COR-252** **BGND-252**	**Alarm title**	**【The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position】**
Description	The 2D mechanism is converted and the target position cannot be reached. Please enter a reasonable target position.		
Possible Cause	The program coordinate position exceeds the limit of the machine.		
Solution	The position is not reasonable. Please re-check the NC program.		

7.183 COR-253 Unworkable parallel mechanism posture

Alarm ID	**COR -253** **BGND-253**	**Alarm title**	**【Unworkable parallel mechanism posture】**
Description	Parallel mechanism posture that is not feasible.		
Possible Cause	Currently at a parallel mechanism posture that is not feasible.		

Alarm ID	COR -253 BGND-253	Alarm title	【Unworkable parallel mechanism posture】
Solution	After Reset, use the axial movement mode to leave this posture, or modify the target point.		

7.184 COR-301 OPEN command format error

Alarm ID	COR-301 BGND-301	Alarm title	【OPEN command format error】
Description	OPEN command format error [command format] OPEN ("file name", "write file mode") (i) Designated as '**a**': Represent **retains** the original file content and **continuously** outputs the new data in the text file. (ii) Not specified or designated as '**w**': Represents the **emptying** of the original content and **re-outputs** the new material in the text file.		
Possible Cause	"Write mode" in the OPEN command format specifies an error. This alarm is issued if the string is specified as a string other than 'a' or 'w'.		
Solution	Set "FileWrite mode" to "a" or "w" according to the requirement of FileWrite.		

7.185 COR-302 G10 L1150 command format error

Alarm ID	COR-302 BGND-302	Alarm title	【G10 L1150 command format error】
Description	G10 L1150 command format error.		
Possible Cause	G10 L1150 parameter numbering (P argument) is a non-integer or the input is out of range or not entered. G10 L1150 parameter value (R argument) is out of range or not entered.		
Solution	Set the G10 L1150 parameter numbering and values (P and R arguments) correctly.		

7.186 COR-303 The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)

Alarm ID	COR-303 BGND-303	Alarm title	【The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)】
Description	The basic spindle uses the spindle synchronization (G114.1, G51.2) commands in the position control mode.		
Possible Cause	When using this function, the basic spindle cannot be in position control mode.		
Solution	Write the NC program or PLC correctly so that the principal axis leaves the position control mode before the same period (C63).		

7.187 COR-304 Thread cutting geometrical axis name error

Alarm ID	COR-304 BGND-304	Alarm title	【Thread cutting geometrical axis name error】
Description	Thread cutting (G33, G34) command format is incorrect. Only the geometric axis names such as X, Y, Z, X1, Y1, Z1, etc. are supported.		
Possible Cause	When the thread cutting (G33, G34) does not have X, Y, Z, X1, Y1, Z1... and other geometric axis arguments, this alarm is issued.		
Solution	Adjust the axis names (Pr321~) of the geometrical axis to XYZ, X1/Y1/Z1...		

7.188 COR-305 Relative position input method is forbidden in current mode

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Description	The mode in which the system is currently running cannot be used with the G91 incremental command.		

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Possible Cause	G43.5 cannot be used with the G91 incremental command function: G43.5 determines the tool attitude through the tool vectors I, J, and K. The tool attitude is expressed only in absolute quantities.		
Solution	Check the NC program to confirm that G91 was not executed in G43.5 mode and G43.5 was not executed in G91 mode		

7.189 COR-306 Advanced look-ahead function without the specified M code

Alarm ID	COR-306 BGND-306	Alarm title	【Advanced look-ahead function without the specified M code】
Description	The advanced look-ahead function is enabled, and the specified M code is not found after the complete NC program is interpreted.		
Possible Cause	1. Pr3599 is not set. 2. The NC programming error.		
Solution	1. Confirm whether the setting of Pr3599 is correct. 2. Check the NC program to confirm that an command has been given to speedily predecode M code (Pr3599).		

7.190 COR-307 Advanced look-ahead of M code, wrong designation of argument P

Alarm ID	COR-307 BGND-307	Alarm title	【Advanced look-ahead of M code, wrong designation of argument P】
Description	The advanced look-ahead M code (Pr3599) must specify the P argument.		
Possible Cause	When using the advanced look-ahead function, the P argument is not specified or the P argument is not an integer		
Solution	Check the NC program and confirm that there is a P argument and specify a 4-digit code subprogram.		

7.191 COR-308 Insufficient system memory, increase a single step displacement for advanced look-ahead function

Alarm ID	COR-308 BGND-308	Alarm title	【Insufficient system memory, increase a single step displacement for advanced look-ahead function】
Description	The system look-ahead memory is insufficient, and the subsequent move command cannot be interpreted and is stuck.		
Possible Cause	There are too many very short blocks in the program, so the interpretation conditions cannot be met.		
Solution	Check the NC program and increase a single step displacement amount appropriately.		

7.192 COR-309 Advanced look-ahead function failed

Alarm ID	COR-309 BGND-309	Alarm title	【Advanced look-ahead function failed】
Description	The advanced look-ahead function cannot be executed correctly.		
Possible Cause	1. The interpolation mode is incorrect during advanced look-ahead. 2. The contents of the look-ahead NC program cannot be stored correctly.		
Solution	1. Check if the subprogram of the look-ahead NC programs contains commands other than G01, G02, and G03. 2. Check if disk space is adequate.		

7.193 COR-310 Look-ahead NC programs failed

Alarm ID	COR-310 BGND-310	Alarm title	【Look-ahead NC programs failed】
Description	Look-ahead NC program cannot be executed correctly.		

Alarm ID	COR-310 BGND-310	Alarm title	【Look-ahead NC programs failed】
Possible Cause	1. The condition of the look-ahead NC program does not match: a. The version of the advanced look-ahead, the interpolation time, the number of axes, do not match. b. There is an abnormal interruption in the look-ahead, resulting in incomplete content of the look-ahead. c. To look-ahead the axis of movement in the subprogram, enter the advanced look-ahead M code in the main program. The coordinates before (i.e. M298) should be the same. (i.e. if the subprogram has X, Y, and Z three-axis movement commands, if the X, Y, and Z coordinates before look-ahead M code are modified after the look-ahead is completed, the look-ahead NC program will be executed. Issue this alarm.) 2. The contents of the look-ahead NC program cannot be read correctly.		
Solution	After advanced look-ahead under correct conditions, execute the look-ahead NC programs.		

7.194 COR-311 Hard disk storage space is insufficient to place look-ahead NC programs

Alarm ID	COR-311 BGND-311	Alarm title	【Hard disk storage space is insufficient to place look-ahead NC programs】
Description	The hard disk storage space is not enough to place the look-ahead NC program.		
Possible Cause	There is not enough storage space on the hard drive.		
Solution	Release hard disk storage space.		

7.195 COR-320 Friction Compensation Adjustment setting is illegal

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Description	Friction compensation adjustment setting is illegal.		

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Possible Cause	1. Selected adjustment mode is not supported. 2. No axis is assigned for adjustment. 3. Radius or feedrate is not configured in Comp ON/Comp OFF mode . 4. No. of circular setting is not configured in Comp Learning mode.		
Solution	1. Select other compensation mode and try again. 2. Please set a set of available axes. 3. Please set correct radius or feedrate. 4. Please set correct No. of circular setting. If the alarm still exists, please contact Syntec OEM.		

7.196 COR-321 The arguments (P,Q,R) retrieved from the I point position must be integers

Alarm ID	COR-321 BGND-321	Alarm title	【The arguments (P,Q,R) retrieved from the I point position must be integers】
Description	The I value position taken argument (G10 L1010 P_ Q_ R_) is not an integer.		
Possible Cause	NC programming error.		
Solution	Modify the NC program to specify correct P, Q, and R arguments.		

7.197 COR-322 I point position capture failed

Alarm ID	COR-322 BGND-322	Alarm title	【I point position capture failed】
Description	I value position capture function (G10 L1010 P_ Q_ R_), startup failed.		
Possible Cause	1. The specified P argument (axis number) does not exist. 2. The specified Q argument (I value) is not supported. 3. Exceeding the limit of the number of used groups, currently support to start three I value location captures at the same time.		

Alarm ID	COR-322 BGND-322	Alarm title	【I point position capture failed】
Solution	1. Modify the program to specify correct P, Q, and R arguments. 2. Turn off settings that are not in use.		

7.198 COR-323 The arguments (P,R,I,J,K) captured from the driver signal position must be integers

Alarm ID	COR-323 BGND-323	Alarm title	【The arguments (P,R,I,J,K) captured from the driver signal position must be integers】
Description	The argument of the drive signal position (G10 L1011 P_ R_ I_ J_ K_) is not an integer.		
Possible Cause	The NC programming error.		
Solution	Modify the program to specify correct P, R, I, J, and K arguments.		

7.199 COR-324 Driver signal position captured failed to activate

Alarm ID	COR-324 BGND-324	Alarm title	【Driver signal position captured failed to activate】
Description	The drive signal position capture function (G10 L1011 P_ R_ I_ J_ K_) failed to activate.		
Possible Cause	1. The range of R values set by the specified P and R arguments is invalid (occupied by the system or out of range). 2. The specified I argument (axis number) does not exist. 3. The specified J argument (which set of latches to use) is out of range. 4. The specified K argument (select signal source) is out of range.		
Solution	Modify the NC program to specify correct arguments.		

7.200 COR-325 In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited

Alarm ID	**COR-325** **BGND-325**	**Alarm title**	**【In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited】**
Description	In the polar coordinate interpolation (G12.1) mode, the diameter & radius axis programming switching command (G10.9) cannot be used.		
Possible Cause	In polar coordinate interpolation (G12.1) mode, if switch the diameter & radius axis program with the diameter & radius axis programming switching command (G10.9), this alarm is issued.		
Solution	In the polar coordinate interpolation (G12.1) mode, do not use the diameter & radius axis to program the switching command (G10.9). Please follow Pr281~Axis radius axis or diameter axis set value to move the command. [Remarks] For milling machine G12.1, the X axis is programmed in radius axis. For lathe machine G12.1, and the X axis programming mode can be set using Pr4020 (G12.1 X axis programming).		

7.201 COR-326 Diameter & radius axis programming command argument error

Alarm ID	**COR-326** **BGND-326**	**Alarm title**	**【Diameter & radius axis programming command argument error】**
Description	Diameter and Radius axis programming (G10.9) switching, command arguments are written incorrectly.		
Possible Cause	No axial programming is specified, or programming is specified as a value other than 0 and 1.		
Solution	1. Specify any axial direction after diameter & radius axis programming (G10.9). 2. Specify the argument value (programming mode) as 0 or 1.		

7.202 COR-327 Skip function argument input error

Alarm ID	COR-327 BGND-327	Alarm title	【Skip function argument input error】
Description	Skip function (G31) input argument is incorrect.		
Possible Cause	Skip function (G31) specifies both P and R arguments.		
Solution	Please modify the Skip function (G31), no specified both P and R.		

7.203 COR-328 G10 L1800 command argument number, out of specification limit

Alarm ID	COR-328 BGND-328	Alarm title	【G10 L1800 command argument number, out of specification limit】
Description	The command argument number is outside the specification limit. i.e. G10 L1800 I514 P10 R1, I argument is out of range.		
Possible Cause	The NC programming error.		
Solution	Please confirm the argument value of G10 L1800.		

P.S. Valid version of COR-328 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

7.204 COR-329 G10 L1800 command number has exceeded the single block limit

Alarm ID	COR-329 BGND-329	Alarm title	【G10 L1800 command number has exceeded the single block limit】
Description	The number of G10 L1800 commands with different arguments exceeds the upper limit allowed by a single block, up to 5, or too many G10 L1800 commands.		
Possible Cause	The NC programming error.		
Solution	Please reduce the number of G10 L1800 commands with different arguments to less than 5 in a block, or reduce the number of G10 L1800s as a whole.		

P.S. Valid version of COR-329 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

7.205 COR-330 Illegal interrupt signal format

Alarm ID	COR-330 BGND-330	Alarm title	【Illegal interrupt signal format】
Description	The command argument number is outside the specification limit. i.e. M96 P5566 I4 Q100 R1 L1000; I argument specifies the error signal source.		
Possible Cause	The NC programming error..		
Solution	Please check the NC program to ensure that the command arguments are correct.		

7.206 COR-331 This product limits T Code function

Alarm ID	COR-331 BGND-331	Alarm title	【This product limits T Code function】
Description	This product only supports T0~T4.		
Possible Cause	This product only supports T0~T4.		
Solution	Use a products that support the full T-code function.		

7.207 COR-332 Interrupt type subprogram (M96, M97) execution failure

Alarm ID	COR-332 BGND-332	Alarm title	【Interrupt type subprogram (M96, M97) execution failure】
Description	When the interrupt type subprogram function is used, the execution fails when the interrupt signal is triggered. Note: Due to the pre-solved relationship when issuing this alarm, there is no guarantee that it will stop at the problem line.		
Possible Cause	Only support to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). Triggering interrupt signals in other subprograms is not supported.		

Alarm ID	COR-332 BGND-332	Alarm title	【Interrupt type subprogram (M96, M97) execution failure】
Solution	1. Make sure to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). 2. Set #1510 to 4 (the second bit is set to on) and only display the line number of the main program. This can be regarded as triggering the interrupt in the main program, but this setting will re-execute the subprogram when it returns.		

7.208 COR-333 Single end point exceeds hardware stroke limit

Alarm ID	COR-333 BGND-333	Alarm title	【Single end point exceeds hardware stroke limit】
Description	The coordinate position in the program exceeds the hardware stroke limit set by the machine.		
Possible Cause	1. The NC program is wrong. 2. The hardware stroke limit signal is abnormal.		
Solution	1. Check the NC program and correct the coordinate position. 2. First confirm whether it is occasionally happen during machining, and MOT-25, 26 is no longer exists after the system enters the feedhold. If so, it may be that an axis hardware stroke limit signal is triggered, but then disappears; please check the limit switch wiring or the hardware itself for abnormality.		

7.209 COR-334 G10 L1810 illegal signal condition command format

Alarm ID	COR-334 BGND-334	Alarm title	【G10 L1810 illegal signal condition command format】
Description	G10 L1810 command format error.		
Possible Cause	1. One of the I, Q, and R arguments is not set. 2. The I, Q, R, and J arguments are out of range, set to a negative, or a decimal value.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-334 : before 10.118.34 (included).

7.210 COR-335 G10 L1810 signal waiting condition is too many or repeat the same signal

Alarm ID	**COR-335** **BGND-335**	**Alarm title**	**【G10 L1810 signal waiting condition is too many or repeat the same signal】**
Description	The G10 L1810 signal waiting condition exceeds the allowable number, or the G10 L1810 repeatedly sets the same signal.		
Possible Cause	1. G10 L1810 continuously sets more than 5 commands. 2. The same G10 L1810 signal command has been set.		
Solution	Please combine multiple signals into one signal to wait, or reduce the signal waiting condition.		

7.211 COR-336 G10 L1820 illegal waiting signal condition command format

Alarm ID	**COR-336** **BGND-336**	**Alarm title**	**【G10 L1820 illegal waiting signal condition command format】**
Description	G10 L1820 command format error.		
Possible Cause	P, K arguments are out of range, set negative or decimal.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-336 : before 10.118.34 (included).

7.212 COR-337 G10 L1820 without G10 L1810 command in the front

Alarm ID	**COR-337** **BGND-337**	**Alarm title**	**【G10 L1820 without G10 L1810 command in the front】**
Description	The G10 L1820 command needs to wait for the G10 L1810 signal wait condition. At least one G10 L1810 command must be programmed first.		
Possible Cause	The G10 L1820 command was not written before the G10 L1820 command.		

Alarm ID	COR-337 BGND-337	Alarm title	【G10 L1820 without G10 L1810 command in the front】
Solution	Modify the NC program and write the G10 L1810 command before the G10 L1820 command.		

7.213 COR-338 G10 L1820 signal condition waiting timeout

Alarm ID	COR-338 BGND-338	Alarm title	【G10 L1820 signal condition waiting timeout】
Description	G10 L1820 waiting signal condition exceeds the waiting time.		
Possible Cause	The signal status condition was not reached within the waiting time.		
Solution	1. Check if the G10 L1810 setting signal condition is correct. 2. Check if the PLC wait signal is processed correctly. 3. Check the hardware device to confirm why the signal status is not reached.		

7.214 COR-339 Chopping axis prohibition movement command

Alarm ID	COR-339 BGND-339	Alarm title	【Chopping axis prohibition movement command】
Description	The chopping axial direction does not accept any movement commands.		
Possible Cause	After using the chopping function (G81.1, G81.2, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.		
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.		

7.215 COR-340 Chopping axis prohibits changing coordinate system

Alarm ID	COR-340 BGND-340	Alarm title	【Chopping axis prohibits changing coordinate system】
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.		
Possible Cause	1. After using the chopping function (G81.1, G81.2, C86), switch the coordinate system before closing and affect the chopping axis. 2. Simultaneously use of chopping function (G81.1, G81.2, C86) and tilted work plane machining function (G68.2, G68.3). 3. Simultaneously use the chopping function (G81.1, G81.2, C86) and the axis exchange function (C133~C136). Note: C86 support version: 10.118.19 and earlier.		
Solution	1. Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. 2. Check if the NC program uses the chopping function (G81.1, G81.2,C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). 3. Check if the NC program has the chopping function (G81.1, G81.2, C86) and the shaft exchange of the chopping shaft. (C133~C136). Note 1: C86 support version: 10.118.19 and earlier. Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658). Note 3: Coordinate system related operations: external coordinate offset, MPG offset.		

7.216 COR-341 Chopping axial switching error

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Description	The specified axis cannot be switched to the chopping axis.		
Possible Cause	1. This axis has been designated as the PLC axis. 2. The axis has been designated as the spindle. 3. This axis has been designated as a chopping axis by other paths.		

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Solution	1. Do not specify the PLC axis as the chopping axis. 2. Do not specify the spindle as the chopping axis. 3. Check if the multi-path repeats the chopping function for the same axis (G81.1, G81.2, C86). Note: C86 support version: 10.118.19 and earlier version.		

7.217 COR-342 Chopping axis prohibits non-linear kinematic transform

Alarm ID	COR-342 BGND-342	Alarm title	【Chopping axis prohibits non-linear kinematic transform】
Description	Non-linear kinematic transform is prohibited for the chopping axis.		
Possible Cause	1. Using non-linear kinematic transform when chopping function is enabled. 2. The machine type used is two-dimensional kinemetic (special model).		
Solution	1. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the tangential control (G41.1, G42.1). 3. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 4. The machine configuration used is the two-dimensional kinemetic (special model), and the chopping function (G81.1, G81.2, C86) cannot be used. Note: C86 support version: 10.118.19 and earlier.		

7.218 COR-343 The main program does not use the ending M code

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Description	M02/M30/M99 is not used at the end of the main program.		
Possible Cause	1. Programming error. 2. NC program is damaged. 3. Some machining information was lost due to file transmission errors.		

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Solution	1. Check the NC program to reset the alarm and clear it. 2. Close the Pr3853-NC program error check function.		

7.219 COR-344 The NC program under invalid paragraph

Alarm ID	COR-344 BGND-344	Alarm title	【The NC program under invalid paragraph】
Description	1. There is continuous blank/annotation/garbled in the NC program, and the size is more than 8KB. 2. There are no valid commands in the NC program.		
Possible Cause	1. Programmning error. 2. NC program is damaged.		
Solution	1. Avoid writing lots of consecutive blanks or annotations. 2. Please check if the NC program is damaged. After the problem is solved, reset to remove the alarm. 3. Close the Pr3853 - NC program error check function.		

7.220 COR-345 Illegally format input to programmable data

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Description	Programmable data input argument format error.		

SYNTEC

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Possible Cause	1. Wobble command a. One of the I, Q, R arguments is not set. b. I, Q, R arguments out of range. c. I argument not integer. 2. G10 L1803 command a. One of the I, Q, P, R arguments is not set. b. I, Q, P, R, J, E arguments out of range. c. I, Q, R, J, K, E arguments not integer. 3. G10 L1805 、G10 L1810 command a. One of the I, Q, R arguments is not set. b. I, Q, R, J arguments out of range. c. I, Q, R, J arguments not integer. 4. G10 L1820 command a. P, K arguments out of range. b. P, K arguments not positive integer. 5. G10 L4500 command a. One of the P, Q, R arguments is not set. b. P argument is not a string. c. Q, R arguments out of range. d. Q, R arguments not integer. 6. G10 L1030 command a. P、Q、R、X、Y、Z、I、J、K、U、V arguments out of range. b. Q、X、Y、Z、I、J、K、U、V arguments not integer. 7. G10 L1031 command a. R argument is not set. b. R, V arguments out of range. c. R, V arguments not integer. 8. G10 L1500 command a. One of the P、Q arguments is not set. b. One of the P、Q arguments is out of range. c. One of the P、Q arguments is not an integer. 9. G10 L1350 command a. P, E argument is not set. b. P, E argument is out of range. c. P argument not integer.		
Solution	Please refer to command description to troubleshoot.		

7.221 COR-346 Illegally use of programmable data entry

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Description	Programmable data input conflicts with specifications.		
Possible Cause	1. Between wobble command, only one block is allowed to wobble. 2. G10 L1803 commands with different arguments exceeds the upper limit allowed by a single block, up to 12. 3. Too many G10 L1803 and G10 L1805 commands. 4. The feature as L-argument specified is unsupported for current version (from 10.118.41P, 10.118.49). 5. Any axis belongs to multiple coordinates.		
Solution	1. Remove the exceeding number of wobble moving blocks. 2. Reduce the number of G10 L1803 commands in a single movement block to less than 12. 3. Reduce G10 L1803 and G10 L1805 commands. 4. Update to the version which supports the feature as L-argument specified. 5. Please set Pr701~ of the axis to only belong to a single axis group.		

7.222 COR-347 This function only supports Syntec axis

Alarm ID	COR-347 BGND-347	Alarm title	【This function only supports Syntec axis】
Description	Only Syntec axis can use this function.		
Possible Cause	The weave function only supports Syntec drives.		
Solution	Please use Syntec driver.		

7.223 COR-348 APP command error

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error】
Description	There is an error in the APP command.		

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error】
Possible Cause	1. APP name is not a string. 2. The length of APP name is over 20 characters. 3. No APP name is specified.		
Solution	Please make sure that APP commands are used correctly in the NC program.		

7.224 COR-349 This function only supports Syntec encoder

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Description	Limited to use with Syntec magnetic gear or ring gear encoders.		
Possible Cause	When complex threading cycle(G78 or G78.2) enables chip removal function, the active spindle must use Syntec magnetic gear or ring gear encoders. In dual-feedback system, it is allowed to use chip removal function when either of the encoders is Syntec magnetic gear or ring gear encoder.		
Solution	1. Set the argument of D to 0. 2. Please use Syntec magnetic gear or ring gear encoders.		

7.225 COR-350 Invalid EnIP Macro

Alarm ID	COR-350 BGND-350	Alarm title	【Invalid EnIP Macro】
Description	EnIP macro syntax error.		
Possible Cause	The format or range of the input macro arguments is incorrect.		
Solution	Refer to the EnIP macro syntax description.		

7.226 COR-351 Communication command abnormal

Alarm ID	COR-351 BGND-351	Alarm title	【Communication command abnormal】
Description	1. The execution of the macro communication command failed. 2. The macro communication to driver failed.		
Possible Cause	1. The controller does not support EnIP communication. 2. The EnIP function input request data value range is incorrect. 3. An error occurred while sending or receiving a communication command.		
Solution	1. Please contact the Syntec OEM. 2. Each request data of EnIP represents a 1 value ranging from 0 to 255. 3. This alarm is accompanied by a more detailed error alarm, refer to the alarm description for troubleshooting.		

7.227 COR-352 Communication command response data is too long

Alarm ID	COR-352 BGND-352	Alarm title	【Communication command response data is too long】
Description	The device return data length is too long.		
Possible Cause	The storage space provided by the user for returning data is insufficient.		
Solution	Make sure that the storage space for the returned data is sufficient.		

7.228 COR-353 Invalid argument of CHKINF

Alarm ID	COR–353 BGND-353	Alarm title	【Invalid argument of CHKINF】
Description	The type of argument of CHKINF() is incorrect, or the category number is out of range.		
Reason	Programming error.		

Solution	1. Please check the NC program. 2. Check the type of first argument is integer. 3. Check the type of second argument is string. 4. Ensure the category number is in the range, please refer to OpenCNC_Macro Development Manual.

7.229 COR-354 With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.

Alarm ID	**COR-354** **BGND-354**	**Alarm title**	**【With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.】**
Description	With Rotate Tool Center Point function(G43.4, G43.5) enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.		
Reason	The NC file was written incorrectly.		
Solution	Please modify the NC file, cancel Rotate Tool Center Point function(G49), and then use G10 L1150 to set the Five-Axis mechanism parameter.		

7.230 COR-355 Need geometric axes positioning after tool length compensation changed

Alarm ID	**COR–355** **BGND-355**	**Alarm title**	**Need geometric axes positioning after tool length compensation changed**
Description	Positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) is required after executing tool length compensation commands, if tool radius compensation commands needed.		
Reason	No positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) between executing tool length compensation commands and tool radius compensation commands. Tool length compensation commands: H, T(Lathe), G43, G44, G43.4, G43.5, G49, G10 L1050, G10 L1051 Tool radius compensation commands: G41, G42		
Solution	Please modify the NC file. Add positioning command of the axes on work plane(G17: XY, G18: ZX, G19: YZ) after tool length compensation commands and before tool radius compensation commands. Positioning commands: G90 G00, G90 G01, G92		

7.231 COR-356 Invalid APP command

Alarm ID	**COR–356** **BGND-356**	**Alarm title**	**【Invalid APP command】**
Description	There is a syntax error on the Modbus command(G10 L1900/1901/1910/1911).		
Possible Cause	The format or range of the input argument is incorrect.		
Solution	Please refer to the Industry Machine Application Manual.		

7.232 COR-357 Illegal Modbus packet contents

Alarm ID	**COR–357** **BGND-357**	**Alarm title**	**【Illegal Modbus packet contents 】**
Description	The contents of the Modbus command packet is illegal.		
Possible Cause	The contents of the customized package is incorrect.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

7.233 COR-358 Modbus communication timeout

Alarm ID	**COR–358** **BGND-358**	**Alarm title**	**【Modbus communication timeout】**
Description	The Modbus communication experienced a timeout.		
Possible Cause	Abnormal communication or unestablished communication causes the connection timeout.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

7.234 COR-359 Modbus communication failure

Alarm ID	COR–359 BGND-359	Alarm title	【Modbus communication failure】
Description	The Modbus communication failed.		
Possible Cause	1. There is a communication error or device setting error.		
Solution	1. Please refer to R5039, as this register provides the error code for troubleshooting		

7.235 COR-361 The number of non-moving blocks exceeds the permissible value under tool radius compensation

Alarm ID	COR-361 BGND-361	Alarm title	【The number of non-moving blocks exceeds the permissible value under tool radius compensation】
Description	The machining program has programmed too many non-moving blocks when tool radius compensation is enable.		
Possible Cause	Programming error.		
Solution	1. Modify the machining program and issue the non-moving blocks when tool radius compensation is disable. 2. Modify the machining program and reduce the number of non-moving blocks when tool radius compensation is enable.		

7.236 COR-362 Multi-axis multi-signal skip function (G31.10, G31.11) command error

Alarm ID	COR-362 BGND-362	Alarm 标题	【Multi-axis multi-signal skip function (G31.10, G31.11) command error】
说明	Multi-axis multi-signal skip function (G31.10, G31.11) command error		

可能原因	1. Setting command (G31.10) or execution command (G31.11) cannot be issued alone, and other commands cannot be issued between them. Always set first then execute. 2. A set of multi-axis multi-signal skip function is composed of one G31.10 command at least and one G31.11 command. Set first then execute. Both commands should be issued in a set of multi-axis multi-signal skip function, and it is not allowed to insert other commands in between. 3. Supports up to six G31.10 commands in a set of multi-axis multi-signal skip function. 4. Same axis is used in different G31.10 commands in a set of multi-axis multi-signal skip function. E.g., a. Same axis name is used in different G31.10 commands. b. Assign same axis by using virtual axis function in different G31.10 commands. (E.g., Virtual axis Z is corresponding to axis Z1 and Z2. Using axis Z in one G31.10 command, and Z1 in another G31.10 command.) c. Assign same axis by misusing alias of axis. (E.g., There are axis X, Y, Z1, Z2, Z3, Z4, the alias of Z1 is Z.)
排除方法	1. Make sure there is always an execution command (G31.11) behind setting command (G31.10), and there is no execution command (G31.11) left alone. 2. Make sure there are six G31.10 commands at most in each set of multi-axis multi-signal skip function. 3. Make sure there is no axis assigned repeatedly in each set of multi-axis multi-signal skip function.

7.237 COR-363 Invalid axis removal/axis borrowing function

Alarm ID	**COR-363**	**Alarm title**	**【Invalid axis removal/axis borrowing function】**
Description	Axis removal (G52.1) or axis borrowing (G52.2) function is used incorrectly.		
Reason	1. Without any argument of P, Q, R in the G52.1 and G52.2 commands. 2. The axes corresponding to the P, Q, R arguments does not exist in the system. 3. I, J, K arguments are integers but out of the settable range. 4. Issue the G52.1 or G52.2 command for non-roaming axis. 5. When using the axis removal command, the axis to be removed hasn't been borrowed by the axis group. 6. When using the axis borrowing command, the axis to be borrowed has been borrowed by the axis group. 7. It is not allowed to set the related axes of nonlinear mechanism transform as roaming axes, including: • RTCP (G43.4, G43.5); • Tilted plane function (G68.2, G68.3 and G53.1, G53.3, G53.6); • Tangential control function (G41.1, G42.1); • Polar coordinate conversion (G12.1); • Two-Dimensional Kinemetic (special model). 8. Issue the G52.1 or G52.2 command for Indexing axis.		

Solution	1. In the G52.1 or G52.2 command, at least one argument among P, Q, R is required. 2. Please refer to Pr321~ setting value, modify the NC program, and specify the correct axes. 3. Please refer to the command format, modify the NC program, and set the parameters within the settable range. 4. Please set Pr701~ to belong to multiple axis groups, and set Pr742 to 1(i.e., set the axis to be borrowed as the roaming axis.); Or please modify the processing program and delete the G52.1 or G52.2 command. 5. Check the NC program, fix the P, Q, R argument assignment error. 6. Check the NC program, fix the P, Q, R argument assignment error. 7. Please set Pr742 to 0 to avoid the disallowed usages in No.7 of the possible reason. 8. Check and modify the NC program, don't remove or borrow indexing axis.

7.238 COR-364 Axis borrowing function failed to borrow

Alarm ID	**COR-364**	**Alarm title**	**【Axis borrowing function failed to borrow】**
Description	Use axis borrowing (G52.2) function, unable to borrow all specified axes successfully.		
Reason	Because the axis to be borrowed is being borrowed by other axis groups, it cannot be borrowed successfully.		
Solution	1. Check the NC program and confirm that the axis to be borrowed has been removed by other axis groups before issuing the borrow instruction. 2. Enlarge the K argument setting value, so that extends the waiting time. 3. Set the I parameter to 0, and continue to wait for other axis groups to remove the axis.		

7.239 COR-365 Issue a movement command to the unborrowed roaming axis

Alarm ID	**COR-365**	**Alarm title**	**【Issue a movement command to the unborrowed roaming axis】**
Description	Issue a movement command to the unborrowed roaming axis.		
Reason	1. When Pr742 is set to 1, the axis group or PLC Rn sub-program component issue a movement command to the unborrowed roaming axis. 2. The tool break retract is performed on the path with the roaming axis command, and the roaming axis that has not been borrowed successfully, so an alarm is issued.		

Solution	1. For the axis group or PLC Rn sub-program component, before issuing a movement command to the roaming axis, use the G52.2 command to successfully borrow the axis. 2. Modify the tool break retract setting line number or serial number before G52.2 command.

7.240 COR-366 The parameters of rotary axis auxiliary brake are set incorrectly

Alarm ID	**COR-366** **BGND-366**	**Alarm Title**	**【The parameters of rotary axis auxiliary brake are set incorrectly】**
Description	Using the rotary axis auxiliary brake, the parameters mapped to the enable group are set incorrectly.		
Possible Cause	Parameters are incorrect. Enable G10 L1500 rotary axis auxiliary brake: 1. The axis ID of the rotary axis auxiliary brake is not specified. 2. The axis type of the specified axis is linear. 3. The M code of the rotary axis auxiliary brake is not specified.		
Solution	Modify parameters Pr3741~Pr3744.		

7.241 COR-367 Prohibited commands in rotary axis auxiliary brake mode

Alarm ID	**COR-367** **BGND-367**	**Alarm Title**	**【Prohibited commands in rotary axis auxiliary brake mode】**
Description	When the rotary axis auxiliary brake is enabled, prohibited commands are used in the nc program.		
Possible Cause	1. The axis, which is in rotary axis auxiliary brake mode, is specified by skip function(G28.1, G31, G31.11). 2. The axis, which is in rotary axis auxiliary brake mode, is specified by chopping function(G81.1, G81.2). 3. Enable the rotary axis auxiliary brake in RTCP mode(G43.4, G43.5). 4. Enable both the rotary axis auxiliary brake mode and the polar coordinate mode(G12.1).		
Solution	Check the NC program.		

7.242 COR-368 The position command of the indexing axis is inaccurate

Alarm ID	COR-368 BGND-368	Alarm title	【The position command of the indexing axis is inaccurate】
Description	The index position command isn't an integral multiple of the minimum precision angle.		
Possible Cause	The index position or movement command at current program isn't an integral multiple of the minimum precision angle.		
Solution	Please check the program, correct the position or movement command of the index table to an integer multiple of the minimum precision angle.		

7.243 COR-369 Illegal indexing axis command

Alarm ID	COR-369 BGND-369	Alarm title	【Illegal indexing axis command】
Description	The command of the indexing axis in the program is illegal.		
Possible Cause	1. There are both indexing axis and normal axes in the same G01、G02、G03、G02.4、G03.4、G33、G34 command. 2. G12.1 polar coordinate interpolation specifies the indexing axis as the rotation axis. 3. G28.1, G31, G31.10 skip function specifies the indexing axis. 4. G80.1, G80.2, G81.1, G81.2 command the indexing axis to chopping.		
Solution	1. Please check and modify the program, and split the indexing axis and normal axis into different G01、G02、G03、G02.4、G03.4、G33、G34 command. 2. Don't specify an indexing axis as a rotation axis for polar coordinate interpolation. 3. Don't use the indexing axis for the skip function. 4. Don't use indexing axis for chopping.		

7.244 COR-370 Failed to enable chopping function

Alarm ID	COR-370 BGND-370	Alarm title	【Failed to enable chopping function】
Description	Failed to enable the chopping function .		

Alarm ID	COR-370 BGND-370	Alarm title	【Failed to enable chopping function】
Possible Cause	1. G81.2 or G81.1 command repeatedly a. G81.1 command repeatedly. b. G81.2 and G81.1 command simultaneously. 2. Using G81.2 to make 2 axis in the same axis group do advanced chopping simultaneously.		
Solution	Please activate or deactivate the chopping function correctly.		

7.245 COR-401 Path planning plug-in error

Alarm ID	COR–401 BGND-401	Alarm title	Path planning plug-in error
Description	Path planning plug-in error.		
Reason	Path planning plug-in error leads to system abnormality.		
Solution	Please contact OEM Syntec.		

SYNTEC

8 Background Execute Alarm - BGND

Alarm ID	**COR–001** **BGND-001**	**Alarm title**	**Array Variable is empty**
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

Alarm ID	**COR–002** **BGND-002**	**Alarm title**	**File not exist**
Description	If the file that the system wants to read does not exist i.e. use M98 (or G65, G66, and etc.) to call a non existing file.		
Reason	Programming error.		
Solution	Check the NC program to make sure the existence of the file.		

Alarm ID	**COR–003** **BGND-003**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of MACRO is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		
Reason	Programming error		
Solution	Check the NC program to ensure that the denominator is not equal to 0.		

Alarm ID	**COR–004** **BGND-004**	**Alarm title**	**Operation domain error**
Description	Operation domain error.		
Reason	Programming error		

Solution	Please check the NC program.		
Alarm ID	**COR–005** **BGND-005**	**Alarm title**	**Program loading failure**
Description	MACRO syntax error.		
Reason	Programming error.		
Solution	Please check the NC program.		
Alarm ID	**COR–006** **BGND-006**	**Alarm title**	**Arc not on work plane**
Description	1. In clockwise, counter-clockwise arc cutting (G02, G03) syntax, if the vector from center of circle to starting point doesn't exist on the arc working surface. i.e. execute G17 G02 I50. K10. , and system will issue this alarm. 2. In spiral interpolation (G02, G03) syntax, when corresponding to G17, G18, and G19 three surface conditions, if K, J , and I arguments are not zero, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that G02 and G03 are used correctly.		
Alarm ID	**COR–007** **BGND-007**	**Alarm title**	**Arc radius too short**
Description	In G02 and G03 syntax, if Arc radius is smaller than 10 to the power of minus 10 BLU (10^-10), system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.		
Alarm ID	**COR–008** **BGND-008**	**Alarm title**	**Arc destination not on arc**

Description	In G02 and G03 syntax, if the Arc end point coordinate is not on the circle, system will issue this alarm. From 8.31 version, adding Pr3807 setting arc final point is not on the check window, allowing to set window range's error in Pr3807. When arc end point position's error is smaller than Pr3807 setting window's range, system will auto correct the circle's center position, and make the end point position to locate on the circle correctly. When arc end point position error is bigger than Pr3807 window range setting, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.		
Alarm ID	**COR–009** **BGND-009**	**Alarm title**	**G65 layers called by Macro too deep**
Description	Use G65 to call MACRO program layer more than 12 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 12 layers.		
Alarm ID	**COR–010** **BGND-010**	**Alarm title**	**G66 layers called by Macro too deep**
Description	Use G66 to call MACRO program layer more than 4 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 4 layers.		
Alarm ID	**COR–011** **BGND-011**	**Alarm title**	**Subprogram call too deep**
Description	Use M98 to call subprogram that has more than 16 layers.		
Reason	Programming error.		

Solution	Check NC program to ensure that M98 calls subprogram that has less than 16 layers.		
Alarm ID	**COR–012** **BGND-012**	**Alarm title**	**G66 mode not cancel by G67**
Description	G66 and G67 need to be used in pairs, when numbers of G67 is more than G66 in one NC program, this alarm will be issued.		
Reason	Programming error.		
Solution	Check NC program to ensure that G66 and G67 are used in pairs		
Alarm ID	**COR–013** **BGND-013**	**Alarm title**	**G65, G66 must be at the end of the block**
Description	G65 and G66 are MACRO, so in one same block the program on the right hand side of G65 and G66 will be processed as G65 and G66's arguments. So in one same block, if there is other G code command please write them in the left hand side of G65 and G66. In one same block, if the right hand side of G65 and G66 has G code or M code, system will issue this alarm.		
Reason	Programming error.		
Solution	Please check the NC program.		
Alarm ID	**COR–014** **BGND-014**	**Alarm title**	**Absent program number**
Description	The right hand side of G65 and G66 doesn’t have P argument to specify program number.		
Reason	Programming error.		
Solution	Please check the NC program to ensure G65 and G66 use P argument to specify program number.		
Alarm ID	**COR–015** **BGND-015**	**Alarm title**	**Too many M code**
Description	There are more than 3 M codes in a single block.		

Reason	Programming error.		
Solution	Please check the NC program to ensure that there are equal or less than 3 M codes in a single block		
Alarm ID	**COR–016** **BGND-016**	**Alarm title**	**Illegal variable access**
Description	Attempted to access a variable that does not exist.		
Reason	1. Programming error. 2. Illegal access to # or @ variable. 3. The parameter of SYSDATA function is out of system data range. 4. The controller does not support the status variable which DRVDATA is to read. 5. The program that attempts to access AR or MAR variables does NOT lie within the scope of App Macros. 6. The AR or MAR variables to access do NOT lie within the range defined by the App. 7. Non-integer numbers are used to specify the addresses of AR or MAR variables. ONLY integers are acceptable.		
Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable. 4. Make sure the variable which DRVDATA is to read can be found on the "Controller Axis Info." page. 5. Make sure the AR and the MAR variables are accessed ONLY in App Macros. 6. Make sure the AR and the MAR variables to access lie within the range defined by the App. 7. Make sure only integers are used to specify the addresses of AR or MAR variables.		

Alarm ID	**COR–017** **BGND-017**	**Alarm title**	**Label not found**
Description	Cannot find out corresponding serial number N in GOTO command.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	**COR–018** **BGND-018**	**Alarm title**	**Line number not found**
Description	Input line number is incorrect.		

Reason	1. When Pr3851 sets as 888800, input breakpoint's line number exceeds NC program's maximum line number. 2. When Pr3851 sets as 999900, input breakpoint's line number exceeds NC program's maximum line number. 3. When Pr3851 sets as 999901, input breakpoint's line number exceeds NC program's maximum line number or positioning command line number cannot be found after scanning through entire NC program. 4. When M99 Q_ sub-program returns to the line number which is assigned by main program, the line number Q_ assigned is over main program's maximum line number.		
Solution	1. Input the correct program line number. 2. When Pr3851 sets as 999901, modify NC program by inserting positioning command after initial assigned breakpoint.		
Alarm ID	**COR–019** **BGND-019**	**Alarm title**	**sub-program has no M99**
Description	When main program calls sub-procedure, if sub-program finish executing and need to return the main program, there is no M99.		
Reason	Programming error.		
Solution	Write in M99 when sub-program finish executing and return the main program.		
Alarm ID	**COR–020** **BGND-020**	**Alarm title**	**Too many G code**
Description	There are more than 10 G codes in a single block.		
Reason	Programming error.		
Solution	Breakdown that single block which has over 10 G codes into single blocks that has less than 10 G codes.		
Alarm ID	**COR–021** **BGND-021**	**Alarm title**	**Too many (I,J,K) triples**
Description	Repeat too much I, J, and K command in the same block.		
Reason	Programming error.		

Solution	Please check the NC program.		
Alarm ID	**COR–022** **BGND-022**	**Alarm title**	**Use undefined workpiece coordinate**
Description	Use an undefined G54 coordinate system.		
Reason	Programming error.		
Solution	Use the correct G54 coordinate system.		
Alarm ID	**COR-023** **BGND-023**	**Alarm Title**	**Semantic error**
Description	1. When using G code, different G code can input different argument (P_, L_, R_, and etc.), if argument setting is wrong, and system will issue this alarm. 2. The argument of macro function is wrong.		
Reason	Programming error.		
Solution	Check the NC program to ensure the argument is used correctly by referring the manual.		
Alarm ID	**COR–024** **BGND-024**	**Alarm title**	**Invalid arc radius value**
Description	When executing G02, G03, appointed Arc end point and given radius is contradicted, which the given radius cannot meet appointing Arc end point. i.e. G91 G03 X1500 Y4000 R2000.		
Reason	Programming error.		
Solution	Check the program and recalculate.		
Alarm ID	**COR–025** **BGND-025**	**Alarm title**	**Macro stack is overflow or STKTOP[ ] argument error**

Description	1. STACK can store maximum 4095 values. If stored value exceeding the maximum number, controller issue this alarm. 2. In STKTOP[n], n is started from 0, if the value of n is bigger than the value storing in stack-1, controller issue this alarm
Reason	1. Too much value store in STACK. 2. STKTOP[] arguments exceeds the value storing in STACK.
Solution	1.Stack is full, do not use store command anymore. 2. Input a reasonable argument in STKTOP[].

Alarm ID	**COR–026** **BGND-026**	**Alarm title**	**macro stack is empty**
Description	Empty stack still want to pop value, system issue this alarm.		
Reason	The numbers of Push commands and Pop commands are not the same.		
Solution	Check the program to ensure that the number of Push commands is the same with that of Pop commands.		

Alarm ID	**COR–027** **BGND-027**	**Alarm title**	**Invalid macro arguments**
Description	Macro issue self-defined alarm.		
Possible Cause	Once Macro determines the self-defined alarm error condition is meet, NC program will be stopped and alarm will appear.		
Solution	According to the display content of alarm to find out the error.		

Alarm ID	**COR–028** **BGND-028**	**Alarm title**	**System program error, can’t normally machining.**
Description	Use Quiet Mode in MACRO, but can't leave Quiet Mode when program finished.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–029 BGND-029	Alarm title	Tool length offset change at arc
Description	G43, G44, G49 only receive linear interpolation command in the next block.		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR-030 BGND-030	Alarm Title	Cutting speed command is 0
Description	When execute cutting command, given F code argument is zero.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the argument of F code shall not be 0.		

Alarm ID	COR–031 BGND-031	Alarm title	Radius compensation cancel at arc
Description	G40 only receives linear interpolation command in the next block		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–032 BGND-032	Alarm title	Radius compensation activate at arc
Description	G41,G42 only receive linear interpolation command in the next block.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–033 BGND-033	Alarm title	Improper use of A, R, or C command

Description	Timing of using A, R, or C command is incompatible with the specification.
Reason	Programming error.
Solution	Check NC program to confirm that if the blocks are compatible with specifications

Alarm ID	**COR–034** **BGND-034**	**Alarm title**	**Command syntax has conflict**
Description	No application currently.		
Reason			
Solution			

Alarm ID	**COR–035** **BGND-035**	**Alarm title**	**Corner is too small can’t insert round corner chamfer**
Description	Angel between blocks, which to be inserted round corner or chamfer is too small so system can’t calculate it.		
Reason	Programming error.		
Solution	1. Check NC program to confirm that whether the block is compatible with specifications. 2. Check and confirm the working plane and round corner plane are the same.		

Alarm ID	**COR–036** **BGND-036**	**Alarm title**	**Inappropriate A angle command**
Description	A is valid only in linear interpolation of single block.		
Reason	Programming error.		
Solution	Check NC program to confirm that whether A angle command is compatible with specifications		

Alarm ID	**COR–037** **BGND-037**	**Alarm title**	**Chamfer value bigger than displacement**

Description	Chamfer value is bigger than pre-block and post-block length, so system can't calculate it.		
Reason	Programming errors.		
Solution	1. Check NC program to confirm that whether inserted value of chamfer is compatible with specifications. 2. Check to confirm whether C value is compatible with specification.		
Alarm ID	**COR–038** **BGND-038**	**Alarm title**	**Incorrect block jump's switch number.**
Description	Incorrect block jump switch number. i.e. / 2 G00 X100. ; If assigned number is bigger than 9 or equals to 0, system will issue this alarm.		
Reason	Block jump switch number is 0 or bigger than 9.		
Solution	Check the NC program, and confirm block jump switch number is between 1 and 9.		

Alarm ID	**COR–039** **BGND-039**	**Alarm title**	**Measure function can't start tool radius compensation**
Description	Measure function can't start tool radius compensation.		
Reason	Measure function related instruction (i.e., G31, G31.10, G31.11) is executed after tool compensation command.		
Solution	Confirm there is no measure function related command is executed after tool compensation command.		

Alarm ID	**COR–040** **BGND-040**	**Alarm title**	**Block end point exceed software stroke limit**
Description	The machine coordinate in the program exceeds the software stroke limit.		
Reason	Programing error.		
Solution	Check the NC program, and correct coordinate position.		

Alarm ID	**COR–041** **BGND-041**	**Alarm title**	**GOTO label must be integer**
Description	The input GOTO label is not an integer. i.e. GOTO 1 Correct GOTO 1. Wrong N1; Correct N1.; Wrong		
Reason	Programming error.		
Solution	Check the NC program, and input integer in GOTO argument.		

Alarm ID	**COR–042** **BGND-042**	**Alarm title**	**Logic operand is non-integer or empty**
Description	Logic operand must be non-integer or empty.		
Reason	Logic operand has floating point. i.e. #1=1.5And3. System will issue this alarm.		
Solution	Please check NC program. Confirm logic operand is integer or empty.		

Alarm ID	**COR–043** **BGND-043**	**Alarm title**	**ASIN, ACOS operand must between ±1.0**
Description	ASIN() and ACOS() operand is not between -1.0 and 1.0.		
Reason	Programming error.		
Solution	Check the NC program.		

Alarm ID	**COR–044** **BGND-044**	**Alarm title**	**SQRT operand should not be negative**
Description	The square root of a negative value will be imaginary number, but the controller does not provide imaginary number function.		

Reason	Programming error.		
Solution	Check the NC program; enter a positive value in SQRT operand.		
Alarm ID	**COR–045** **BGND-045**	**Alarm title**	**L address should be integer**
Description	The L address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in L address.		
Alarm ID	**COR–046** **BGND-046**	**Alarm title**	**O address should be integer**
Description	The O address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in O address.		
Alarm ID	**COR–047** **BGND-047**	**Alarm title**	**M address should be integer**
Description	The M address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in M address.		
Alarm ID	**COR–048** **BGND-048**	**Alarm title**	**Spindle speed S should be integer**
Description	The spindle speed S is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in the spindle speed S.		

Alarm ID	**COR–049** **BGND-049**	**Alarm title**	**Tool length compensation H should be integer**
Description	Tool length compensation H is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool length compensation H.		
Alarm ID	**COR–050** **BGND-050**	**Alarm title**	**Tool radius compensation D should be integer**
Description	Tool radius compensation D is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool radius compensation.		
Alarm ID	**COR–051** **BGND-051**	**Alarm title**	**Tool number T should be integer**
Description	Tool number T is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use tool number T in integer.		
Alarm ID	**COR–052** **BGND-052**	**Alarm title**	**Sub-program number, P, should be integer**
Description	Sub-program number P is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use the sub-program number P in integer.		
Alarm ID	**COR–053** **BGND-053**	**Alarm title**	**Repeat count L should be integer**

Description	Repeat count L is not an integer.
Reason	Programming error.
Solution	Please check the NC program, and use the repeat count L in integer.

Alarm ID	**COR–054** **BGND-054**	**Alarm title**	**Incompatible data type**
Description	Data format is incompatible with controller specifications.		
Reason	NC program is not compatible with the SYNTEC controller.		
Solution	Make sure that the NC program data format is compatible with Syntec controller specifications.		

Alarm ID	**COR–055** **BGND-055**	**Alarm title**	**Tool length compensation H out of range**
Description	The tool length compensation H exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool length compensation H, is in the range of tool number.		

Alarm ID	**COR–056** **BGND-056**	**Alarm title**	**G10 table index P is out of range**
Description	G10's format is G10 L_ P_ R_; Different number L will correspond to different number P. i.e. L10 corresponding number P is tool number. Input P1000 means the 1000th tool. If that tool number doesn't exist, and controller will issue this alarm.		
Reason	Programming error.		
Solution	Confirm G10 data table address number P is in reasonable range.		

Alarm ID	**COR–057** **BGND-057**	**Alarm title**	**Tool radius D out of range**

Description	The tool radius number D exceeds the range of tool number.
Reason	Programming error.
Solution	Make sure that the tool radius number D is in the range of tool number.

Alarm ID	**COR–058** **BGND-058**	**Alarm title**	**Tool nose compensation D is out of range**
Description	The tool nose compensation D exceeds the range of tool number.		
Reason	Program error.		
Solution	Make sure that the tool radius compensation D is in the range of tool number.		

Alarm ID	**COR–059** **BGND-059**	**Alarm title**	**Subprogram Call H must be integer**
Description	Subprogram call H is not an integer.		
Reason	Program error.		
Solution	Confirm the subprogram call H is an integer.		

Alarm ID	**COR–060** **BGND-060**	**Alarm title**	**M99 return number P must integer**
Description	The M99 return sequence number P is not an integer.		
Reason	Program error.		
Solution	Confirm the M99 return sequence number P is an integer.		

Alarm ID	**COR–061** **BGND-061**	**Alarm title**	**Workpiece number is out of range**
Description	The number of work pieces exceeds the allowable range of the controller.		
Reason	Program error.		

Solution	Make sure the number of work pieces is in the allowed range of the controller.

Alarm ID	**COR–062** **BGND-062**	**Alarm title**	**Dwell skip source Q must be integer**
Description	Dwell skip source Q is not an integer.		
Reason	Program error.		
Solution	Change the dwell skip source Q into an integer.		

Alarm ID	**COR–063** **BGND-063**	**Alarm title**	**Dwell skip source Q out of range**
Description	Dwell skip source Q exceeds the allowed range.		
Reason	Program error.		
Solution	Make sure the dwell skip source Q is in the allowed range.		

Alarm ID	**COR–064** **BGND-064**	**Alarm title**	**Invalid P address**
Description	Invalid P address.		
Reason	1. Address P is less than 0. 2. Address P is not integer. 3. Address P is out of range.		
Solution	1. Make sure address P is greater or equal to 0. 2. Make sure address P is integer. 3. Make sure address P is within the range.		

Alarm ID	**COR–065** **BGND-065**	**Alarm title**	**,A command format error**
Description	In NC program ,A command's format is wrong.		
Reason	Programming error.		

Solution	Please confirm ,A command format is one of the followings: 1. In the same block command as ,A, Moving instruction only includes one of x axis or z axis. i.e. G01 X10. ,A30. 2. Use ,A advanced function in two separate line. The first line will assign ,A angle, and the second line will assign X, Z and ,A at the same time. i.e. G01 ,A30. G01 X0. Z50. ,A45

Alarm ID	**COR–066** **BGND-066**	**Alarm title**	**Inc. axis command and abs. axis command conflict**
Description	Both G91 and G90 are in the same line.		
Reason	Programming error.		
Solution	Decide to use incremental or absolute command in one line, and enter the correct command.		

Alarm ID	**COR–067** **BGND-067**	**Alarm title**	**Arc center vector and radius conflict**
Description	The arc end point is not on the arc created by the arc starting point and the specify center.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	**COR–068** **BGND-068**	**Alarm title**	**Quiet Mode not support imperial Metric switch command**
Description	The single block command in Quiet mode operation cannot switch Metric/ imperial unit.		
Reason	Programming error.		
Solution	Please check the NC program.		

Alarm ID	COR–069 BGND-069	Alarm title	Round corner and chamfer cmd. conflict
Description	Chamfer command and round corner command are in the same line.		
Reason	Program error.		
Solution	Do not let chamfer command and round corner command exist in the same line.		

Alarm ID	COR–070 BGND-070	Alarm title	Invalid G Code
Description	Enter incorrect G code to controller.		
Reason	Program error.		
Solution	Enter the valid G-code.		

Alarm ID	COR–071 BGND-071	Alarm title	No main program name assigned
Description	The name of main program is not specified.		
Reason	The NC program is not loaded.		
Solution	Specify the name of main program.		

Alarm ID	COR–072 BGND-072	Alarm title	Threading command exceeds max. cutting speed
Description	Threading command exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of threading.		

Alarm ID	COR–073 BGND-073	Alarm title	Tapping command exceed max. cutting speed

Description	Tapping command speed exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of tapping.		
Alarm ID	**COR–074** **BGND-074**	**Alarm title**	**Tool radius too big, path overcut**
Description	1. During machining notch, the notch width is smaller than two times of tool radius. 2. During machining trapezoidal, the height of trapezoidal is less than two times of tool radius.		
Reason	Tool compensation causes path overcut.		
Solution	Please check the NC program, and decide whether this part of the machining should cancels tool radius compensation.		
Alarm ID	**COR–075** **BGND-075**	**Alarm title**	**Exact stop wait timeout**
Description	After 2 seconds after sending Exact stop (G09/G61) command, the difference between feedback and command exceeds allowable value (G01 is determined by Pr421~; G00 is determined by Pr481~).		
Reason	Servo vibration.		
Solution	1. Servo tuning. 2. Modify parameters Pr421~ or Pr481~.		
Alarm ID	**COR–076** **BGND-076**	**Alarm title**	**G04 dwell time cannot be negative**
Description	Input value of dwell time G04 is negative.		
Reason	Program error.		
Solution	Check the NC program, and enter a positive or zero value in G04 argument.		

Alarm ID	COR-077 BGND-077	Alarm Title	NURBS curve format is wrong
Description	G6.2 format is wrong.		
Reason	Input argument P, K, R is against allowable format.		
Solution	Refer the program manual to correct the command syntax.		

Alarm ID	COR-078 BGND-078	Alarm Title	NURBS curve system memory insufficient
Description	NURBS curve memory is insufficient.		
Reason	System processing defective.		
Solution	Contact OEM Syntec.		

Alarm ID	COR-079 BGND-079	Alarm Title	Number of Nurbs curve nodes more than the upper limit
Description	NURBS curve control point is over limit.		
Reason	NC file G6.2 curve is too long and cause using too many control point.		
Solution	Re-generate the CAM, and restrict the number of curve nodes less than 400.		

Alarm ID	COR-080 BGND-080	Alarm Title	Threading pitch is negative
Description	Threading pitch distance can’t be negative value.		
Reason	Threading pitch distance setting error.		
Solution	Check and correct the configured threading pitch to be positive value.		

Alarm ID	**COR-081** **BGND-081**	**Alarm Title**	**Use arc interpolation under tool offset function. Arc command doesn't use assigned angle.**
Description	In tool offset function (G45~G48), G02 and G03 can only be 90 degree or 270 degree.		
Reason	In NC program's tool offset function (G45~G48) block, all arc interpolation angle can only be 90 degree or 270 degree.		
Solution	Check if G45~G48 is 90 or 270 degree when using them in the arc interpolation (G02/G03). Note, IJK to specify circle center shall be used.		

Alarm ID	**COR-082** **BGND-082**	**Alarm Title**	**Tool compensation function and tool offset function can't use at the same time**
Description	Can't use tool offset function (G45~G47) and tool compensation function (G41, G42) or tool radius compensation at the same time.		
Reason	NC program edit error.		
Solution	Check NC program, and confirm tool offset function (G45~G47) and tool compensation function (G41, G42) are not used at the same time.		

Alarm ID	**COR-083** **BGND-083**	**Alarm Title**	**When pull tool back to pause point, and move axis position manually.**
Description	Before pull tool back to pause point, press stop and manually move tool.		
Reason	Operation error.		
Solution	Please wait until tool is pulled exactly back to pause point, then execute the manual position movement.		

Alarm ID	**COR-084** **BGND-084**	**Alarm Title**	**P Argument unassigned when using the rapid drilling**
Description	When using rapid drilling function, must set P argument (bottom hole dwell time), or system will issue this alarm.		

Alarm ID	COR-084 BGND-084	Alarm Title	P Argument unassigned when using the rapid drilling
Reason	Programming error.		
Solution	Check the NC program to ensure the P Argument is assigned when using the G code of rapid drilling.		

Alarm ID	COR-085 BGND-085	Alarm Title	F Argument unassigned in Inverse Time Feed mode
Description	Under Inverse Time Feed mode, if F argument is not assigned in command line and system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure there is F command is given following in each line after Inverse Time Feed mode.		

Alarm ID	COR-086 BGND-086	Alarm Title	R and I arguments unassigned in G10 L1501 mode
Description	G10 L1501 argument only has argument R or argument I.		
Reason	Programming error.		
Solution	Check the NC program to ensure the R and I arguments are not used independently.		

Alarm ID	COR-087 BGND-087	Alarm Title	P and Q arguments must be different in G10 L1501 mode
Description	P and Q argument in G10 L1501 mode setting are the same.		
Reason	Programming error.		
Solution	Please check program, and confirm P and Q argument in G10 L1501 mode aren’t the same.		

Alarm ID	COR-088 BGND-088	Alarm Title	No time-axis command after the I argument in G10 L1501 mode
Description	In spring machine dedicated machine's motion plan, if use I argument in G10 L1501, before the next G10 L1502 command, user can’t command P argument axis number (time axis) in G1 0L1501.		
Reason	Programming error.		
Solution	Check the NC program.		

Alarm ID	COR-089 BGND-089	Alarm Title	Main axis must be increment command in G10L1501 mode
Description	G10 L1501 mode spindle must be incremental command.		
Reason	Programming error.		
Solution	Under G10 L1501 mode, confirm the command which sends to spindle is incremental command.		

Alarm ID	COR-090 BGND-090	Alarm Title	C-axis zero-speed check failed in G12.1 mode
Description	When start to switch polar coordinate, C axis must be totally stop.		
Reason	When C axis uses as spindle, but spindle not yet stop rotating.		
Solution	Switch the spindle to C-axis mode before activating polar coordinate.		

Alarm ID	COR-091 BGND-091	Alarm Title	Main axis synchronization – basic main axis number error
Description	When using spindle synchronous function G114.1, Pr4021, Pr4023, Pr4025 can’t be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4021/Pr4023/Pr4025 are correct.		

Alarm ID	COR-092 BGND-092	Alarm Title	Main axis synchronization – synchronization main axis number error
Description	When using spindle synchronous function G114.1, Pr4022, Pr4024, Pr4026 can't be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4022/Pr4024/Pr4026 are correct.		

Alarm ID	COR-093 BGND-093	Alarm Title	Main axis synchronization – spindle type error
Description	When using spindle synchronous and Superimposition function, setting wrong spindle motor type.		
Reason	Parameter setting error.		
Solution	Check 1791~Pr1800 to set correct spindle configurations by referring to HELP.		

Alarm ID	COR-094 BGND-094	Alarm Title	Under spindle load rigid tapping spindle rotation speed exceed
Description	When using spindle load function to do rigid tapping, tool axis must synchronize with offset axis speed, and then do end surface taping according to user setting tapping speed. If tool axis speed is over spindle rotation speed maximum value, and system will issue this alarm.		
Reason	Command is over range that mechanical can bear.		
Solution	1. Reduce the rotation speed (tapping speed) of tool axis. 2. Reduce the rotation speed of workpiece axis.		

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (P argument) must be bigger than zero.		
Reason	Programming error.		

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Solution	Check the value of P argument in G51.2.		

Alarm ID	COR-096 BGND-096	Alarm Title	In polygon cutting function, synchronous spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (Q argument) must be bigger than zero.		
Reason	Programming error.		
Solution	Check the value of Q argument in G51.2.		

Alarm ID	COR-097 BGND-097	Alarm Title	Axis coupling function ON or OFF failed
Description	Axis coupling function ON or OFF failed.		
Reason	Too much axis coupling groups is used.		
Solution	Check if the number of axis coupling groups is more than 16 (including the number of axis coupling group configured in parameter).		

Alarm ID	COR-098 BGND-098	Alarm Title	Parameter learning argument error
Description	When using learning function, if argument (P, Q, R, K) setting is wrong, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure the arguments are in compliance with specifications.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

Alarm ID	COR-099 BGND-099	Alarm Title	Tapping learning condition is not matched
Description	Condition of tapping learning function is not matched		
Reason	When activating tapping learning function, if tapping condition (tapping depth, R point height, feedrate, rotation speed, and etc.) is different, it can't apply to same learning data. If force different tapping condition to use same learning data, it may misuse learning data, and cause tapping error or even tool break.		
Solution	Check tapping instruction in tapping learning function, and confirm all tapping instruction arguments are the same.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

Alarm ID	COR-100 BGND-100	Alarm Title	Unsupported G code command or option software is not activated
Description	Different controllers will have correspond G code, but not all G code can use.		
Reason	1. This controller type may not support this G code command. 2. This controller type will not support serial bus spindle (C-Type) to use lathe G21, G33, G34, G78 commands. 3. This controller type will not support serial bus spindle (A-Type) to use lathe G32, G73, G76, G92 commands. 4. This controller type can support this G code command, but the option software function has not been purchased, which makes the G code unusable. 5. Loader path and Woodworking label path only support part of G codes: G00, G01, G02, G03, G04, G04.1, G09, G10, G17, G18, G19, G22, G23, G31, G52, G53, G54, G55, G56, G57, G58, G59, G59.x, G90, G91, G92. 6. The setting of Pr3802 is incorrect. This controller type does not support the G62 command.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Description	When using spindle synchronize, if basic spindle and synchronous spindle pitch setting are different and will cause synchronous abnormal condition. And will issue this alarm for protection.		

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Possible Cause	Parameter setting error.		
Solution	Check the spindle used in Pr161~180 to ensure the parameter configurations are the same.		

Alarm ID	COR-102 BGND-102	Alarm Title	Under Spindle synchronization – synchronization command duplicated or conflicted
Description	When spindle synchronizing, repeat G51.2, G114.1 or G114.3 commands.		
Possible Cause	Haven't use G113 to cancel G114.1 and G114.3, or use G50.2 to cancel G51.2, and again set a repeated G114.1, G114.3 or G51.2 commands.		
Solution	Cancel the G114.1 and G114.3 mode with G113, or cancel G51.2 mode with G50.2. Then execute the G114.1/G114.3/G51.2 commands to avoid the angle difference for repeated machining.		

Alarm ID	COR-103 BGND-103	Alarm Title	Invalid precision channel setting
Description	Invalid precision channel setting.		
Possible Cause	Using G64, G62P_ or G120.1 P_Q_ to choose invalid precision channel. Currently can only choose: 1. G64、G62 P0~P9、G62/G64 P21 ~ P23 2. G120.1 P0、G120.1 P1 Q1 ~ G120.1 P3 Q3		
Solution	To select precision channel with the G64/G62 P_ or G120.1 P_ Q_. Check the NC program to ensure if any non-existing precision channel is selected.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Description	The virtual axis function failed to be functioned.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Possible Cause	1. The P and Q arguments in G10 L800 P_ Q_ or G10 L801 P_ Q_ are set incorrectly. a. No P, Q arguments. b. The P argument is set incorrectly and the value is invalid. c. The Q argument is set incorrectly, the value is invalid; or the corresponding axis is not set to the axis of the parameter table that has been turned on. 2. In the same machining, G10 L800 and G10 L801 appear at the same time. 3. The robot does not support the virtual axis function.		
Solution	1. Check the virtual axis commands to ensure the P and Q Arguments are configured. 2. Check the virtual axis commands to ensure the P Argument shall be within 100~999 and the Q Argument shall be within 1~16 (axis number) or 100~999 (axis name). 3. Check the axial direction (Q Argument) mapped with the virtual axis to ensure the parameter table is configured to the enabled axial direction. 4. Check if the G10 L800 and G10 L801 shall not be existent in the same NC program.		

Alarm ID	COR-105 BGND-105	Alarm Title	Axial coupling configuration error
Description	The axis coupling function is set incorrectly.		
Possible Cause	In G10 L900 P_ Q_ R_, the P and Q arguments are set incorrectly.		
Solution	1. Check the name or number of coupling axis is mapped to ensure the axial direction is enabled. 2. Chedk the specified coupling axis is not assigned to inclined axes control. 3. Check the specified coupling axis is not assigned to indexing axis.		

SYNTEC

Alarm ID	COR-106 BGND-106	Alarm Title	Prohibit to use G5.1 in G61/G63/ G63.2 mode
Description	1. G5.1 path smoothing. In G61, G63, G63.2 mode, it is forbidden to use G5.1 smoothing function, otherwise the system will issue an alarm; If G61, G63, G63.2 are enabled in G5.1 mode, the system will stop the smoothing function until it jumps out of G61, G63, G63.2 mode and then activate again. 2. G05 High Precision Contour Control. a. 10.116.16J previous version, in G61, G63, G63.2 mode, it is forbidden to use G05 High Precision Contour Control function, otherwise the system will issue an alarm; If G61, G63, G63.2 is enabled in G05 mode, no alarm will be issued, but the High Precision Contour Control function is disabled, It is necessary to command G05 again to activate. b. above 10.116.16K versions, no alarm will be issued, and the G05 High Precision Contour Control function will be disabled, and can be valid after G61, G63 and G63.2.		
Possible Cause	The cutting mode setting in the NC program is incorrect.		
Solution	1. Do not mix use G5.1 path smoothing with G61/G63/G63.2 functions. 2. In the version before 10.116.16J (included), do not mix use the G05 High Precision Contour Control with the G61/G63/G63.2 functions.		

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Description	The G5.1 and G05 commands are in the wrong format.		
Possible Cause	1. The format of the G5.1 path smoothing command in the NC program is incorrect. 2. The G05 high-precision cutting mode command format in the NC program is incorrect.		

SYNTEC

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Solution	Confirm the following command formats are correct not have these error: 1. G5.1 a. Q argument: None, more than 2, or less than 0. b. E argument: None or less than 0. 2. G05 a. System issue alarm when using G05 in following cases for each version: i. G05 P argument is not 10000 nor 0. ii. G05 E argument is not positive. b. Activate command G05 P10000 X0 Y0 Z0 α_ β_ in 10.116.36 or above versions: i. More than 5 axial directions are assigned. ii. The geometry axis argument not 0. iii. The rotary axis argument is configured to 0. iv. The axial direction of geometry axis is configured but this of rotary axis is not. v. The axis of the rotation axis is not set when the axis of the geometry axis is not set. vi. More than 2 axial directions of rotation axes are configured. vii. Any axial arguments is negative. c. In the version before 10.116.16B, there is the 4th axis command in addition to the block movement commands of X, Y, or Z axes after G05 is executed.		

Alarm ID	COR-108 BGND-108	Alarm Title	G10 L1501/L1502 command format error
Description	There is no NC program between G10 L1501 and G10 L1502.		
Possible Cause	The NC program was written incorrectly.		
Solution	Check the NC program to ensure the programming between G10 L1501 and G10 L1502 is correct.		

Alarm ID	COR-109 BGND-109	Alarm Title	The second software stroke limit command error, activating failure
Description	The second software stroke limit (G22) command is written incorrectly, causing the second software stroke limit fail to activate.		
Possible Cause	The arguments written after the second software stroke limit (G22) command are wrong, and the same group parameters should exist at the same time.		
Solution	Check if the arguments in the same set after G22 are defined completely.		

Alarm ID	COR-110 BGND-110	Alarm Title	NC program of spring machine motion plan more than 500 lines
Description	The number of machining lines between G10 L1501 and G10 L1502 exceeds 500 lines.		
Possible Cause	There are too many single blocks in the NC program.		
Solution	Check and refine the NC program.		

Alarm ID	COR-111 BGND-111	Alarm Title	Axis exchange function – axial configuration error
Description	Under the axis exchange function, the axial setting is incorrect.		
Possible Cause	The two axial's axis exchange parameters are set incorrectly.		
Solution	Check if Pr3721 and Pr3722 are mapped to physical axial directions (Pr21~) and to 2 different axis.		

Alarm ID	COR-112 BGND-112	Alarm Title	Axis exchange function – diameter/radius axis configuration error
Description	Under the axis exchange function, the diameter axis and radius axis are set incorrectly.		
Possible Cause	The parameters of the diameter and radius axis of the two axis exchange are set incorrectly.		
Solution	Check if the diameter and radius axis configurations (Pr281~) of the 2 axis to be exchanged is consistent.		

Alarm ID	COR-113 BGND-113	Alarm Title	Axis exchange function – ON or OFF timing error
Description	When the axis exchange function is enabled, the enabled or disabled timing error.		
Possible Cause	When the cross-path axis is exchanged, the G04.1 waiting action is not performed.		
Solution	Check PLC to ensure no G04.1 waiting command is executed in more than 1 path.		

Alarm ID	COR-114 BGND-114	Alarm Title	Axis exchange function – path configuration error
Description	When using the axis exchange function, the path setting is incorrect.		
Possible Cause	The axis used for axis exchange belong to multi-paths.		
Solution	Check path parameter (Pr701~) if the two axis to be exchanged are mapped to the multiple path.		

Alarm ID	COR-115 BGND-115	Alarm Title	G92.1 axial configuration error
Description	When using the G92.1 rotation function, the axial setting is incorrect.		
Possible Cause	The parameter sets the first 3 axis of the path contains the rotary axis.		
Solution	Check the parameter to ensure the first 3 axis are linear axis.		

Alarm ID	COR-116 BGND-116	Alarm Title	Absolute coordinate system synchronization failed
Description	Absolute coordinate system failed to synchronize when using C35.		
Possible Cause	Machine is not stationary or the program is continuously interpreted.		
Solution	To execute the synchronization of coordinate system in C35 mode shall ensure the machine is still and the program stops to interpret (i.e. Using M code shall turn C38 off).		

Alarm ID	COR-117 BGND-117	Alarm Title	M code not disabled in the interrupt subprogram
Description	If Pr3600 is set to n, when the interrupt type subprogram is used, it starts with M(n) and ends with M(n+1). If M(n) is used and M(n+1) is not used to end the function, it will cause an action error; therefore, this alarm is issued for protection.		
Possible Cause	The NC programming error.		
Solution	Check the NC program to ensure the M(n+1) command is given.		

Alarm ID	COR-118 BGND-118	Alarm Title	Prohibit G53 commands in tool tip control mode
Description	G53 command cannot be used in the tool point control mode.		
Possible Cause	1. The NC programming error. 2. The machine type is the tool point control mode.		
Solution	1. Please check the NC program, make sure that the G53 command is not within the validity range of G43.4 or G43.5. 2. Please check the NC program, make sure that the G53 command is not within the validity of G12.1. 3. If the machine configuration used is the tool point control mode, the G53 command cannot be used.		

Alarm ID	COR-119 BGND-119	Alarm Title	G10 L16 command format error
Description	Since the command G10 L16 (virtual circle radius) does not support functions such as G05, G7.1, G12.1, G93, G95, and G05, therefore, an alarm will be issued if G10 L16 is used when the above command is issued.		
Possible Cause	1. The axis type is linear axis (i.e. G10 L16 X50., and X is set to a linear axis, this alarm is issued). 2. When using G10 L16 while command G05, G7.1, G12.1, G93, G95 and etc. functions are also given.		
Solution	1. Check Pr221~ to ensure the correct axis type. 2. G10 L16 and G7.1/G12.1/G93/G95/G05 shall not be executed at the same time.		

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Description	Different controller products can correspond to different maximum number of simultaneous cutting axis, and if this limit is exceeded, an alarm will be issued. (i.e. G01 X10. Y10 .Z10. Represents the number of simultaneous axes is 3).		
Possible Cause	Please refer to "System Information" under the controller HMI screen to check "Machine Properties" and "Machine Code". Please refer to the catalogue for each product specification. Following examples: 1. If "Machine Properties" is Mill, "Machine Code" is 200A-5, and the maximum number of axis is 9. 2. If the "Machine Properties" is Lathe, the "Machine Code" is 6B, and the maximum number of axis is 4.		

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Solution	The number of simultaneous axis in the NC program shall be according to the product specifications.		

Alarm ID	COR-121 BGND-121	Alarm Title	LN operator shall be positive
Description	The operands after the LN function cannot be negative.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Description	The base operand of the POW function must not be a negative number.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-123 BGND-123	Alarm Title	Illegal STR2INT input or too long string
Description	The syntax of the STR2INT function is incorrect or the string is too long.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Description	This model does not support S code commands.		
Possible Cause	This model does not support S code commands.		

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Solution	Replace a controller model that supports S codes.		

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Description	T code command is illegal.		
Possible Cause	1. This model does not support T code commands. 2. The T code command is out of the T code range supported by the product.		
Solution	Replace a controller model that supports T codes.		

Alarm ID	COR-126 BGND-126	Alarm Title	H code commands unsupported in the machine
Description	This model does not support H code commands.		
Possible Cause	This model does not support H code commands.		
Solution	Replace a controller model that supports H codes.		

Alarm ID	COR-127 BGND-127	Alarm Title	D code commands unsupported in the machine
Description	This model does not support D code commands		
Possible Cause	This model does not support D code commands		
Solution	Replace a controller model that supports D codes.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Description	The total number of M codes and T codes in the same block exceeds 20.		
Possible Cause	NC programming error.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Solution	Check the NC program to ensure the total number of M and T codes are not more than 20 in one single block.		

Alarm ID	COR-132 BGND-132	Alarm Title	Illegal character in the program name
Description	When open a file using Macro, the specified file name contains illegal characters.		
Possible Cause	NC programming error.		
Solution	Check the specified file names.		

Alarm ID	COR-133 BGND-133	Alarm Title	Command unsupported in Three-points arc interpolation mode
Description	This command is not supported in the three-point arc interpolation mode (G02.4, G03.4)		
Possible Cause	1. The tool radius compensation function is not turned off before using this function. 2. It is not supported in G62 cutting mode. 3. In the three-point arc interpolation mode (G02.4, G03.4), A, C, and R commands are not supported.		
Solution	Check the NC program to ensure there is no command described in the Possible Cause.		

Alarm ID	COR-134 BGND-134	Alarm Title	Three-points arc interpolation command format error
Description	The three-point arc interpolation (G02.4, G03.4) command format is wrong.		
Possible Cause	The three-point arc interpolation (G02.4, G03.4) command is regarded as a group of two lines, which can be specified continuously. The end point of the previous arc is the starting point of the next arc, but the F command can only be in the odd line. This alarm is issued if the total number of lines in the command is odd or the number of lines in the F command is even.		
Solution	Check the three-point arc interpolation G02.4/G03.4 command format in NC program.		

Alarm ID	COR-135 BGND-135	Alarm Title	Read/write command format error for R value
Description	SETRREGBIT, READRREGBIT bring in an argument with type error or range error.		
Possible Cause	1. SETRREGBIT (R value number, specify BIT, on or off) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31. c. If the third argument is not 0 (off) or 1 (on). 2. READRREGBIT (R value number, specify BIT) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31.		
Solution	Correct the argument type or range in the commands, SETRREGBIT and READRREGBIT		

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Description	After a certain serial bus axial tuning fails, without rebooting, cycle start is triggered.		
Possible Cause	The serial bus axial tuning failed.		
Solution	Reboot controller.		

Alarm ID	COR–137 BGND-137	Alarm title	Path synchronization waiting's P-argument sequence error
Description	The machining multi-path program uses the path synchronization waiting (G04.1) to wait for the synchronization action: 1. When the Q argument is not selected, when two programs go to the position of the path synchronization waiting (G04.1) at the same time, if the P arguments substituted are different, this alarm is issued for reminder. 2. When the next Q argument (decimal) specifies the path waiting for each other, when the two programs go to the position of the path synchronization waiting (G04.1) at the same time, and wait for each other, but the P arguments substituted are different, this alarm is issued for reminder.		

Possible Cause	1. In the NC program of multiple paths, the order of P arguments is abnormal or the number is incorrect. 2. Machining is performed using M99, but in each program, the path without Q argument is synchronously waiting (G04.1), and the number is different. 3. Machining is performed using M99, but in each program, the path with the same Q argument is synchronously waiting (G04.1), and the number is different. 4. The two paths “wait for each other”, but the P argument substituted are not the same. i.e. G04.1 P1 Q123 under the first path and G04.1 P2 Q124 under the second path.
Solution	Please check the number of Synchronization wait between paths (G04.1) and the order of the assigned value for argument P.

Alarm ID	**COR-138** **BGND-138**	**Alarm Title**	**Read/write command format error at the I/O/A point**
Description	The system provides 512 I/O/A points, the input I/O/A point number shall be within 0~511 when using the commands, SETDO, SETABIT, READDI, READDO, and READABIT.		
Possible Cause	NC Programming error.		
Solution	Check if there is any of I/O/A point number in the commands, SETDO, SETABIT, READDI, READDO, and READABIT in the NC program is over the range of 0 ~ 511.		

Alarm ID	**COR-139** **BGND-139**	**Alarm Title**	**Polynomial solution error**
Description	Determine the polynomial cannot be derived to solutions in case of the alarm.		
Possible Cause	Numerical processing calculation errors cause root failure.		
Solution	Check the value of system data no. 321, and provide it to the controller OEM Syntec for solutions.		

Alarm ID	**COR-140** **BGND-140**	**Alarm Title**	**Invalid high-precision contour control mode using**
Description	1. Turn on G05 high-speed high-precision mode in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Possible Cause	1. In the RTCP/STCP mode, turn on the G05 high-precision contour control mode with commands, such as G05 P10000. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		
Solution	1. Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-precision contour control mode. If (1), remove the command to turn on the G05 high-precision contour control mode in the RTCP/STCP mode. If (2), turn off the RTCP/STCP mode before turning on the G05 high-precision contour control mode. 2. When the high-precision contour control mode is enabled during processing, do not use the single block stop C40 at the same time. 3. When the STCP mode is enabled during processing, do not use the single block stop C40 at the same time.		

Alarm ID	COR-141 BGND-141	Alarm Title	G68.3 command format error
Description	[command format] G68.3 X_ Y_ Z_ R_; // The origin and z-axis rotation angle in the characteristic coordinate system. G68.3 P1 X_ Y_ Z_; // The origin of the characteristic coordinate system, and the coordinate system is determined with the tool rotation angle.		
Possible Cause	G68.3 command format, X, Y and Z are all exist or non-exist at the same time.		
Solution	Check if G68.3 command format is correct.		

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Description	K Argument error while using Spindle synchronization function.		
Possible Cause	The inputted K Argument is not within the valid range.		

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Solution	Input a K Argument within the valid range to enable the Spindle synchronization.		

Alarm ID	COR-143 BGND-143	Alarm Title	Programmable data input specified axis does not exist
Description	In G10 L1501 mode, the P argument is set incorrectly.		
Possible Cause	The axial direction substituted by the P Argument is inexistent.		
Solution	Check Pr21~ to ensure the axial direction is ON.		

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Description	When the machining multiple path program applies the G04.1 to wait for synchronization, the Q Argument assigns the paths to be waited for each other (decimally). The alarm is used to remind the users if the Q Argument type error or the assigned path is inexistent.		
Possible Cause	1. The Q Argument is not a positive integer (negative number, decimal point and zero are all illegal). 2. The path assigned by the Q Argument is inexistent. i.e. Q24 is used to assign the 2^{nd} and 4^{th} path to wait for each other. However, the Pr731=3 and there are only 3 CNC main system paths. 3. The Q Argument includes 0 when assigning the paths. i.e. Q103. 4. The paths assigned by the Q Argument do not include this in which the command is. ie. In the 2^{nd} path, the G04.1 P1 Q13 assigns the 1^{st} and 3^{rd} paths to wait for each other, but it does not assign the path in which it is.		
Solution	Check the Path synchronization waiting (G04.1) in the program to ensure the type and the assigned path are correct for a given Q Argument.		

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Description	Spindle positioning failed.		

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Possible Cause	1. Incorrect Spindle ID is assigned. 2. The Spindle is operating in the tapping mode when the Spindle positioning function is ON. 3. After starting the positioning, C61 is OFF so that the positioning is broken.		
Solution	1. Check the Spindle parameter configuration. 2. Check the NC program to ensure the execution of Spindle positioning in non-tapping mode. 3. Check the PLC to ensure C61 is ON.		

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Description	Argument type error in a block.		
Possible Cause	The two following situations may trigger this alarm: 1. Input float value to an argument required integer value. 2. Input integer value to an argument required float value.		
Solution	Correct the argument type in the block.		

Alarm ID	COR-147 BGND-147	Alarm Title	Spindle is not enabled when path machining
Description	The machining Spindle assigned by the path is not enabled when executing the commands related to the Spindle.		
Possible Cause	When executing the G33/G34/G63/G74/G84, the Spindle assigned by the R791~R794 is not enabled.		
Solution	1. Check if the PLC switches R791~R794 values correctly in the block and whole NC program about the alarm. 2. Check if the Pr1621~Pr1628 configurations are correct (cooperate with R791~R794).		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Description	When using the tool retract function, a given improper argument results in the function cannot be executed.		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Possible Cause	1. In the Tilted working plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned axial position retraction. 2. In the non-Tilted working plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned retraction along the tool vector.		
Solution	1. In the slopping plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, apply G10.6 R_ to assign the tool retraction function. 2. In the non-slopping plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, apply G10.6 X_ to assign the tool retraction function.		

Alarm ID	COR-149 BGND-149	Alarm Title	Tilted working plane machining tool alignment P Argument over range
Description	P Argument is over range in Tilted working plane machining tool alignment mode.		
Possible Cause	P Argument is not within 0~2.		
Solution	Do not input the P Argument or input the correction configuration for the P Argument in Tilted working plane machining tool alignment mode.		

Alarm ID	COR-151 BGND-151	Alarm Title	1st rotation axis entering illegal range
Description	1st rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1st rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3009 and Pr3010 are configured correctly. The determination of such two configurations is related to Pr3007. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	COR-152 BGND-152	Alarm Title	2nd rotation axis entering illegal range
Description	2nd rotation axis entering illegal range		
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3011 and Pr3012 are configured correctly. The determination of such two configurations is related to Pr3008. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	COR-153 BGND-153	Alarm Title	Tool direction unknown
Description	Tool direction unknown.		
Possible Cause	5-axis configurations and machine mechanism is incompatible.		
Solution	The tool cannot reach the destination. It may be caused by the incompatible 5-axis configurations and machine mechanism. Please check all 5-axis configurations.		

Alarm ID	COR-154 BGND-154	Alarm Title	No 5-axis function
Description	No 5-axis function.		
Possible Cause	Pr3001 is not configured when executing G53.1 tool alignment command.		
Solution	Check if Pr3001 is configured to 0. If yes, configure the other non-zero values based on the 5-axis mechanism type and reboot.		

Alarm ID	COR-155 BGND-155	Alarm Title	5-axis tool direction error
Description	5-axis tool direction error.		

Alarm ID	COR-155 BGND-155	Alarm Title	5-axis tool direction error
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd rotation axis (Pr3005 and Pr3006) configuration error.		
Solution	Check if the Pr3002 is configured correctly, or if the Pr3005 or Pr3006 is configured correctly. The alarm will be triggered in case the 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

Alarm ID	COR-156 BGND-156	Alarm Title	5-axis axial direction error
Description	5-axis axial direction error.		
Possible Cause	Incorrect configurations are mapped to the axial direction parameters of 5 axis.		
Solution	Check if each axial direction is configured completely (Pr21~), if Pr3005, Pr3006, Pr3007 and Pr3008 is configured correctly, or if the axis name (Pr321~) is mapped to Pr3005 and Pr3006.		

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Description	Incompatible direction of 5-axis tool direction and this of rotation axis.		
Possible Cause	1. The 2nd axial direction and the tool direction are the same in the Spindle type. 2. The 1st axial direction and the tool direction are the same in the table type. 3. The 1st axial direction and the tool direction are the same in the hybrid type.		
Solution	Check if the tool direction and the rotation axial direction are the same depended on the used 5-axis mechanism type.		

P.S. Valid version of COR-157 : before 10.118.41M, 10.118.47 (included).

Alarm ID	COR-158 BGND-158	Alarm Title	Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode
Description	Since the G43.5 mode specifies the tool attitude based on the tool vector I, J and K, it shall not be executed for the 1st and 2nd rotation axis commands which can also specify the tool attitude.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the movement commands of the 1st and 2nd rotation axis are over the valid range in the G43.5 mode.		

Alarm ID	COR-159 BGND-159	Alarm Title	Illegal tool vector
Description	In NC program, a movement block assigns an incorrect tool vector.		
Possible Cause	Programming error. i.e. Execute G01 X_ Y_ Z_ I0 J0 K0 in the G43.5 mode, and the I0 J0 K0 refers to the 0 vector, 0 vector is illegal.		
Solution	Check the NC program to ensure the assigned tool vector is correct in the block indicated by the alarm line number		

Alarm ID	COR-160 BGND-160	Alarm Title	5-axis mechanism chain switched when the 5-axis function is ON
Description	In NC program, the command G10 L5000 [P_] is executed to switch the 5-axis mechanism when turning on the 5-axis function (RTCP or the tool vector alignment on the characteristic coordinate system is completed).		
Possible Cause	Programming error. i.e. Execute G10 L5000 [P_] in the G43.4 mode.		
Solution	Check the NC program to ensure the 5-axis function is not turned on in the block indicated by the alarm line number.		

Alarm ID	COR-161 BGND-161	Alarm Title	Selected 5-axis mechanism chain is not ON
Description	In NC program, the 5-axis mechanism chain parameters assigned by the command G10 L5000 [P_] is not ON.		
Possible Cause	Programming error. i.e. The assigned 5-axis mechanism chain is not configured correctly when executing a command G10 L5000 [P_].		
Solution	Check the 5-axis mechanism chain parameters to ensure the assigned 5-axis mechanism chain function is turned on correctly: 1. 1st set: Pr3001 2. 2nd set: Pr3101 3. 3rd set: Pr5501 4. 4th set: Pr5601		

Alarm ID	COR-162 BGND-162	Alarm Title	4-axis RTCP configuration error
Description	The controller will issue the alarm in case the 4-axis RTCP configuration is incorrect.		
Possible Cause	The 5-axis mechanism parameters, Pr3001, Pr3101, Pr5501 and Pr5601 are configured to 1~3 in case the specific 4-axis tool tip control function (option-29) is turned on but the tool tip control function (option-12) is not.		
Solution	1. Configure the 5-axis mechanism parameters to 4 or 5 correctly. 2. Turn on the tool tip control function (option-12).		

Alarm ID	COR-163 BGND-163	Alarm Title	Multi-kinematic chain command Q Argument setting error.
Description	Command G10 L5000P_ Q_, Q argument range error.		
Possible Cause	Command G10 L5000P_ Q_, Q argument range error.		
Solution	While using G10 L5000P_ Q_, check Q argument to be within 0~4, and is a integer.		

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Description	Command G10 L5000P_ Q_ specified the 5-Axis kinematic chain, and the 5-Axis mechanism parameter setting error.		
Possible Cause	While executing G10 L5000P_ Q_, Q argument is given, but the 5-Axis mechanism parameter of the designated 5-Axis kinematic-chain is not a spindle-type 5-Axis machine.		
Solution	Please check the designated 5-Axis kinematic-chain. The 5-Axis mechanism configuration parameter must be a spindle-type 5-Axis machine. 1. The first group : Pr3001 is 1. 2. The second group : Pr3101 is 1. 3. The third group : Pr5501 is 1. 4. The fourth group : Pr5601 is 1.		

Alarm ID	COR-165 BGND-165	Alarm Title	Multi-kinematic chain command not illegal.
Description	Command G10 L5000P_ Q_ is used for switching 5-Axis kinematic chain, and only provides partial 5-Axis mechanism function command.		
Possible Cause	G10 L5000 P_ Q_ command, the Q argument is set to 2~4 (not the first group of sub-kinematic chain), and only supports the following 5-Axis machine function command. 1. RTCP: G43.4. 2. RTCP: G43.5. 3. Tilted working plane : G68.2 + Tool alignment functions. 4. Tilted working plane : G68.3. Notice: Tool alignment functions include G53.1, G53.3, G53.6, ...		
Solution	When using the G10 L5000 P_ Q_ command to switch multi-kinematic chains, please use the supported 5-Axis machine function command.		

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Description	Option13 (Characteristic Coordinate System Option) was not active, therefore the CNC couldn't execute relevant commands.		

Alarm ID	COR-166 BGND-166	Alarm Title	**Characteristic Coordinate System Option not supported**
Possible Cause	One or more commands below were given while the Option13 (Characteristic Coordinate System Option) was inactive: 1. G68.2, G68.3 2. G53.1, G53.3, G53.6 3. Other commands that are relevant to Option13 (Characteristic Coordinate System Option)		
Solution	1. Activate Option13 (Characteristic Coordinate System Option) 2. Avoid using the commands listed above		

Alarm ID	COR-167 BGND-167	Alarm Title	Program File name conflicts in NcFiles
Description	Program File name conflicts in NcFiles folder.		
Possible Cause	When Pr3220 is set to 1, "non-main system", "PLC axis", "pre- and post-processing auxiliary programs" restrict calling files from the Macro folder, and prohibit the NcFiles folder from having the same file name.		
Solution	1. Set Pr3220 to 0. 2. Remove the conflicting files in the NcFiles folder or change the file name.		

Alarm ID	COR-168 BGND-168	Alarm Title	**Illegal tool compensation.**
Description	Illegal tool compensation		
Possible Cause	1. Trying to use non-zero tool length or tool radius compensation value on a machine type which doesn't support tool compensation		
Solution	1. Set tool compensation value as zero 2. Turn off tool compensation related functions		

Alarm ID	COR-169 BGND-169	Alarm Title	Overlap on the same axis
Description	Overlap on the same axis.		
Possible Cause	Using G1.10 with argument Q on the same axis with two consecutive blocks.		
Solution	Please check NC Program and avoid overlapping on the same axis with two consecutive blocks.		

Alarm ID	COR-170 BGND-170	Alarm 标题	【G43.4 L2 illegal argument value】
说明	When using G43.4 L2. E_ R_, Illegal argument value is assigned		
可能原因	Illegal argument value 1. R value is negative 2. E value is not in the range of 0.001 to 179.999		
排除方法	Please check the E_ R_ argument values		

Alarm ID	COR–201 BGND-201	Alarm title	Program file not exist
Description	The specified program does not exist.		
Possible Cause	The specified program does not exist.		
Solution	Ensure that program file exists.		

Alarm ID	COR–202 BGND-202	Alarm title	Communication link error
Description	Communication link is dropped.		
Possible Cause	The transmission communication link is dropped.		

Solution	Reconnect a good transmission communication link.		
Alarm ID	**COR–203** **BGND-203**	**Alarm title**	**Illegal NC file format**
Description	NC program format not valid, as a result, the system cannot fully interpret the NC program.		
Possible Cause	1. The NC file is in macro format. 2. Use M98 to call a multi-path subprogram (including $1 and $2), and the size of the subprogram is greater than 60KB(60000bytes).		
Solution	1. Update the controller to 10.114.50I or the later version. 2. For M98 application, reduce the size of subprogram. Or, split it into individual single-path subprograms, then call each of them by coordinate respectively.		
Alarm ID	**COR–204** **BGND-204**	**Alarm title**	**File size too large**
Description	Program file is too large.		
Possible Cause	If the size of MACRO program is larger than 60KB(60000bytes) and use the syntax with scope statement. The syntax with scope statement contains: IF, CASE, REPEAT, FOR, WHILE		
Solution	1. Reduce the program size, or split program into two subprograms. 2. Remove all the syntax with scope statement.		
Alarm ID	**COR–205** **BGND-205**	**Alarm title**	**File content is empty**
Description	After controller loads the program, the file content found to be null.		
Possible Cause	Program loading error or CF card damaged.		
Solution	Reload program or replace CF card.		
Alarm ID	**COR–206** **BGND-206**	**Alarm title**	**Loading page lock failure**

Description	New NC program fails to required the system to distribute loading page.
Possible Cause	Lack of memory when multi-system executes large-size program.
Solution	Please upgrade to 9.240.27/ 9.242.5/10.104.36/10.111.41/10.113.5 or contact OEM Syntec.

Alarm ID	**COR–207** **BGND-207**	**Alarm title**	**Specified sequence number not found**
Description	Specified sequence number is not found.		
Possible Cause	Programming error.		
Solution	Use a sequence number within the NC program range.		

Alarm ID	**COR–208** **BGND-208**	**Alarm title**	**Cannot use jump statement in sequential file**
Description	Using jump command to execute sequential file.		
Possible Cause	Use the jump command when executing the sequence file.		
Solution	Do not use jump command to execute sequential file.		

Alarm ID	**COR–209** **BGND-209**	**Alarm title**	**【File format error】**
Description	Invalid File Format.		
Possible Cause	1. It is not a MACRO format file. For example, using APP syntax to call MARCO G200, G200 APP "appname", but this G200 file is not declared as a MACRO format file. 2. It is not a ISO format file.		
Solution	Please check the NC program, which should be written according to the file opening requirements.		

Alarm ID	COR-250 BGND-250	Alarm title	Wrong mechanism axial setting
Description	The axial setting of the mechanism is wrong.		
Possible Cause	The number of axis and the axis name required for the corresponding mechanism conversion are not set.		
Solution	Set sufficient number of axis (Pr21~), and required axis names (Pr321~).		

Alarm ID	COR-251 BGND-251	Alarm title	【The joint is not set to rotate in the right/left-hand direction】
Description	The joint is not set to the direction of rotation as the right hand, left hand rule.		
Possible Cause	Pr4141~Pr4150 are not set to the right-hand rule or the left-hand rule.		
Solution	Set the rotation direction of each joint, 0 is not allowed.		

Alarm ID	COR-252 BGND-252	Alarm title	【The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position】
Description	The 2D mechanism is converted and the target position cannot be reached. Please enter a reasonable target position.		
Possible Cause	The program coordinate position exceeds the limit of the machine.		
Solution	The position is not reasonable. Please re-check the NC program.		

Alarm ID	COR -253 BGND-253	Alarm title	【Unworkable parallel mechanism posture】
Description	Parallel mechanism posture that is not feasible.		
Possible Cause	Currently at a parallel mechanism posture that is not feasible.		

Alarm ID	COR -253 BGND-253	Alarm title	【Unworkable parallel mechanism posture】
Solution	After Reset, use the axial movement mode to leave this posture, or modify the target point.		

Alarm ID	COR-301 BGND-301	Alarm title	【OPEN command format error】
Description	OPEN command format error [command format] OPEN ("file name", "write file mode") (i) Designated as '**a**': Represent **retains** the original file content and **continuously** outputs the new data in the text file. (ii) Not specified or designated as '**w**': Represents the **emptying** of the original content and **re-outputs** the new material in the text file.		
Possible Cause	"Write mode" in the OPEN command format specifies an error. This alarm is issued if the string is specified as a string other than 'a' or 'w'.		
Solution	Set "FileWrite mode" to "a" or "w" according to the requirement of FileWrite.		

Alarm ID	COR-302 BGND-302	Alarm title	【G10 L1150 command format error】
Description	G10 L1150 command format error.		
Possible Cause	G10 L1150 parameter numbering (P argument) is a non-integer or the input is out of range or not entered. G10 L1150 parameter value (R argument) is out of range or not entered.		
Solution	Set the G10 L1150 parameter numbering and values (P and R arguments) correctly.		

Alarm ID	COR-303 BGND-303	Alarm title	【The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)】
Description	The basic spindle uses the spindle synchronization (G114.1, G51.2) commands in the position control mode.		

Alarm ID	COR-303 BGND-303	Alarm title	【The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)】
Possible Cause	When using this function, the basic spindle cannot be in position control mode.		
Solution	Write the NC program or PLC correctly so that the principal axis leaves the position control mode before the same period (C63).		

Alarm ID	COR-304 BGND-304	Alarm title	【Thread cutting geometrical axis name error】
Description	Thread cutting (G33, G34) command format is incorrect. Only the geometric axis names such as X, Y, Z, X1, Y1, Z1, etc. are supported.		
Possible Cause	When the thread cutting (G33, G34) does not have X, Y, Z, X1, Y1, Z1... and other geometric axis arguments, this alarm is issued.		
Solution	Adjust the axis names (Pr321~) of the geometrical axis to XYZ, X1/Y1/Z1...		

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Description	The mode in which the system is currently running cannot be used with the G91 incremental command.		
Possible Cause	G43.5 cannot be used with the G91 incremental command function: G43.5 determines the tool attitude through the tool vectors I, J, and K. The tool attitude is expressed only in absolute quantities.		
Solution	Check the NC program to confirm that G91 was not executed in G43.5 mode and G43.5 was not executed in G91 mode		

Alarm ID	COR-306 BGND-306	Alarm title	【Advanced look-ahead function without the specified M code】
Description	The advanced look-ahead function is enabled, and the specified M code is not found after the complete NC program is interpreted.		
Possible Cause	1. Pr3599 is not set. 2. The NC programming error.		

Alarm ID	COR-306 BGND-306	Alarm title	【Advanced look-ahead function without the specified M code】
Solution	1. Confirm whether the setting of Pr3599 is correct. 2. Check the NC program to confirm that an command has been given to speedily predecode M code (Pr3599).		

Alarm ID	COR-307 BGND-307	Alarm title	【Advanced look-ahead of M code, wrong designation of argument P】
Description	The advanced look-ahead M code (Pr3599) must specify the P argument.		
Possible Cause	When using the advanced look-ahead function, the P argument is not specified or the P argument is not an integer		
Solution	Check the NC program and confirm that there is a P argument and specify a 4-digit code subprogram.		

Alarm ID	COR-308 BGND-308	Alarm title	【Insufficient system memory, increase a single step displacement for advanced look-ahead function】
Description	The system look-ahead memory is insufficient, and the subsequent move command cannot be interpreted and is stuck.		
Possible Cause	There are too many very short blocks in the program, so the interpretation conditions cannot be met.		
Solution	Check the NC program and increase a single step displacement amount appropriately.		

Alarm ID	COR-309 BGND-309	Alarm title	【Advanced look-ahead function failed】
Description	The advanced look-ahead function cannot be executed correctly.		
Possible Cause	1. The interpolation mode is incorrect during advanced look-ahead. 2. The contents of the look-ahead NC program cannot be stored correctly.		
Solution	1. Check if the subprogram of the look-ahead NC programs contains commands other than G01, G02, and G03. 2. Check if disk space is adequate.		

Alarm ID	COR-310 BGND-310	Alarm title	【Look-ahead NC programs failed】
Description	Look-ahead NC program cannot be executed correctly.		
Possible Cause	1. The condition of the look-ahead NC program does not match: a. The version of the advanced look-ahead, the interpolation time, the number of axes, do not match. b. There is an abnormal interruption in the look-ahead, resulting in incomplete content of the look-ahead. c. To look-ahead the axis of movement in the subprogram, enter the advanced look-ahead M code in the main program. The coordinates before (i.e. M298) should be the same. (i.e. if the subprogram has X, Y, and Z three-axis movement commands, if the X, Y, and Z coordinates before look-ahead M code are modified after the look-ahead is completed, the look-ahead NC program will be executed. Issue this alarm.) 2. The contents of the look-ahead NC program cannot be read correctly.		
Solution	After advanced look-ahead under correct conditions, execute the look-ahead NC programs.		

Alarm ID	COR-311 BGND-311	Alarm title	【Hard disk storage space is insufficient to place look-ahead NC programs】
Description	The hard disk storage space is not enough to place the look-ahead NC program.		
Possible Cause	There is not enough storage space on the hard drive.		
Solution	Release hard disk storage space.		

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Description	Friction compensation adjustment setting is illegal.		
Possible Cause	1. Selected adjustment mode is not supported. 2. No axis is assigned for adjustment. 3. Radius or feedrate is not configured in Comp ON/Comp OFF mode . 4. No. of circular setting is not configured in Comp Learning mode.		

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Solution	1. Select other compensation mode and try again. 2. Please set a set of available axes. 3. Please set correct radius or feedrate. 4. Please set correct No. of circular setting. If the alarm still exists, please contact Syntec OEM.		

Alarm ID	COR-321 BGND-321	Alarm title	【The arguments (P,Q,R) retrieved from the I point position must be integers】
Description	The I value position taken argument (G10 L1010 P_ Q_ R_) is not an integer.		
Possible Cause	NC programming error.		
Solution	Modify the NC program to specify correct P, Q, and R arguments.		

Alarm ID	COR-322 BGND-322	Alarm title	【I point position capture failed】
Description	I value position capture function (G10 L1010 P_ Q_ R_), startup failed.		
Possible Cause	1. The specified P argument (axis number) does not exist. 2. The specified Q argument (I value) is not supported. 3. Exceeding the limit of the number of used groups, currently support to start three I value location captures at the same time.		
Solution	1. Modify the program to specify correct P, Q, and R arguments. 2. Turn off settings that are not in use.		

Alarm ID	COR-323 BGND-323	Alarm title	【The arguments (P,R,I,J,K) captured from the driver signal position must be integers】
Description	The argument of the drive signal position (G10 L1011 P_ R_ I_ J_ K_) is not an integer.		
Possible Cause	The NC programming error.		

Alarm ID	COR-323 BGND-323	Alarm title	【The arguments (P,R,I,J,K) captured from the driver signal position must be integers】
Solution	Modify the program to specify correct P, R, I, J, and K arguments.		

Alarm ID	COR-324 BGND-324	Alarm title	【Driver signal position captured failed to activate】
Description	The drive signal position capture function (G10 L1011 P_ R_ I_ J_ K_) failed to activate.		
Possible Cause	1. The range of R values set by the specified P and R arguments is invalid (occupied by the system or out of range). 2. The specified I argument (axis number) does not exist. 3. The specified J argument (which set of latches to use) is out of range. 4. The specified K argument (select signal source) is out of range.		
Solution	Modify the NC program to specify correct arguments.		

Alarm ID	COR-325 BGND-325	Alarm title	【In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited】
Description	In the polar coordinate interpolation (G12.1) mode, the diameter & radius axis programming switching command (G10.9) cannot be used.		
Possible Cause	In polar coordinate interpolation (G12.1) mode, if switch the diameter & radius axis program with the diameter & radius axis programming switching command (G10.9), this alarm is issued.		
Solution	In the polar coordinate interpolation (G12.1) mode, do not use the diameter & radius axis to program the switching command (G10.9). Please follow Pr281~Axis radius axis or diameter axis set value to move the command. [Remarks] For milling machine G12.1, the X axis is programmed in radius axis. For lathe machine G12.1, and the X axis programming mode can be set using Pr4020 (G12.1 X axis programming).		

Alarm ID	COR-326 BGND-326	Alarm title	【Diameter & radius axis programming command argument error】
Description	Diameter and Radius axis programming (G10.9) switching, command arguments are written incorrectly.		
Possible Cause	No axial programming is specified, or programming is specified as a value other than 0 and 1.		
Solution	1. Specify any axial direction after diameter & radius axis programming (G10.9). 2. Specify the argument value (programming mode) as 0 or 1.		

Alarm ID	COR-327 BGND-327	Alarm title	【Skip function argument input error】
Description	Skip function (G31) input argument is incorrect.		
Possible Cause	Skip function (G31) specifies both P and R arguments.		
Solution	Please modify the Skip function (G31), no specified both P and R.		

Alarm ID	COR-328 BGND-328	Alarm title	【G10 L1800 command argument number, out of specification limit】
Description	The command argument number is outside the specification limit. i.e. G10 L1800 I514 P10 R1, I argument is out of range.		
Possible Cause	The NC programming error.		
Solution	Please confirm the argument value of G10 L1800.		

P.S. Valid version of COR-328 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

Alarm ID	COR-329 BGND-329	Alarm title	【G10 L1800 command number has exceeded the single block limit】
Description	The number of G10 L1800 commands with different arguments exceeds the upper limit allowed by a single block, up to 5, or too many G10 L1800 commands.		
Possible Cause	The NC programming error.		

Alarm ID	COR-329 BGND-329	Alarm title	【G10 L1800 command number has exceeded the single block limit】
Solution	Please reduce the number of G10 L1800 commands with different arguments to less than 5 in a block, or reduce the number of G10 L1800s as a whole.		

P.S. Valid version of COR-329 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

Alarm ID	COR-330 BGND-330	Alarm title	【Illegal interrupt signal format】
Description	The command argument number is outside the specification limit. i.e. M96 P5566 I4 Q100 R1 L1000; I argument specifies the error signal source.		
Possible Cause	The NC programming error..		
Solution	Please check the NC program to ensure that the command arguments are correct.		

Alarm ID	COR-331 BGND-331	Alarm title	【This product limits T Code function】
Description	This product only supports T0~T4.		
Possible Cause	This product only supports T0~T4.		
Solution	Use a products that support the full T-code function.		

Alarm ID	COR-332 BGND-332	Alarm title	【Interrupt type subprogram (M96, M97) execution failure】
Description	When the interrupt type subprogram function is used, the execution fails when the interrupt signal is triggered. Note: Due to the pre-solved relationship when issuing this alarm, there is no guarantee that it will stop at the problem line.		
Possible Cause	Only support to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). Triggering interrupt signals in other subprograms is not supported.		

Alarm ID	COR-332 BGND-332	Alarm title	【Interrupt type subprogram (M96, M97) execution failure】
Solution	1. Make sure to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). 2. Set #1510 to 4 (the second bit is set to on) and only display the line number of the main program. This can be regarded as triggering the interrupt in the main program, but this setting will re-execute the subprogram when it returns.		

Alarm ID	COR-333 BGND-333	Alarm title	【Single end point exceeds hardware stroke limit】
Description	The coordinate position in the program exceeds the hardware stroke limit set by the machine.		
Possible Cause	1. The NC program is wrong. 2. The hardware stroke limit signal is abnormal.		
Solution	1. Check the NC program and correct the coordinate position. 2. First confirm whether it is occasionally happen during machining, and MOT-25, 26 is no longer exists after the system enters the feedhold. If so, it may be that an axis hardware stroke limit signal is triggered, but then disappears; please check the limit switch wiring or the hardware itself for abnormality.		

Alarm ID	COR-334 BGND-334	Alarm title	【G10 L1810 illegal signal condition command format】
Description	G10 L1810 command format error.		
Possible Cause	1. One of the I, Q, and R arguments is not set. 2. The I, Q, R, and J arguments are out of range, set to a negative, or a decimal value.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-334 : before 10.118.34 (included).

Alarm ID	COR-335 BGND-335	Alarm title	【G10 L1810 signal waiting condition is too many or repeat the same signal】
Description	The G10 L1810 signal waiting condition exceeds the allowable number, or the G10 L1810 repeatedly sets the same signal.		

Alarm ID	COR-335 BGND-335	Alarm title	【G10 L1810 signal waiting condition is too many or repeat the same signal】
Possible Cause	1. G10 L1810 continuously sets more than 5 commands. 2. The same G10 L1810 signal command has been set.		
Solution	Please combine multiple signals into one signal to wait, or reduce the signal waiting condition.		

Alarm ID	COR-336 BGND-336	Alarm title	【G10 L1820 illegal waiting signal condition command format】
Description	G10 L1820 command format error.		
Possible Cause	P, K arguments are out of range, set negative or decimal.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-336 : before 10.118.34 (included).

Alarm ID	COR-337 BGND-337	Alarm title	【G10 L1820 without G10 L1810 command in the front】
Description	The G10 L1820 command needs to wait for the G10 L1810 signal wait condition. At least one G10 L1810 command must be programmed first.		
Possible Cause	The G10 L1820 command was not written before the G10 L1820 command.		
Solution	Modify the NC program and write the G10 L1810 command before the G10 L1820 command.		

Alarm ID	COR-338 BGND-338	Alarm title	【G10 L1820 signal condition waiting timeout】
Description	G10 L1820 waiting signal condition exceeds the waiting time.		
Possible Cause	The signal status condition was not reached within the waiting time.		
Solution	1. Check if the G10 L1810 setting signal condition is correct. 2. Check if the PLC wait signal is processed correctly. 3. Check the hardware device to confirm why the signal status is not reached.		

Alarm ID	COR-339 BGND-339	Alarm title	【Chopping axis prohibition movement command】
Description	The chopping axial direction does not accept any movement commands.		
Possible Cause	After using the chopping function (G81.1, G81.2, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.		
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.		

Alarm ID	COR-340 BGND-340	Alarm title	【Chopping axis prohibits changing coordinate system】
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.		
Possible Cause	1. After using the chopping function (G81.1, G81.2, C86), switch the coordinate system before closing and affect the chopping axis. 2. Simultaneously use of chopping function (G81.1, G81.2, C86) and tilted work plane machining function (G68.2, G68.3). 3. Simultaneously use the chopping function (G81.1, G81.2, C86) and the axis exchange function (C133~C136). Note: C86 support version: 10.118.19 and earlier.		
Solution	1. Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. 2. Check if the NC program uses the chopping function (G81.1, G81.2,C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). 3. Check if the NC program has the chopping function (G81.1, G81.2, C86) and the shaft exchange of the chopping shaft. (C133~C136). Note 1: C86 support version: 10.118.19 and earlier. Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658). Note 3: Coordinate system related operations: external coordinate offset, MPG offset.		

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Description	The specified axis cannot be switched to the chopping axis.		

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Possible Cause	1. This axis has been designated as the PLC axis. 2. The axis has been designated as the spindle. 3. This axis has been designated as a chopping axis by other paths.		
Solution	1. Do not specify the PLC axis as the chopping axis. 2. Do not specify the spindle as the chopping axis. 3. Check if the multi-path repeats the chopping function for the same axis (G81.1, G81.2, C86). Note: C86 support version: 10.118.19 and earlier version.		

Alarm ID	COR-342 BGND-342	Alarm title	【Chopping axis prohibits non-linear kinematic transform】
Description	Non-linear kinematic transform is prohibited for the chopping axis.		
Possible Cause	1. Using non-linear kinematic transform when chopping function is enabled. 2. The machine type used is two-dimensional kinemetic (special model).		
Solution	1. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the tangential control (G41.1, G42.1). 3. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 4. The machine configuration used is the two-dimensional kinemetic (special model), and the chopping function (G81.1, G81.2, C86) cannot be used. Note: C86 support version: 10.118.19 and earlier.		

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Description	M02/M30/M99 is not used at the end of the main program.		
Possible Cause	1. Programming error. 2. NC program is damaged. 3. Some machining information was lost due to file transmission errors.		
Solution	1. Check the NC program to reset the alarm and clear it. 2. Close the Pr3853-NC program error check function.		

Alarm ID	COR-344 BGND-344	Alarm title	【The NC program under invalid paragraph】
Description	1. There is continuous blank/annotation/garbled in the NC program, and the size is more than 8KB. 2. There are no valid commands in the NC program.		
Possible Cause	1. Programmning error. 2. NC program is damaged.		
Solution	1. Avoid writing lots of consecutive blanks or annotations. 2. Please check if the NC program is damaged. After the problem is solved, reset to remove the alarm. 3. Close the Pr3853 - NC program error check function.		

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Description	Programmable data input argument format error.		

SYNTEC

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Possible Cause	1. Wobble command a. One of the I, Q, R arguments is not set. b. I, Q, R arguments out of range. c. I argument not integer. 2. G10 L1803 command a. One of the I, Q, P, R arguments is not set. b. I, Q, P, R, J, E arguments out of range. c. I, Q, R, J, K, E arguments not integer. 3. G10 L1805 、G10 L1810 command a. One of the I, Q, R arguments is not set. b. I, Q, R, J arguments out of range. c. I, Q, R, J arguments not integer. 4. G10 L1820 command a. P, K arguments out of range. b. P, K arguments not positive integer. 5. G10 L4500 command a. One of the P, Q, R arguments is not set. b. P argument is not a string. c. Q, R arguments out of range. d. Q, R arguments not integer. 6. G10 L1030 command a. P、Q、R、X、Y、Z、I、J、K、U、V arguments out of range. b. Q、X、Y、Z、I、J、K、U、V arguments not integer. 7. G10 L1031 command a. R argument is not set. b. R, V arguments out of range. c. R, V arguments not integer. 8. G10 L1500 command a. One of the P、Q arguments is not set. b. One of the P、Q arguments is out of range. c. One of the P、Q arguments is not an integer. 9. G10 L1350 command a. P, E argument is not set. b. P, E argument is out of range. c. P argument not integer.		
Solution	Please refer to command description to troubleshoot.		

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Description	Programmable data input conflicts with specifications.		

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Possible Cause	1. Between wobble command, only one block is allowed to wobble. 2. G10 L1803 commands with different arguments exceeds the upper limit allowed by a single block, up to 12. 3. Too many G10 L1803 and G10 L1805 commands. 4. The feature as L-argument specified is unsupported for current version (from 10.118.41P, 10.118.49). 5. Any axis belongs to multiple coordinates.		
Solution	1. Remove the exceeding number of wobble moving blocks. 2. Reduce the number of G10 L1803 commands in a single movement block to less than 12. 3. Reduce G10 L1803 and G10 L1805 commands. 4. Update to the version which supports the feature as L-argument specified. 5. Please set Pr701~ of the axis to only belong to a single axis group.		

Alarm ID	COR-347 BGND-347	Alarm title	【This function only supports Syntec axis】
Description	Only Syntec axis can use this function.		
Possible Cause	The weave function only supports Syntec drives.		
Solution	Please use Syntec driver.		

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error 】
Description	`There is an error in the APP command.`		
Possible Cause	1. APP name is not a string. 2. The length of APP name is over 20 characters. 3. No APP name is specified.		
Solution	Please make sure that APP commands are used correctly in the NC program.		

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Description	Limited to use with Syntec magnetic gear or ring gear encoders.		

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Possible Cause	When complex threading cycle(G78 or G78.2) enables chip removal function, the active spindle must use Syntec magnetic gear or ring gear encoders. In dual-feedback system, it is allowed to use chip removal function when either of the encoders is Syntec magnetic gear or ring gear encoder.		
Solution	1. Set the argument of D to 0. 2. Please use Syntec magnetic gear or ring gear encoders.		

Alarm ID	COR-350 BGND-350	Alarm title	【Invalid EnIP Macro】
Description	EnIP macro syntax error.		
Possible Cause	The format or range of the input macro arguments is incorrect.		
Solution	Refer to the EnIP macro syntax description.		

Alarm ID	COR-351 BGND-351	Alarm title	【Communication command abnormal】
Description	1. The execution of the macro communication command failed. 2. The macro communication to driver failed.		
Possible Cause	1. The controller does not support EnIP communication. 2. The EnIP function input request data value range is incorrect. 3. An error occurred while sending or receiving a communication command.		
Solution	1. Please contact the Syntec OEM. 2. Each request data of EnIP represents a 1 value ranging from 0 to 255. 3. This alarm is accompanied by a more detailed error alarm, refer to the alarm description for troubleshooting.		

Alarm ID	COR-352 BGND-352	Alarm title	【Communication command response data is too long】
Description	The device return data length is too long.		
Possible Cause	The storage space provided by the user for returning data is insufficient.		

Alarm ID	COR-352 BGND-352	Alarm title	【Communication command response data is too long】
Solution	Make sure that the storage space for the returned data is sufficient.		

Alarm ID	COR–353 BGND-353	Alarm title	【Invalid argument of CHKINF】
Description	The type of argument of CHKINF() is incorrect, or the category number is out of range.		
Reason	Programming error.		
Solution	1. Please check the NC program. 2. Check the type of first argument is integer. 3. Check the type of second argument is string. 4. Ensure the category number is in the range, please refer to OpenCNC_Macro Development Manual.		

Alarm ID	COR-354 BGND-354	Alarm title	【With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.】
Description	With Rotate Tool Center Point function(G43.4, G43.5) enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.		
Reason	The NC file was written incorrectly.		
Solution	Please modify the NC file, cancel Rotate Tool Center Point function(G49), and then use G10 L1150 to set the Five-Axis mechanism parameter.		

Alarm ID	COR–355 BGND-355	Alarm title	Need geometric axes positioning after tool length compensation changed
Description	Positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) is required after executing tool length compensation commands, if tool radius compensation commands needed.		
Reason	No positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) between executing tool length compensation commands and tool radius compensation commands. Tool length compensation commands: H, T(Lathe), G43, G44, G43.4, G43.5, G49, G10 L1050, G10 L1051 Tool radius compensation commands: G41, G42		

Solution	Please modify the NC file. Add positioning command of the axes on work plane(G17: XY, G18: ZX, G19: YZ) after tool length compensation commands and before tool radius compensation commands. Positioning commands: G90 G00, G90 G01, G92

Alarm ID	**COR–356** **BGND-356**	**Alarm title**	**【Invalid APP command】**
Description	There is a syntax error on the Modbus command(G10 L1900/1901/1910/1911).		
Possible Cause	The format or range of the input argument is incorrect.		
Solution	Please refer to the Industry Machine Application Manual.		

Alarm ID	**COR–357** **BGND-357**	**Alarm title**	**【Illegal Modbus packet contents】**
Description	The contents of the Modbus command packet is illegal.		
Possible Cause	The contents of the customized package is incorrect.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

Alarm ID	**COR–358** **BGND-358**	**Alarm title**	**【Modbus communication timeout】**
Description	The Modbus communication experienced a timeout.		
Possible Cause	Abnormal communication or unestablished communication causes the connection timeout.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

Alarm ID	**COR–359** **BGND-359**	**Alarm title**	**【Modbus communication failure】**
Description	The Modbus communication failed.		
Possible Cause	1. There is a communication error or device setting error.		

Alarm ID	COR–359 BGND-359	Alarm title	【Modbus communication failure】
Solution	1. Please refer to R5039, as this register provides the error code for troubleshooting		

Alarm ID	COR-361 BGND-361	Alarm title	【The number of non-moving blocks exceeds the permissible value under tool radius compensation】
Description	The machining program has programmed too many non-moving blocks when tool radius compensation is enable.		
Possible Cause	Programming error.		
Solution	1. Modify the machining program and issue the non-moving blocks when tool radius compensation is disable. 2. Modify the machining program and reduce the number of non-moving blocks when tool radius compensation is enable.		

Alarm ID	COR-362 BGND-362	Alarm 标题	【Multi-axis multi-signal skip function (G31.10, G31.11) command error】
说明	Multi-axis multi-signal skip function (G31.10, G31.11) command error		
可能原因	1. Setting command (G31.10) or execution command (G31.11) cannot be issued alone, and other commands cannot be issued between them. Always set first then execute. 2. A set of multi-axis multi-signal skip function is composed of one G31.10 command at least and one G31.11 command. Set first then execute. Both commands should be issued in a set of multi-axis multi-signal skip function, and it is not allowed to insert other commands in between. 3. Supports up to six G31.10 commands in a set of multi-axis multi-signal skip function. 4. Same axis is used in different G31.10 commands in a set of multi-axis multi-signal skip function. E.g., a. Same axis name is used in different G31.10 commands. b. Assign same axis by using virtual axis function in different G31.10 commands. (E.g., Virtual axis Z is corresponding to axis Z1 and Z2. Using axis Z in one G31.10 command, and Z1 in another G31.10 command.) c. Assign same axis by misusing alias of axis. (E.g., There are axis X, Y, Z1, Z2, Z3, Z4, the alias of Z1 is Z.)		

排除方法	1. Make sure there is always an execution command (G31.11) behind setting command (G31.10), and there is no execution command (G31.11) left alone. 2. Make sure there are six G31.10 commands at most in each set of multi-axis multi-signal skip function. 3. Make sure there is no axis assigned repeatedly in each set of multi-axis multi-signal skip function.

Alarm ID	**COR-366** **BGND-366**	**Alarm Title**	**【The parameters of rotary axis auxiliary brake are set incorrectly】**
Description	Using the rotary axis auxiliary brake, the parameters mapped to the enable group are set incorrectly.		
Possible Cause	Parameters are incorrect. Enable G10 L1500 rotary axis auxiliary brake: 1. The axis ID of the rotary axis auxiliary brake is not specified. 2. The axis type of the specified axis is linear. 3. The M code of the rotary axis auxiliary brake is not specified.		
Solution	Modify parameters Pr3741~Pr3744.		

Alarm ID	**COR-367** **BGND-367**	**Alarm Title**	**【Prohibited commands in rotary axis auxiliary brake mode】**
Description	When the rotary axis auxiliary brake is enabled, prohibited commands are used in the nc program.		
Possible Cause	1. The axis, which is in rotary axis auxiliary brake mode, is specified by skip function(G28.1, G31, G31.11). 2. The axis, which is in rotary axis auxiliary brake mode, is specified by chopping function(G81.1, G81.2). 3. Enable the rotary axis auxiliary brake in RTCP mode(G43.4, G43.5). 4. Enable both the rotary axis auxiliary brake mode and the polar coordinate mode(G12.1).		
Solution	Check the NC program.		

Alarm ID	**COR-370** **BGND-370**	**Alarm title**	**【Failed to enable chopping function】**
Description	Failed to enable the chopping function .		

Alarm ID	COR-370 BGND-370	Alarm title	【Failed to enable chopping function】
Possible Cause	1. G81.2 or G81.1 command repeatedly a. G81.1 command repeatedly. b. G81.2 and G81.1 command simultaneously. 2. Using G81.2 to make 2 axis in the same axis group do advanced chopping simultaneously.		
Solution	Please activate or deactivate the chopping function correctly.		

Alarm ID	COR–401 BGND-401	Alarm title	Path planning plug-in error
Description	Path planning plug-in error.		
Reason	Path planning plug-in error leads to system abnormality.		
Solution	Please contact OEM Syntec.		

Alarm ID	BGND-501	Alarm title	Register not enough, program execution failed
Description	Insufficient controller remaining registers cause the background execution unit to fail to activate.		
Possible Cause	1. Use too many background execution units in ladder. 2. The system remaining register is too low.		
Solution	Contact machinery manufacturer.		

Alarm ID	BGND-502	Alarm title	Invalid program command
Description	Incorrect G code is used in program or use M, T code to call MACRO.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program to confirm that the unsupported G/M/T codes are not used.		

Alarm ID	BGND-503	Alarm title	Illegal G code argument
Description	Enter incorrect G10 argument format or range in the program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and refer to manual to input correct argument.		
Alarm ID	BGND-504	Alarm title	S code not supported in background execute mode
Description	S code command is used in program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm S code is not used.		
Alarm ID	BGND-505	Alarm title	F code not supported in background execute mode
Description	F code command is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program and confirm F code is not used.		
Alarm ID	BGND-506	Alarm title	H code not supported in background execute mode
Description	H code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm H codes is not used.		

Alarm ID	**BGND-507**	**Alarm title**	**D code not supported in background execute mode**
Description	D ode is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm D codes is not used.		
Alarm ID	**BGND-508**	**Alarm title**	**T code not supported in background execute mode**
Description	T code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm T codes is not used.		
Alarm ID	**BGND-509**	**Alarm title**	**M code not supported in background execute mode**
Description	M code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm M codes is not used.		

Alarm ID	**BGND-510**	**Alarm title**	**【Quantities of Background execute unit exceeds capacity】**
Description	The number of enabled BGND components exceeds the upper limit (20), and can not enable more BGND components.		
Possible Cause	Use too many background execute unit in ladder.		

Solution	Contact machinery manufacturer.

8.1 BGND-001 ~ BGND-499

Alarm ID	**BGND-001 ~ BGND-499**	**Alarm title**	**Titles are the same as COR alarm with corresponding ID**
Description	Please refer to COR alarm with corresponding ID		
Possible Cause	Please refer to COR alarm with corresponding ID		
Solution	Please refer to COR alarm with corresponding ID		

8.2 BGND-001 Array Variable is empty

Alarm ID	**COR–001 BGND-001**	**Alarm title**	**Array Variable is empty**
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

8.3 BGND-002 File not exist

Alarm ID	**COR–002 BGND-002**	**Alarm title**	**File not exist**
Description	If the file that the system wants to read does not exist i.e. use M98 (or G65, G66, and etc.) to call a non existing file.		
Reason	Programming error.		

Solution	Check the NC program to make sure the existence of the file.

8.4 BGND-003 Devide by zero error

Alarm ID	**COR–003** **BGND-003**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of MACRO is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		
Reason	Programming error		
Solution	Check the NC program to ensure that the denominator is not equal to 0.		

8.5 BGND-004 Operation domain error

Alarm ID	**COR–004** **BGND-004**	**Alarm title**	**Operation domain error**
Description	Operation domain error.		
Reason	Programming error		
Solution	Please check the NC program.		

8.6 BGND-005 Program loading failure

Alarm ID	**COR–005** **BGND-005**	**Alarm title**	**Program loading failure**
Description	MACRO syntax error.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.7 BGND-006 Arc not on work plane

Alarm ID	COR–006 BGND-006	Alarm title	Arc not on work plane
Description	1. In clockwise, counter-clockwise arc cutting (G02, G03) syntax, if the vector from center of circle to starting point doesn't exist on the arc working surface. i.e. execute G17 G02 I50. K10. , and system will issue this alarm. 2. In spiral interpolation (G02, G03) syntax, when corresponding to G17, G18, and G19 three surface conditions, if K, J , and I arguments are not zero, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that G02 and G03 are used correctly.		

8.8 BGND-007 Arc radius too short

Alarm ID	COR–007 BGND-007	Alarm title	Arc radius too short
Description	In G02 and G03 syntax, if Arc radius is smaller than 10 to the power of minus 10 BLU (10^-10), system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.		

8.9 BGND-008 Arc destination not on arc

Alarm ID	COR–008 BGND-008	Alarm title	Arc destination not on arc

Description	In G02 and G03 syntax, if the Arc end point coordinate is not on the circle, system will issue this alarm. From 8.31 version, adding Pr3807 setting arc final point is not on the check window, allowing to set window range's error in Pr3807. When arc end point position's error is smaller than Pr3807 setting window's range, system will auto correct the circle's center position, and make the end point position to locate on the circle correctly. When arc end point position error is bigger than Pr3807 window range setting, system will issue this alarm.
Reason	Programming error.
Solution	Check the NC program to ensure that the Arc radius of G02 and G03 are used correctly.

8.10 BGND-009 G65 layers called by Macro too deep

Alarm ID	**COR–009** **BGND-009**	**Alarm title**	**G65 layers called by Macro too deep**
Description	Use G65 to call MACRO program layer more than 12 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 12 layers.		

8.11 BGND-010 G66 layers called by Macro too deep

Alarm ID	**COR–010** **BGND-010**	**Alarm title**	**G66 layers called by Macro too deep**
Description	Use G66 to call MACRO program layer more than 4 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that G65 calls MACRO program less than or equal to 4 layers.		

8.12 BGND-011 Subprogram layer call too deep

Alarm ID	COR–011 BGND-011	Alarm title	Subprogram call too deep
Description	Use M98 to call subprogram that has more than 16 layers.		
Reason	Programming error.		
Solution	Check NC program to ensure that M98 calls subprogram that has less than 16 layers.		

8.13 BGND-012 G66 mode not cancel by G67

Alarm ID	COR–012 BGND-012	Alarm title	G66 mode not cancel by G67
Description	G66 and G67 need to be used in pairs, when numbers of G67 is more than G66 in one NC program, this alarm will be issued.		
Reason	Programming error.		
Solution	Check NC program to ensure that G66 and G67 are used in pairs		

8.14 BGND-013 G65, G66 must be at the end of the block

Alarm ID	COR–013 BGND-013	Alarm title	G65, G66 must be at the end of the block
Description	G65 and G66 are MACRO, so in one same block the program on the right hand side of G65 and G66 will be processed as G65 and G66's arguments. So in one same block, if there is other G code command please write them in the left hand side of G65 and G66. In one same block, if the right hand side of G65 and G66 has G code or M code, system will issue this alarm.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.15 BGND-014 Absent program number

Alarm ID	COR–014 BGND-014	Alarm title	Absent program number
Description	The right hand side of G65 and G66 doesn't have P argument to specify program number.		
Reason	Programming error.		
Solution	Please check the NC program to ensure G65 and G66 use P argument to specify program number.		

8.16 BGND-015 Too many M code

Alarm ID	COR–015 BGND-015	Alarm title	Too many M code
Description	There are more than 3 M codes in a single block.		
Reason	Programming error.		
Solution	Please check the NC program to ensure that there are equal or less than 3 M codes in a single block		

8.17 BGND-016 Illegal variable access

Alarm ID	COR–016 BGND-016	Alarm title	Illegal variable access
Description	Attempted to access a variable that does not exist.		
Reason	1. Programming error. 2. Illegal access to # or @ variable. 3. The parameter of SYSDATA function is out of system data range. 4. The controller does not support the status variable which DRVDATA is to read. 5. The program that attempts to access AR or MAR variables does NOT lie within the scope of App Macros. 6. The AR or MAR variables to access do NOT lie within the range defined by the App. 7. Non-integer numbers are used to specify the addresses of AR or MAR variables. ONLY integers are acceptable.		

Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable. 4. Make sure the variable which DRVDATA is to read can be found on the "Controller Axis Info." page. 5. Make sure the AR and the MAR variables are accessed ONLY in App Macros. 6. Make sure the AR and the MAR variables to access lie within the range defined by the App. 7. Make sure only integers are used to specify the addresses of AR or MAR variables.

8.18 BGND-017 Label not found

Alarm ID	**COR–017** **BGND-017**	**Alarm title**	**Label not found**
Description	Cannot find out corresponding serial number N in GOTO command.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.19 BGND-018 Line number not found

Alarm ID	**COR–018** **BGND-018**	**Alarm title**	**Line number not found**
Description	Input line number is incorrect.		
Reason	1. When Pr3851 sets as 888800, input breakpoint's line number exceeds NC program's maximum line number. 2. When Pr3851 sets as 999900, input breakpoint's line number exceeds NC program's maximum line number. 3. When Pr3851 sets as 999901, input breakpoint's line number exceeds NC program's maximum line number or positioning command line number cannot be found after scanning through entire NC program. 4. When M99 Q_ sub-program returns to the line number which is assigned by main program, the line number Q_ assigned is over main program's maximum line number.		
Solution	1. Input the correct program line number. 2. When Pr3851 sets as 999901, modify NC program by inserting positioning command after initial assigned breakpoint.		

8.20 BGND-019 sub-program has no M99

Alarm ID	COR–019 BGND-019	Alarm title	sub-program has no M99
Description	When main program calls sub-procedure, if sub-program finish executing and need to return the main program, there is no M99.		
Reason	Programming error.		
Solution	Write in M99 when sub-program finish executing and return the main program.		

8.21 BGND-020 Too many G code

Alarm ID	COR–020 BGND-020	Alarm title	Too many G code
Description	There are more than 10 G codes in a single block.		
Reason	Programming error.		
Solution	Breakdown that single block which has over 10 G codes into single blocks that has less than 10 G codes.		

8.22 BGND-021 Too many (I,J,K) parameters

Alarm ID	COR–021 BGND-021	Alarm title	Too many (I,J,K) triples
Description	Repeat too much I, J, and K command in the same block.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.23 BGND-022 Use undefined G54 workpiece coordinate

Alarm ID	**COR–022** **BGND-022**	**Alarm title**	**Use undefined workpiece coordinate**
Description	Use an undefined G54 coordinate system.		
Reason	Programming error.		
Solution	Use the correct G54 coordinate system.		

8.24 BGND-023 Semantic error

Alarm ID	**COR-023** **BGND-023**	**Alarm Title**	**Semantic error**
Description	1. When using G code, different G code can input different argument (P_, L_, R_, and etc.), if argument setting is wrong, and system will issue this alarm. 2. The argument of macro function is wrong.		
Reason	Programming error.		
Solution	Check the NC program to ensure the argument is used correctly by referring the manual.		

8.25 BGND-024 Invalid arc radius value

Alarm ID	**COR–024** **BGND-024**	**Alarm title**	**Invalid arc radius value**
Description	When executing G02, G03, appointed Arc end point and given radius is contradicted, which the given radius cannot meet appointing Arc end point. i.e. G91 G03 X1500 Y4000 R2000.		
Reason	Programming error.		
Solution	Check the program and recalculate.		

8.26 BGND-025 Macro stack is overflow or STKTOP[ ] argument error

Alarm ID	**COR–025** **BGND-025**	**Alarm title**	**Macro stack is overflow or STKTOP[ ] argument error**
Description	1. STACK can store maximum 4095 values. If stored value exceeding the maximum number, controller issue this alarm. 2. In STKTOP[n], n is started from 0, if the value of n is bigger than the value storing in stack-1, controller issue this alarm		
Reason	1. Too much value store in STACK. 2. STKTOP[] arguments exceeds the value storing in STACK.		
Solution	1.Stack is full, do not use store command anymore. 2. Input a reasonable argument in STKTOP[].		

8.27 BGND-026 Macro stack is empty

Alarm ID	**COR–026** **BGND-026**	**Alarm title**	**macro stack is empty**
Description	Empty stack still want to pop value, system issue this alarm.		
Reason	The numbers of Push commands and Pop commands are not the same.		
Solution	Check the program to ensure that the number of Push commands is the same with that of Pop commands.		

8.28 BGND-027 Invalid macro arguments

Alarm ID	**COR–027** **BGND-027**	**Alarm title**	**Invalid macro arguments**
Description	Macro issue self-defined alarm.		
Possible Cause	Once Macro determines the self-defined alarm error condition is meet, NC program will be stopped and alarm will appear.		
Solution	According to the display content of alarm to find out the error.		

8.29 BGND-028 System program error, can’t normally machining.

Alarm ID	**COR–028** **BGND-028**	**Alarm title**	**System program error, can’t normally machining.**
Description	Use Quiet Mode in MACRO, but can't leave Quiet Mode when program finished.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.30 BGND-029 Tool length offset change at arc

Alarm ID	**COR–029** **BGND-029**	**Alarm title**	**Tool length offset change at arc**
Description	G43, G44, G49 only receive linear interpolation command in the next block.		
Possible Cause	Programming error.		
Solution	Please check the NC program.		

8.31 BGND-030 Cutting speed command is 0

Alarm ID	**COR-030** **BGND-030**	**Alarm Title**	**Cutting speed command is 0**
Description	When execute cutting command, given F code argument is zero.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the argument of F code shall not be 0.		

8.32 BGND-031 Radius compensation cancel at arc

Alarm ID	**COR–031** **BGND-031**	**Alarm title**	**Radius compensation cancel at arc**

Description	G40 only receives linear interpolation command in the next block
Possible Cause	Programming error.
Solution	Please check the NC program.

8.33 BGND-032 Radius compensation activate at arc

Alarm ID	**COR–032** **BGND-032**	**Alarm title**	**Radius compensation activate at arc**
Description	G41,G42 only receive linear interpolation command in the next block.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.34 BGND-033 Improper use of A, R, or C command

Alarm ID	**COR–033** **BGND-033**	**Alarm title**	**Improper use of A, R, or C command**
Description	Timing of using A, R, or C command is incompatible with the specification.		
Reason	Programming error.		
Solution	Check NC program to confirm that if the blocks are compatible with specifications		

8.35 BGND-034 Command syntax has conflict

Alarm ID	**COR–034** **BGND-034**	**Alarm title**	**Command syntax has conflict**
Description	No application currently.		
Reason			

Alarm ID	COR–034 BGND-034	Alarm title	Command syntax has conflict
Solution			

8.36 BGND-035 Corner is too small can’t insert round corner chamfer

Alarm ID	COR–035 BGND-035	Alarm title	Corner is too small can’t insert round corner chamfer
Description	Angel between blocks, which to be inserted round corner or chamfer is too small so system can’t calculate it.		
Reason	Programming error.		
Solution	1. Check NC program to confirm that whether the block is compatible with specifications. 2. Check and confirm the working plane and round corner plane are the same.		

8.37 BGND-036 Inappropriate A angle command

Alarm ID	COR–036 BGND-036	Alarm title	Inappropriate A angle command
Description	A is valid only in linear interpolation of single block.		
Reason	Programming error.		
Solution	Check NC program to confirm that whether A angle command is compatible with specifications		

8.38 BGND-037 Chamfer value bigger than displacement

Alarm ID	COR–037 BGND-037	Alarm title	Chamfer value bigger than displacement
Description	Chamfer value is bigger than pre-block and post-block length, so system can’t calculate it.		

Reason	Programming errors.
Solution	1. Check NC program to confirm that whether inserted value of chamfer is compatible with specifications. 2. Check to confirm whether C value is compatible with specification.

8.39 BGND-038 Incorrect block jump's switch number.

Alarm ID	**COR–038** **BGND-038**	**Alarm title**	**Incorrect block jump's switch number.**
Description	Incorrect block jump switch number. i.e. / 2 G00 X100. ; If assigned number is bigger than 9 or equals to 0, system will issue this alarm.		
Reason	Block jump switch number is 0 or bigger than 9.		
Solution	Check the NC program, and confirm block jump switch number is between 1 and 9.		

8.40 BGND-039 Measure function can't start tool radius compensation

Alarm ID	**COR–039** **BGND-039**	**Alarm title**	**Measure function can't start tool radius compensation**
Description	Measure function can't start tool radius compensation.		
Reason	Measure function related instruction (i.e., G31, G31.10, G31.11) is executed after tool compensation command.		
Solution	Confirm there is no measure function related command is executed after tool compensation command.		

8.41 BGND-040 Block end point exceed software stroke limit

Alarm ID	**COR–040** **BGND-040**	**Alarm title**	**Block end point exceed software stroke limit**
Description	The machine coordinate in the program exceeds the software stroke limit.		

Reason	Programing error.
Solution	Check the NC program, and correct coordinate position.

8.42 BGND-041 GOTO label must be integer

Alarm ID	**COR–041** **BGND-041**	**Alarm title**	**GOTO label must be integer**
Description	The input GOTO label is not an integer. i.e. GOTO 1 Correct GOTO 1. Wrong N1; Correct N1.; Wrong		
Reason	Programming error.		
Solution	Check the NC program, and input integer in GOTO argument.		

8.43 BGND-042 Logic operand is non-integer or empty

Alarm ID	**COR–042** **BGND-042**	**Alarm title**	**Logic operand is non-integer or empty**
Description	Logic operand must be non-integer or empty.		
Reason	Logic operand has floating point. i.e. #1=1.5And3. System will issue this alarm.		
Solution	Please check NC program. Confirm logic operand is integer or empty.		

8.44 BGND-043 ASIN, ACOS operand must between ±1.0

Alarm ID	**COR–043** **BGND-043**	**Alarm title**	**ASIN, ACOS operand must between ±1.0**

Description	ASIN() and ACOS() operand is not between -1.0 and 1.0.
Reason	Programming error.
Solution	Check the NC program.

8.45 BGND-044 SQRT operand should not be negative

Alarm ID	**COR–044** **BGND-044**	**Alarm title**	**SQRT operand should not be negative**
Description	The square root of a negative value will be imaginary number, but the controller does not provide imaginary number function.		
Reason	Programming error.		
Solution	Check the NC program; enter a positive value in SQRT operand.		

8.46 BGND-045 L address should be integer

Alarm ID	**COR–045** **BGND-045**	**Alarm title**	**L address should be integer**
Description	The L address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in L address.		

8.47 BGND-046 O address should be integer

Alarm ID	**COR–046** **BGND-046**	**Alarm title**	**O address should be integer**
Description	The O address is not an integer.		
Reason	Programming error.		

Solution	Check the NC program, and use integer in O address.

8.48 BGND-047 M address should be integer

Alarm ID	**COR–047** **BGND-047**	**Alarm title**	**M address should be integer**
Description	The M address is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in M address.		

8.49 BGND-048 Spindle speed S should be integer

Alarm ID	**COR–048** **BGND-048**	**Alarm title**	**Spindle speed S should be integer**
Description	The spindle speed S is not an integer.		
Reason	Programming error.		
Solution	Check the NC program, and use integer in the spindle speed S.		

8.50 BGND-049 Tool length compensation H should be integer

Alarm ID	**COR–049** **BGND-049**	**Alarm title**	**Tool length compensation H should be integer**
Description	Tool length compensation H is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool length compensation H.		

8.51 BGND-050 Tool radius compensation D should be integer

Alarm ID	COR–050 BGND-050	Alarm title	Tool radius compensation D should be integer
Description	Tool radius compensation D is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use integer in tool radius compensation.		

8.52 BGND-051 Tool number T should be integer

Alarm ID	COR–051 BGND-051	Alarm title	Tool number T should be integer
Description	Tool number T is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use tool number T in integer.		

8.53 BGND-052 Sub-program number P should be integer

Alarm ID	COR–052 BGND-052	Alarm title	Sub-program number, P, should be integer
Description	Sub-program number P is not an integer.		
Reason	Programming error.		
Solution	Please check the NC program, and use the sub-program number P in integer.		

8.54 BGND-053 Repeat count L should be integer

Alarm ID	COR–053 BGND-053	Alarm title	Repeat count L should be integer

Description	Repeat count L is not an integer.
Reason	Programming error.
Solution	Please check the NC program, and use the repeat count L in integer.

8.55 BGND-054 Incompatible data type

Alarm ID	**COR–054** **BGND-054**	**Alarm title**	**Incompatible data type**
Description	Data format is incompatible with controller specifications.		
Reason	NC program is not compatible with the SYNTEC controller.		
Solution	Make sure that the NC program data format is compatible with Syntec controller specifications.		

8.56 BGND-055 Tool length compensation H out of range

Alarm ID	**COR–055** **BGND-055**	**Alarm title**	**Tool length compensation H out of range**
Description	The tool length compensation H exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool length compensation H, is in the range of tool number.		

8.57 BGND-056 G10 table index P is out of range

Alarm ID	**COR–056** **BGND-056**	**Alarm title**	**G10 table index P is out of range**
Description	G10's format is G10 L_ P_ R_; Different number L will correspond to different number P. i.e. L10 corresponding number P is tool number. Input P1000 means the 1000th tool. If that tool number doesn't exist, and controller will issue this alarm.		

Reason	Programming error.
Solution	Confirm G10 data table address number P is in reasonable range.

8.58 BGND-057 Tool radius D out of range

Alarm ID	**COR–057** **BGND-057**	**Alarm title**	**Tool radius D out of range**
Description	The tool radius number D exceeds the range of tool number.		
Reason	Programming error.		
Solution	Make sure that the tool radius number D is in the range of tool number.		

8.59 BGND-058 Tool nose compensation D is out of range

Alarm ID	**COR–058** **BGND-058**	**Alarm title**	**Tool nose compensation D is out of range**
Description	The tool nose compensation D exceeds the range of tool number.		
Reason	Program error.		
Solution	Make sure that the tool radius compensation D is in the range of tool number.		

8.60 BGND-059 Subprogram call H must be integer

Alarm ID	**COR–059** **BGND-059**	**Alarm title**	**Subprogram Call H must be integer**
Description	Subprogram call H is not an integer.		
Reason	Program error.		
Solution	Confirm the subprogram call H is an integer.		

8.61 BGND-060 M99 return number P must integer

Alarm ID	COR–060 BGND-060	Alarm title	M99 return number P must integer
Description	The M99 return sequence number P is not an integer.		
Reason	Program error.		
Solution	Confirm the M99 return sequence number P is an integer.		

8.62 BGND-061 Workpiece number is out of range

Alarm ID	COR–061 BGND-061	Alarm title	Workpiece number is out of range
Description	The number of work pieces exceeds the allowable range of the controller.		
Reason	Program error.		
Solution	Make sure the number of work pieces is in the allowed range of the controller.		

8.63 BGND-062 Dwell skip source Q must be integer

Alarm ID	COR–062 BGND-062	Alarm title	Dwell skip source Q must be integer
Description	Dwell skip source Q is not an integer.		
Reason	Program error.		
Solution	Change the dwell skip source Q into an integer.		

8.64 BGND-063 Dwell skip source Q out of range

Alarm ID	COR–063 BGND-063	Alarm title	Dwell skip source Q out of range

Description	Dwell skip source Q exceeds the allowed range.
Reason	Program error.
Solution	Make sure the dwell skip source Q is in the allowed range.

8.65 BGND-064 Invalid P address

Alarm ID	**COR–064** **BGND-064**	**Alarm title**	**Invalid P address**
Description	Invalid P address.		
Reason	1. Address P is less than 0. 2. Address P is not integer. 3. Address P is out of range.		
Solution	1. Make sure address P is greater or equal to 0. 2. Make sure address P is integer. 3. Make sure address P is within the range.		

8.66 BGND-065 A command format error

Alarm ID	**COR–065** **BGND-065**	**Alarm title**	**,A command format error**
Description	In NC program ,A command's format is wrong.		
Reason	Programming error.		
Solution	Please confirm ,A command format is one of the followings: 1. In the same block command as ,A, Moving instruction only includes one of x axis or z axis. i.e. G01 X10. ,A30. 2. Use ,A advanced function in two separate line. The first line will assign ,A angle, and the second line will assign X, Z and ,A at the same time. i.e. G01 ,A30. G01 X0. Z50. ,A45		

8.67 BGND-066 Inc. axis command and abs. axis command conflict

Alarm ID	COR–066 BGND-066	Alarm title	Inc. axis command and abs. axis command conflict
Description	Both G91 and G90 are in the same line.		
Reason	Programming error.		
Solution	Decide to use incremental or absolute command in one line, and enter the correct command.		

8.68 BGND-067 Arc center vector and radius conflict

Alarm ID	COR–067 BGND-067	Alarm title	Arc center vector and radius conflict
Description	The arc end point is not on the arc created by the arc starting point and the specify center.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.69 BGND-068 Quiet Mode not support imperial Metric switch command

Alarm ID	COR–068 BGND-068	Alarm title	Quiet Mode not support imperial Metric switch command
Description	The single block command in Quiet mode operation cannot switch Metric/ imperial unit.		
Reason	Programming error.		
Solution	Please check the NC program.		

8.70 BGND-069 Round corner and chamfer cmd. conflict

Alarm ID	COR–069 BGND-069	Alarm title	Round corner and chamfer cmd. conflict
Description	Chamfer command and round corner command are in the same line.		
Reason	Program error.		
Solution	Do not let chamfer command and round corner command exist in the same line.		

8.71 BGND-070 Invalid G Code

Alarm ID	COR–070 BGND-070	Alarm title	Invalid G Code
Description	Enter incorrect G code to controller.		
Reason	Program error.		
Solution	Enter the valid G-code.		

8.72 BGND-071 No main program name assigned

Alarm ID	COR–071 BGND-071	Alarm title	No main program name assigned
Description	The name of main program is not specified.		
Reason	The NC program is not loaded.		
Solution	Specify the name of main program.		

8.73 BGND-072 Threading command exceeds max. cutting speed

Alarm ID	COR–072 BGND-072	Alarm title	Threading command exceeds max. cutting speed

Description	Threading command exceeds the maximum cutting speed.
Reason	Program error.
Solution	Decrease the cutting speed of threading.

8.74 BGND-073 Tapping command exceed max. cutting speed

Alarm ID	**COR–073** **BGND-073**	**Alarm title**	**Tapping command exceed max. cutting speed**
Description	Tapping command speed exceeds the maximum cutting speed.		
Reason	Program error.		
Solution	Decrease the cutting speed of tapping.		

8.75 BGND-074 Tool radius too big, path overcut

Alarm ID	**COR–074** **BGND-074**	**Alarm title**	**Tool radius too big, path overcut**
Description	1. During machining notch, the notch width is smaller than two times of tool radius. 2. During machining trapezoidal, the height of trapezoidal is less than two times of tool radius.		
Reason	Tool compensation causes path overcut.		
Solution	Please check the NC program, and decide whether this part of the machining should cancels tool radius compensation.		

8.76 BGND-075 Exact stop wait timeout

Alarm ID	**COR–075** **BGND-075**	**Alarm title**	**Exact stop wait timeout**

Description	After 2 seconds after sending Exact stop (G09/G61) command, the difference between feedback and command exceeds allowable value (G01 is determined by Pr421~; G00 is determined by Pr481~).
Reason	Servo vibration.
Solution	1. Servo tuning. 2. Modify parameters Pr421~ or Pr481~.

8.77 BGND-076 G04 dwell time cannot be negative

Alarm ID	**COR–076** **BGND-076**	**Alarm title**	**G04 dwell time cannot be negative**
Description	Input value of dwell time G04 is negative.		
Reason	Program error.		
Solution	Check the NC program, and enter a positive or zero value in G04 argument.		

8.78 BGND-077 NURBS curve format is wrong

Alarm ID	**COR-077** **BGND-077**	**Alarm Title**	**NURBS curve format is wrong**
Description	G6.2 format is wrong.		
Reason	Input argument P, K, R is against allowable format.		
Solution	Refer the program manual to correct the command syntax.		

8.79 BGND-078 NURBS curve system memory insufficient

Alarm ID	**COR-078** **BGND-078**	**Alarm Title**	**NURBS curve system memory insufficient**
Description	NURBS curve memory is insufficient.		

Alarm ID	COR-078 BGND-078	Alarm Title	NURBS curve system memory insufficient
Reason	System processing defective.		
Solution	Contact OEM Syntec.		

8.80 BGND-079 Number of NURBS curve nodes more than the upper limit

Alarm ID	COR-079 BGND-079	Alarm Title	Number of Nurbs curve nodes more than the upper limit
Description	NURBS curve control point is over limit.		
Reason	NC file G6.2 curve is too long and cause using too many control point.		
Solution	Re-generate the CAM, and restrict the number of curve nodes less than 400.		

8.81 BGND-080 Threading pitch is negative

Alarm ID	COR-080 BGND-080	Alarm Title	Threading pitch is negative
Description	Threading pitch distance can’t be negative value.		
Reason	Threading pitch distance setting error.		
Solution	Check and correct the configured threading pitch to be positive value.		

8.82 BGND-081 Use arc interpolation under tool offset funct

Alarm ID	COR-081 BGND-081	Alarm Title	Use arc interpolation under tool offset function. Arc command doesn’t use assigned angle.
Description	In tool offset function (G45~G48), G02 and G03 can only be 90 degree or 270 degree.		

Alarm ID	COR-081 BGND-081	Alarm Title	Use arc interpolation under tool offset function. Arc command doesn't use assigned angle.
Reason	In NC program's tool offset function (G45~G48) block, all arc interpolation angle can only be 90 degree or 270 degree.		
Solution	Check if G45~G48 is 90 or 270 degree when using them in the arc interpolation (G02/G03). Note, IJK to specify circle center shall be used.		

8.83 BGND-082 Tool compensation function and tool offset function can't use at the same time

Alarm ID	COR-082 BGND-082	Alarm Title	Tool compensation function and tool offset function can't use at the same time
Description	Can't use tool offset function (G45~G47) and tool compensation function (G41, G42) or tool radius compensation at the same time.		
Reason	NC program edit error.		
Solution	Check NC program, and confirm tool offset function (G45~G47) and tool compensation function (G41, G42) are not used at the same time.		

8.84 BGND-083 When pull tool back to pause point, and move axis position manually.

Alarm ID	COR-083 BGND-083	Alarm Title	When pull tool back to pause point, and move axis position manually.
Description	Before pull tool back to pause point, press stop and manually move tool.		
Reason	Operation error.		
Solution	Please wait until tool is pulled exactly back to pause point, then execute the manual position movement.		

8.85 BGND-084 P Argument unassigned when using the rapid drilling

Alarm ID	COR-084 BGND-084	Alarm Title	P Argument unassigned when using the rapid drilling
Description	When using rapid drilling function, must set P argument (bottom hole dwell time), or system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure the P Argument is assigned when using the G code of rapid drilling.		

8.86 BGND-085 F Argument unassigned in Inverse Time Feed mode

Alarm ID	COR-085 BGND-085	Alarm Title	F Argument unassigned in Inverse Time Feed mode
Description	Under Inverse Time Feed mode, if F argument is not assigned in command line and system will issue this alarm.		
Reason	Programming error.		
Solution	Check the NC program to ensure there is F command is given following in each line after Inverse Time Feed mode.		

8.87 BGND-086 R and I arguments unassigned in G10 L1501 mode

Alarm ID	COR-086 BGND-086	Alarm Title	R and I arguments unassigned in G10 L1501 mode
Description	G10 L1501 argument only has argument R or argument I.		
Reason	Programming error.		
Solution	Check the NC program to ensure the R and I arguments are not used independently.		

8.88 BGND-087 P and Q arguments must be different in G10 L1501 mode

Alarm ID	COR-087 BGND-087	Alarm Title	P and Q arguments must be different in G10 L1501 mode
Description	P and Q argument in G10 L1501 mode setting are the same.		
Reason	Programming error.		
Solution	Please check program, and confirm P and Q argument in G10 L1501 mode aren’t the same.		

8.89 BGND-088 No time-axis command after the I argument in G10 L1501 mode

Alarm ID	COR-088 BGND-088	Alarm Title	No time-axis command after the I argument in G10 L1501 mode
Description	In spring machine dedicated machine's motion plan, if use I argument in G10 L1501, before the next G10 L1502 command, user can’t command P argument axis number (time axis) in G1 0L1501.		
Reason	Programming error.		
Solution	Check the NC program.		

8.90 BGND-089 Main axis must be increment command in G10 L1501 mode

Alarm ID	COR-089 BGND-089	Alarm Title	Main axis must be increment command in G10L1501 mode
Description	G10 L1501 mode spindle must be incremental command.		
Reason	Programming error.		
Solution	Under G10 L1501 mode, confirm the command which sends to spindle is incremental command.		

8.91 BGND-090 C-axis zero-speed check failed in G12.1 mode

Alarm ID	COR-090 BGND-090	Alarm Title	C-axis zero-speed check failed in G12.1 mode
Description	When start to switch polar coordinate, C axis must be totally stop.		
Reason	When C axis uses as spindle, but spindle not yet stop rotating.		
Solution	Switch the spindle to C-axis mode before activating polar coordinate.		

8.92 BGND-091 Main axis synchronization – basic main axis number error

Alarm ID	COR-091 BGND-091	Alarm Title	Main axis synchronization – basic main axis number error
Description	When using spindle synchronous function G114.1, Pr4021, Pr4023, Pr4025 can't be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4021/Pr4023/Pr4025 are correct.		

8.93 BGND-092 Main axis synchronization – synchronization main axis number error

Alarm ID	COR-092 BGND-092	Alarm Title	Main axis synchronization – synchronization main axis number error
Description	When using spindle synchronous function G114.1, Pr4022, Pr4024, Pr4026 can't be zero.		
Reason	Parameter setting error.		
Solution	Check if values of Pr4022/Pr4024/Pr4026 are correct.		

8.94 BGND-093 Main axis synchronization – spindle type error

Alarm ID	COR-093 BGND-093	Alarm Title	Main axis synchronization – spindle type error
Description	When using spindle synchronous and Superimposition function, setting wrong spindle motor type.		
Reason	Parameter setting error.		
Solution	Check 1791~Pr1800 to set correct spindle configurations by referring to HELP.		

8.95 BGND-094 Under spindle load rigid tapping spindle rotation speed exceed

Alarm ID	COR-094 BGND-094	Alarm Title	Under spindle load rigid tapping spindle rotation speed exceed
Description	When using spindle load function to do rigid tapping, tool axis must synchronize with offset axis speed, and then do end surface taping according to user setting tapping speed. If tool axis speed is over spindle rotation speed maximum value, and system will issue this alarm.		
Reason	Command is over range that mechanical can bear.		
Solution	1. Reduce the rotation speed (tapping speed) of tool axis. 2. Reduce the rotation speed of workpiece axis.		

8.96 BGND-095 In polygon cutting function, basic spindle rotation speed ratio is wrong

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (P argument) must be bigger than zero.		
Reason	Programming error.		

Alarm ID	COR-095 BGND-095	Alarm Title	In polygon cutting function, basic spindle rotation speed ratio is wrong
Solution	Check the value of P argument in G51.2.		

8.97 BGND-096 In polygon cutting function, synchronous spindle rotation speed ratio is wrong

Alarm ID	COR-096 BGND-096	Alarm Title	In polygon cutting function, synchronous spindle rotation speed ratio is wrong
Description	When using polygon cutting G51.2 function, basic spindle rotation speed ratio (Q argument) must be bigger than zero.		
Reason	Programming error.		
Solution	Check the value of Q argument in G51.2.		

8.98 BGND-097 Axis coupling function ON or OFF failed

Alarm ID	COR-097 BGND-097	Alarm Title	Axis coupling function ON or OFF failed
Description	Axis coupling function ON or OFF failed.		
Reason	Too much axis coupling groups is used.		
Solution	Check if the number of axis coupling groups is more than 16 (including the number of axis coupling group configured in parameter).		

8.99 BGND-098 Parameter learning argument error

Alarm ID	COR-098 BGND-098	Alarm Title	Parameter learning argument error
Description	When using learning function, if argument (P, Q, R, K) setting is wrong, system will issue this alarm.		

Alarm ID	COR-098 BGND-098	Alarm Title	Parameter learning argument error
Reason	Programming error.		
Solution	Check the NC program to ensure the arguments are in compliance with specifications.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

8.100 BGND-099 Tapping learning condition is not matched

Alarm ID	COR-099 BGND-099	Alarm Title	Tapping learning condition is not matched
Description	Condition of tapping learning function is not matched		
Reason	When activating tapping learning function, if tapping condition (tapping depth, R point height, feedrate, rotation speed, and etc.) is different, it can't apply to same learning data. If force different tapping condition to use same learning data, it may misuse learning data, and cause tapping error or even tool break.		
Solution	Check tapping instruction in tapping learning function, and confirm all tapping instruction arguments are the same.		

P.S. Valid version of COR-336 : before 10.118.60J、10.118.66D、10.118.69 (included).

8.101 BGND-100 Unsupported G code command or option software is not activated

Alarm ID	COR-100 BGND-100	Alarm Title	Unsupported G code command or option software is not activated
Description	Different controllers will have correspond G code, but not all G code can use.		

Alarm ID	COR-100 BGND-100	Alarm Title	Unsupported G code command or option software is not activated
Reason	1. This controller type may not support this G code command. 2. This controller type will not support serial bus spindle (C-Type) to use lathe G21, G33, G34, G78 commands. 3. This controller type will not support serial bus spindle (A-Type) to use lathe G32, G73, G76, G92 commands. 4. This controller type can support this G code command, but the option software function has not been purchased, which makes the G code unusable. 5. Loader path and Woodworking label path only support part of G codes: G00, G01, G02, G03, G04, G04.1, G09, G10, G17, G18, G19, G22, G23, G31, G52, G53, G54, G55, G56, G57, G58, G59, G59.x, G90, G91, G92. 6. The setting of Pr3802 is incorrect. This controller type does not support the G62 command.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

8.102 BGND-101 Under Spindle synchronization – thread pitches of two spindle is inconsistent

Alarm ID	COR-101 BGND-101	Alarm Title	Under Spindle synchronization – thread pitches of two spindle is inconsistent
Description	When using spindle synchronize, if basic spindle and synchronous spindle pitch setting are different and will cause synchronous abnormal condition. And will issue this alarm for protection.		
Possible Cause	Parameter setting error.		
Solution	Check the spindle used in Pr161~180 to ensure the parameter configurations are the same.		

8.103 BGND-102 Under Spindle synchronization – synchronization command duplicated or conflicted

Alarm ID	**COR-102** **BGND-102**	**Alarm Title**	**Under Spindle synchronization – synchronization command duplicated or conflicted**
Description	When spindle synchronizing, repeat G51.2, G114.1 or G114.3 commands.		
Possible Cause	Haven't use G113 to cancel G114.1 and G114.3, or use G50.2 to cancel G51.2, and again set a repeated G114.1, G114.3 or G51.2 commands.		
Solution	Cancel the G114.1 and G114.3 mode with G113, or cancel G51.2 mode with G50.2. Then execute the G114.1/G114.3/G51.2 commands to avoid the angle difference for repeated machining.		

8.104 BGND-103 High-speed high-precision configuration improper

Alarm ID	**COR-103** **BGND-103**	**Alarm Title**	**Invalid precision channel setting**
Description	Invalid precision channel setting.		
Possible Cause	Using G64, G62P_ or G120.1 P_Q_ to choose invalid precision channel. Currently can only choose: 1. G64、G62 P0~P9、G62/G64 P21 ~ P23 2. G120.1 P0、G120.1 P1 Q1 ~ G120.1 P3 Q3		
Solution	To select precision channel with the G64/G62 P_ or G120.1 P_ Q_. Check the NC program to ensure if any non-existing precision channel is selected.		

8.105 BGND-104 Virtual axis function enabling failed

Alarm ID	**COR-104** **BGND-104**	**Alarm Title**	**Virtual axis function enabling failed**
Description	The virtual axis function failed to be functioned.		

Alarm ID	COR-104 BGND-104	Alarm Title	Virtual axis function enabling failed
Possible Cause	1. The P and Q arguments in G10 L800 P_ Q_ or G10 L801 P_ Q_ are set incorrectly. a. No P, Q arguments. b. The P argument is set incorrectly and the value is invalid. c. The Q argument is set incorrectly, the value is invalid; or the corresponding axis is not set to the axis of the parameter table that has been turned on. 2. In the same machining, G10 L800 and G10 L801 appear at the same time. 3. The robot does not support the virtual axis function.		
Solution	1. Check the virtual axis commands to ensure the P and Q Arguments are configured. 2. Check the virtual axis commands to ensure the P Argument shall be within 100~999 and the Q Argument shall be within 1~16 (axis number) or 100~999 (axis name). 3. Check the axial direction (Q Argument) mapped with the virtual axis to ensure the parameter table is configured to the enabled axial direction. 4. Check if the G10 L800 and G10 L801 shall not be existent in the same NC program.		

8.106 BGND-105 Axial coupling configuration error

Alarm ID	COR-105 BGND-105	Alarm Title	Axial coupling configuration error
Description	The axis coupling function is set incorrectly.		
Possible Cause	In G10 L900 P_ Q_ R_, the P and Q arguments are set incorrectly.		
Solution	1. Check the name or number of coupling axis is mapped to ensure the axial direction is enabled. 2. Chedk the specified coupling axis is not assigned to inclined axes control. 3. Check the specified coupling axis is not assigned to indexing axis.		

SYNTEC

8.107 BGND-106 Prohibit to use G5.1 in G61/G63/G63.2 mode

Alarm ID	COR-106 BGND-106	Alarm Title	Prohibit to use G5.1 in G61/G63/ G63.2 mode
Description	1. G5.1 path smoothing. In G61, G63, G63.2 mode, it is forbidden to use G5.1 smoothing function, otherwise the system will issue an alarm; If G61, G63, G63.2 are enabled in G5.1 mode, the system will stop the smoothing function until it jumps out of G61, G63, G63.2 mode and then activate again. 2. G05 High Precision Contour Control. a. 10.116.16J previous version, in G61, G63, G63.2 mode, it is forbidden to use G05 High Precision Contour Control function, otherwise the system will issue an alarm; If G61, G63, G63.2 is enabled in G05 mode, no alarm will be issued, but the High Precision Contour Control function is disabled, It is necessary to command G05 again to activate. b. above 10.116.16K versions, no alarm will be issued, and the G05 High Precision Contour Control function will be disabled, and can be valid after G61, G63 and G63.2.		
Possible Cause	The cutting mode setting in the NC program is incorrect.		
Solution	1. Do not mix use G5.1 path smoothing with G61/G63/G63.2 functions. 2. In the version before 10.116.16J (included), do not mix use the G05 High Precision Contour Control with the G61/G63/G63.2 functions.		

8.108 BGND-107 G5.1/G05 command format error

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Description	The G5.1 and G05 commands are in the wrong format.		
Possible Cause	1. The format of the G5.1 path smoothing command in the NC program is incorrect. 2. The G05 high-precision cutting mode command format in the NC program is incorrect.		

Alarm ID	COR-107 BGND-107	Alarm Title	G5.1/G05 command format error
Solution	Confirm the following command formats are correct not have these error: 1. G5.1 a. Q argument: None, more than 2, or less than 0. b. E argument: None or less than 0. 2. G05 a. System issue alarm when using G05 in following cases for each version: i. G05 P argument is not 10000 nor 0. ii. G05 E argument is not positive. b. Activate command G05 P10000 X0 Y0 Z0 α_ β_ in 10.116.36 or above versions: i. More than 5 axial directions are assigned. ii. The geometry axis argument not 0. iii. The rotary axis argument is configured to 0. iv. The axial direction of geometry axis is configured but this of rotary axis is not. v. The axis of the rotation axis is not set when the axis of the geometry axis is not set. vi. More than 2 axial directions of rotation axes are configured. vii. Any axial arguments is negative. c. In the version before 10.116.16B, there is the 4th axis command in addition to the block movement commands of X, Y, or Z axes after G05 is executed.		

8.109 BGND-108 G10 L1501/L1502 command format error

Alarm ID	COR-108 BGND-108	Alarm Title	G10 L1501/L1502 command format error
Description	There is no NC program between G10 L1501 and G10 L1502.		
Possible Cause	The NC program was written incorrectly.		
Solution	Check the NC program to ensure the programming between G10 L1501 and G10 L1502 is correct.		

8.110 BGND-109 The second software stroke limit command error, activating failure

Alarm ID	COR-109 BGND-109	Alarm Title	The second software stroke limit command error, activating failure
Description	The second software stroke limit (G22) command is written incorrectly, causing the second software stroke limit fail to activate.		
Possible Cause	The arguments written after the second software stroke limit (G22) command are wrong, and the same group parameters should exist at the same time.		
Solution	Check if the arguments in the same set after G22 are defined completely.		

8.111 BGND-110 NC program of spring machine motion plan more than 500 lines

Alarm ID	COR-110 BGND-110	Alarm Title	NC program of spring machine motion plan more than 500 lines
Description	The number of machining lines between G10 L1501 and G10 L1502 exceeds 500 lines.		
Possible Cause	There are too many single blocks in the NC program.		
Solution	Check and refine the NC program.		

8.112 BGND-111 Axis exchange function – axial configuration error

Alarm ID	COR-111 BGND-111	Alarm Title	Axis exchange function – axial configuration error
Description	Under the axis exchange function, the axial setting is incorrect.		
Possible Cause	The two axial's axis exchange parameters are set incorrectly.		
Solution	Check if Pr3721 and Pr3722 are mapped to physical axial directions (Pr21~) and to 2 different axis.		

8.113 BGND-112 Axis exchange function – diameter/radius axis configuration error

Alarm ID	COR-112 BGND-112	Alarm Title	Axis exchange function – diameter/radius axis configuration error
Description	Under the axis exchange function, the diameter axis and radius axis are set incorrectly.		
Possible Cause	The parameters of the diameter and radius axis of the two axis exchange are set incorrectly.		
Solution	Check if the diameter and radius axis configurations (Pr281~) of the 2 axis to be exchanged is consistent.		

8.114 BGND-113 Axis exchange function – ON or OFF timing error

Alarm ID	COR-113 BGND-113	Alarm Title	Axis exchange function – ON or OFF timing error
Description	When the axis exchange function is enabled, the enabled or disabled timing error.		
Possible Cause	When the cross-path axis is exchanged, the G04.1 waiting action is not performed.		
Solution	Check PLC to ensure no G04.1 waiting command is executed in more than 1 path.		

8.115 BGND-114 Axis exchange function – path configuration error

Alarm ID	COR-114 BGND-114	Alarm Title	Axis exchange function – path configuration error
Description	When using the axis exchange function, the path setting is incorrect.		
Possible Cause	The axis used for axis exchange belong to multi-paths.		
Solution	Check path parameter (Pr701~) if the two axis to be exchanged are mapped to the multiple path.		

8.116 BGND-115 G92.1 axial configuration error

Alarm ID	COR-115 BGND-115	Alarm Title	G92.1 axial configuration error
Description	When using the G92.1 rotation function, the axial setting is incorrect.		
Possible Cause	The parameter sets the first 3 axis of the path contains the rotary axis.		
Solution	Check the parameter to ensure the first 3 axis are linear axis.		

8.117 BGND-116 Absolute coordinate system synchronization failed

Alarm ID	COR-116 BGND-116	Alarm Title	Absolute coordinate system synchronization failed
Description	Absolute coordinate system failed to synchronize when using C35.		
Possible Cause	Machine is not stationary or the program is continuously interpreted.		
Solution	To execute the synchronization of coordinate system in C35 mode shall ensure the machine is still and the program stops to interpret (i.e. Using M code shall turn C38 off).		

8.118 BGND-117 M code not disabled in the interrupt subprogram

Alarm ID	COR-117 BGND-117	Alarm Title	M code not disabled in the interrupt subprogram
Description	If Pr3600 is set to n, when the interrupt type subprogram is used, it starts with M(n) and ends with M(n+1). If M(n) is used and M(n+1) is not used to end the function, it will cause an action error; therefore, this alarm is issued for protection.		
Possible Cause	The NC programming error.		
Solution	Check the NC program to ensure the M(n+1) command is given.		

8.119 BGND-118 Prohibit G53 commands in tool tip control mode

Alarm ID	COR-118 BGND-118	Alarm Title	Prohibit G53 commands in tool tip control mode
Description	G53 command cannot be used in the tool point control mode.		
Possible Cause	1. The NC programming error. 2. The machine type is the tool point control mode.		
Solution	1. Please check the NC program, make sure that the G53 command is not within the validity range of G43.4 or G43.5. 2. Please check the NC program, make sure that the G53 command is not within the validity of G12.1. 3. If the machine configuration used is the tool point control mode, the G53 command cannot be used.		

8.120 BGND-119 G10 L16 command format error

Alarm ID	COR-119 BGND-119	Alarm Title	G10 L16 command format error
Description	Since the command G10 L16 (virtual circle radius) does not support functions such as G05, G7.1, G12.1, G93, G95, and G05, therefore, an alarm will be issued if G10 L16 is used when the above command is issued.		
Possible Cause	1. The axis type is linear axis (i.e. G10 L16 X50., and X is set to a linear axis, this alarm is issued). 2. When using G10 L16 while command G05, G7.1, G12.1, G93, G95 and etc. functions are also given.		
Solution	1. Check Pr221~ to ensure the correct axis type. 2. G10 L16 and G7.1/G12.1/G93/G95/G05 shall not be executed at the same time.		

SYNTEC

8.121 BGND-120 More than the maximum number of cutting synchronization axis

Alarm ID	COR-120 BGND-120	Alarm Title	More than the maximum number of cutting synchronization axis
Description	Different controller products can correspond to different maximum number of simultaneous cutting axis, and if this limit is exceeded, an alarm will be issued. (i.e. G01 X10. Y10 .Z10. Represents the number of simultaneous axes is 3).		
Possible Cause	Please refer to "System Information" under the controller HMI screen to check "Machine Properties" and "Machine Code". Please refer to the catalogue for each product specification. Following examples: 1. If "Machine Properties" is Mill, "Machine Code" is 200A-5, and the maximum number of axis is 9. 2. If the "Machine Properties" is Lathe, the "Machine Code" is 6B, and the maximum number of axis is 4.		
Solution	The number of simultaneous axis in the NC program shall be according to the product specifications.		

8.122 BGND-121 LN operator shall be positive

Alarm ID	COR-121 BGND-121	Alarm Title	LN operator shall be positive
Description	The operands after the LN function cannot be negative.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

8.123 BGND-122 The 1st operator of POW shall not be negative

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Description	The base operand of the POW function must not be a negative number.		

Alarm ID	COR-122 BGND-122	Alarm Title	The 1st operator of POW shall not be negative
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

8.124 BGND-123 Illegal STR2INT input or too long string

Alarm ID	COR-123 BGND-123	Alarm Title	Illegal STR2INT input or too long string
Description	The syntax of the STR2INT function is incorrect or the string is too long.		
Possible Cause	The NC program was written incorrectly.		
Solution	Modify the contents of the NC program.		

8.125 BGND-124 S code commands unsupported in the machine

Alarm ID	COR-124 BGND-124	Alarm Title	S code commands unsupported in the machine
Description	This model does not support S code commands.		
Possible Cause	This model does not support S code commands.		
Solution	Replace a controller model that supports S codes.		

8.126 BGND-125 T code commands unsupported in the machine

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Description	T code command is illegal.		

Alarm ID	COR-125 BGND-125	Alarm Title	T code commands unsupported in the machine
Possible Cause	1. This model does not support T code commands. 2. The T code command is out of the T code range supported by the product.		
Solution	Replace a controller model that supports T codes.		

8.127 BGND-126 H code commands unsupported in the machine

Alarm ID	COR-126 BGND-126	Alarm Title	H code commands unsupported in the machine
Description	This model does not support H code commands.		
Possible Cause	This model does not support H code commands.		
Solution	Replace a controller model that supports H codes.		

8.128 BGND-127 D code commands unsupported in the machine

Alarm ID	COR-127 BGND-127	Alarm Title	D code commands unsupported in the machine
Description	This model does not support D code commands		
Possible Cause	This model does not support D code commands		
Solution	Replace a controller model that supports D codes.		

8.129 BGND-131 Too much M/T code in a block

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Description	The total number of M codes and T codes in the same block exceeds 20.		

Alarm ID	COR-131 BGND-131	Alarm Title	Too much M/T code in a block
Possible Cause	NC programming error.		
Solution	Check the NC program to ensure the total number of M and T codes are not more than 20 in one single block.		

8.130 BGND-132 Illegal character in the program name

Alarm ID	COR-132 BGND-132	Alarm Title	Illegal character in the program name
Description	When open a file using Macro, the specified file name contains illegal characters.		
Possible Cause	NC programming error.		
Solution	Check the specified file names.		

8.131 BGND-133 Command unsupported in Three-points arc interpolation mode

Alarm ID	COR-133 BGND-133	Alarm Title	Command unsupported in Three-points arc interpolation mode
Description	This command is not supported in the three-point arc interpolation mode (G02.4, G03.4)		
Possible Cause	1. The tool radius compensation function is not turned off before using this function. 2. It is not supported in G62 cutting mode. 3. In the three-point arc interpolation mode (G02.4, G03.4), A, C, and R commands are not supported.		
Solution	Check the NC program to ensure there is no command described in the Possible Cause.		

8.132 BGND-134 Three-points arc interpolation command format error

Alarm ID	COR-134 BGND-134	Alarm Title	Three-points arc interpolation command format error
Description	The three-point arc interpolation (G02.4, G03.4) command format is wrong.		
Possible Cause	The three-point arc interpolation (G02.4, G03.4) command is regarded as a group of two lines, which can be specified continuously. The end point of the previous arc is the starting point of the next arc, but the F command can only be in the odd line. This alarm is issued if the total number of lines in the command is odd or the number of lines in the F command is even.		
Solution	Check the three-point arc interpolation G02.4/G03.4 command format in NC program.		

8.133 BGND-135 Read/write command format error for R value

Alarm ID	COR-135 BGND-135	Alarm Title	Read/write command format error for R value
Description	SETRREGBIT, READRREGBIT bring in an argument with type error or range error.		
Possible Cause	1. SETRREGBIT (R value number, specify BIT, on or off) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31. c. If the third argument is not 0 (off) or 1 (on). 2. READRREGBIT (R value number, specify BIT) a. If the R value is less than 0 or greater than 65535. b. Specify if the BIT is less than 0 or greater than 31.		
Solution	Correct the argument type or range in the commands, SETRREGBIT and READRREGBIT		

8.134 BGND-136 Please reboot the controller when axis tuning failed

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Description	After a certain serial bus axial tuning fails, without rebooting, cycle start is triggered.		

Alarm ID	COR-136 BGND-136	Alarm Title	Please reboot the controller when axis tuning failed
Possible Cause	The serial bus axial tuning failed.		
Solution	Reboot controller.		

8.135 BGND-137 Path synchronization waiting's P-argument sequence error

Alarm ID	COR–137 BGND-137	Alarm title	Path synchronization waiting's P-argument sequence error
Description	The machining multi-path program uses the path synchronization waiting (G04.1) to wait for the synchronization action: 1. When the Q argument is not selected, when two programs go to the position of the path synchronization waiting (G04.1) at the same time, if the P arguments substituted are different, this alarm is issued for reminder. 2. When the next Q argument (decimal) specifies the path waiting for each other, when the two programs go to the position of the path synchronization waiting (G04.1) at the same time, and wait for each other, but the P arguments substituted are different, this alarm is issued for reminder.		
Possible Cause	1. In the NC program of multiple paths, the order of P arguments is abnormal or the number is incorrect. 2. Machining is performed using M99, but in each program, the path without Q argument is synchronously waiting (G04.1), and the number is different. 3. Machining is performed using M99, but in each program, the path with the same Q argument is synchronously waiting (G04.1), and the number is different. 4. The two paths “wait for each other”, but the P argument substituted are not the same. i.e. G04.1 P1 Q123 under the first path and G04.1 P2 Q124 under the second path.		
Solution	Please check the number of Synchronization wait between paths (G04.1) and the order of the assigned value for argument P.		

8.136 BGND-138 Read/write command format error at the I/O/A point

Alarm ID	COR-138 BGND-138	Alarm Title	Read/write command format error at the I/O/A point
Description	The system provides 512 I/O/A points, the input I/O/A point number shall be within 0~511 when using the commands, SETDO, SETABIT, READDI, READDO, and READABIT.		
Possible Cause	NC Programming error.		
Solution	Check if there is any of I/O/A point number in the commands, SETDO, SETABIT, READDI, READDO, and READABIT in the NC program is over the range of 0 ~ 511.		

8.137 BGND-139 Polynomial solution error

Alarm ID	COR-139 BGND-139	Alarm Title	Polynomial solution error
Description	Determine the polynomial cannot be derived to solutions in case of the alarm.		
Possible Cause	Numerical processing calculation errors cause root failure.		
Solution	Check the value of system data no. 321, and provide it to the controller OEM Syntec for solutions.		

8.138 BGND-140 Prohibit G05 in tool tip control mode

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Description	1. Turn on G05 high-speed high-precision mode in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Possible Cause	1. In the RTCP/STCP mode, turn on the G05 high-precision contour control mode with commands, such as G05 P10000. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		
Solution	1. Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-precision contour control mode. If (1), remove the command to turn on the G05 high-precision contour control mode in the RTCP/STCP mode. If (2), turn off the RTCP/STCP mode before turning on the G05 high-precision contour control mode. 2. When the high-precision contour control mode is enabled during processing, do not use the single block stop C40 at the same time. 3. When the STCP mode is enabled during processing, do not use the single block stop C40 at the same time.		

8.139 BGND-141 G68.3 command format error

Alarm ID	COR-141 BGND-141	Alarm Title	G68.3 command format error
Description	[command format] G68.3 X_ Y_ Z_ R_; // The origin and z-axis rotation angle in the characteristic coordinate system. G68.3 P1 X_ Y_ Z_; // The origin of the characteristic coordinate system, and the coordinate system is determined with the tool rotation angle.		
Possible Cause	G68.3 command format, X, Y and Z are all exist or non-exist at the same time.		
Solution	Check if G68.3 command format is correct.		

8.140 BGND-142 Spindle synchronization – K Argument error

Alarm ID	COR-142 BGND-142	Alarm Title	Spindle synchronization – K Argument error
Description	K Argument error while using Spindle synchronization function.		
Possible Cause	The inputted K Argument is not within the valid range.		
Solution	Input a K Argument within the valid range to enable the Spindle synchronization.		

8.141 BGND-143 Programmable data input specified axis does not exist

Alarm ID	COR-143 BGND-143	Alarm Title	Programmable data input specified axis does not exist
Description	In G10 L1501 mode, the P argument is set incorrectly.		
Possible Cause	The axial direction substituted by the P Argument is inexistent.		
Solution	Check Pr21~ to ensure the axial direction is ON.		

8.142 BGND-144 Path synchronization waiting Q Argument content error

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Description	When the machining multiple path program applies the G04.1 to wait for synchronization, the Q Argument assigns the paths to be waited for each other (decimally). The alarm is used to remind the users if the Q Argument type error or the assigned path is inexistent.		

Alarm ID	COR-144 BGND-144	Alarm Title	Path synchronization waiting Q Argument content error
Possible Cause	1. The Q Argument is not a positive integer (negative number, decimal point and zero are all illegal). 2. The path assigned by the Q Argument is inexistent. i.e. Q24 is used to assign the 2nd and 4th path to wait for each other. However, the Pr731=3 and there are only 3 CNC main system paths. 3. The Q Argument includes 0 when assigning the paths. i.e. Q103. 4. The paths assigned by the Q Argument do not include this in which the command is. ie. In the 2nd path, the G04.1 P1 Q13 assigns the 1st and 3rd paths to wait for each other, but it does not assign the path in which it is.		
Solution	Check the Path synchronization waiting (G04.1) in the program to ensure the type and the assigned path are correct for a given Q Argument.		

8.143 BGND-145 Failed to activate Spindle positioning

Alarm ID	COR-145 BGND-145	Alarm Title	Failed to activate Spindle positioning
Description	Spindle positioning failed.		
Possible Cause	1. Incorrect Spindle ID is assigned. 2. The Spindle is operating in the tapping mode when the Spindle positioning function is ON. 3. After starting the positioning, C61 is OFF so that the positioning is broken.		
Solution	1. Check the Spindle parameter configuration. 2. Check the NC program to ensure the execution of Spindle positioning in non-tapping mode. 3. Check the PLC to ensure C61 is ON.		

8.144 BGND-146 Single block argument type error

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Description	Argument type error in a block.		

Alarm ID	COR-146 BGND-146	Alarm Title	Single block argument type error
Possible Cause	The two following situations may trigger this alarm: 1. Input float value to an argument required integer value. 2. Input integer value to an argument required float value.		
Solution	Correct the argument type in the block.		

8.145 BGND-147 Spindle is not enabled when path machining

Alarm ID	COR-147 BGND-147	Alarm Title	Spindle is not enabled when path machining
Description	The machining Spindle assigned by the path is not enabled when executing the commands related to the Spindle.		
Possible Cause	When executing the G33/G34/G63/G74/G84, the Spindle assigned by the R791~R794 is not enabled.		
Solution	1. Check if the PLC switches R791~R794 values correctly in the block and whole NC program about the alarm. 2. Check if the Pr1621~Pr1628 configurations are correct (cooperate with R791~R794).		

8.146 BGND-148 The use of tool retract function error

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Description	When using the tool retract function, a given improper argument results in the function cannot be executed.		
Possible Cause	1. In the Tilted working plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned axial position retraction. 2. In the non-Tilted working plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, the C21 ON is triggered to execute the assigned retraction along the tool vector.		

Alarm ID	COR-148 BGND-148	Alarm Title	The use of tool retract function error
Solution	1. In the slopping plane machining (G68.2/G68.3) or RTCP (G43.4/G43.5) mode, apply G10.6 R_ to assign the tool retraction function. 2. In the non-slopping plane machining (G68.2/G68.3) or non- RTCP (G43.4/G43.5) mode, apply G10.6 X_ to assign the tool retraction function.		

8.147 BGND-149 Tilted working plane machining tool alignment P Argument over range

Alarm ID	COR-149 BGND-149	Alarm Title	Tilted working plane machining tool alignment P Argument over range
Description	P Argument is over range in Tilted working plane machining tool alignment mode.		
Possible Cause	P Argument is not within 0~2.		
Solution	Do not input the P Argument or input the correction configuration for the P Argument in Tilted working plane machining tool alignment mode.		

8.148 BGND-151 1st rotation axis entering illegal range

Alarm ID	COR-151 BGND-151	Alarm Title	1st rotation axis entering illegal range
Description	1st rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1st rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3009 and Pr3010 are configured correctly. The determination of such two configurations is related to Pr3007. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

8.149 BGND-152 2nd rotation axis entering illegal range

Alarm ID	COR-152 BGND-152	Alarm Title	2nd rotation axis entering illegal range
Description	2nd rotation axis entering illegal range		
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3011 and Pr3012 are configured correctly. The determination of such two configurations is related to Pr3008. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

8.150 BGND-153 Tool direction unknown

Alarm ID	COR-153 BGND-153	Alarm Title	Tool direction unknown
Description	Tool direction unknown.		
Possible Cause	5-axis configurations and machine mechanism is incompatible.		
Solution	The tool cannot reach the destination. It may be caused by the incompatible 5-axis configurations and machine mechanism. Please check all 5-axis configurations.		

8.151 BGND-154 No 5-axis function

Alarm ID	COR-154 BGND-154	Alarm Title	No 5-axis function
Description	No 5-axis function.		
Possible Cause	Pr3001 is not configured when executing G53.1 tool alignment command.		
Solution	Check if Pr3001 is configured to 0. If yes, configure the other non-zero values based on the 5-axis mechanism type and reboot.		

8.152 BGND-155 5-axis tool direction error

Alarm ID	COR-155 BGND-155	Alarm Title	5-axis tool direction error
Description	5-axis tool direction error.		
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd rotation axis (Pr3005 and Pr3006) configuration error.		
Solution	Check if the Pr3002 is configured correctly, or if the Pr3005 or Pr3006 is configured correctly. The alarm will be triggered in case the 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

8.153 BGND-156 5-axis axial direction error

Alarm ID	COR-156 BGND-156	Alarm Title	5-axis axial direction error
Description	5-axis axial direction error.		
Possible Cause	Incorrect configurations are mapped to the axial direction parameters of 5 axis.		
Solution	Check if each axial direction is configured completely (Pr21~), if Pr3005, Pr3006, Pr3007 and Pr3008 is configured correctly, or if the axis name (Pr321~) is mapped to Pr3005 and Pr3006.		

8.154 BGND-157 Incompatible direction of 5-axis tool direction and this of rotation axis

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Description	Incompatible direction of 5-axis tool direction and this of rotation axis.		
Possible Cause	1. The 2nd axial direction and the tool direction are the same in the Spindle type. 2. The 1st axial direction and the tool direction are the same in the table type. 3. The 1st axial direction and the tool direction are the same in the hybrid type.		

Alarm ID	COR-157 BGND-157	Alarm Title	Incompatible direction of 5-axis tool direction and this of rotation axis
Solution	Check if the tool direction and the rotation axial direction are the same depended on the used 5-axis mechanism type.		

P.S. Valid version of COR-157 : before 10.118.41M, 10.118.47 (included).

8.155 BGND-158 Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode

Alarm ID	COR-158 BGND-158	Alarm Title	Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode
Description	Since the G43.5 mode specifies the tool attitude based on the tool vector I, J and K, it shall not be executed for the 1st and 2nd rotation axis commands which can also specify the tool attitude.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the movement commands of the 1st and 2nd rotation axis are over the valid range in the G43.5 mode.		

8.156 BGND-159 Illegal tool vector

Alarm ID	COR-159 BGND-159	Alarm Title	Illegal tool vector
Description	In NC program, a movement block assigns an incorrect tool vector.		
Possible Cause	Programming error. i.e. Execute G01 X_ Y_ Z_ I0 J0 K0 in the G43.5 mode, and the I0 J0 K0 refers to the 0 vector, 0 vector is illegal.		
Solution	Check the NC program to ensure the assigned tool vector is correct in the block indicated by the alarm line number		

8.157 BGND-160 5-axis mechanism chain switched when the 5-axis function is ON

Alarm ID	COR-160 BGND-160	Alarm Title	5-axis mechanism chain switched when the 5-axis function is ON
Description	In NC program, the command G10 L5000 [P_] is executed to switch the 5-axis mechanism when turning on the 5-axis function (RTCP or the tool vector alignment on the characteristic coordinate system is completed).		
Possible Cause	Programming error. i.e. Execute G10 L5000 [P_] in the G43.4 mode.		
Solution	Check the NC program to ensure the 5-axis function is not turned on in the block indicated by the alarm line number.		

8.158 BGND-161 Selected 5-axis mechanism chain is not ON

Alarm ID	COR-161 BGND-161	Alarm Title	Selected 5-axis mechanism chain is not ON
Description	In NC program, the 5-axis mechanism chain parameters assigned by the command G10 L5000 [P_] is not ON.		
Possible Cause	Programming error. i.e. The assigned 5-axis mechanism chain is not configured correctly when executing a command G10 L5000 [P_].		
Solution	Check the 5-axis mechanism chain parameters to ensure the assigned 5-axis mechanism chain function is turned on correctly: 1. 1st set: Pr3001 2. 2nd set: Pr3101 3. 3rd set: Pr5501 4. 4th set: Pr5601		

8.159 BGND-162 4-axis RTCP configuration error

Alarm ID	COR-162 BGND-162	Alarm Title	4-axis RTCP configuration error
Description	The controller will issue the alarm in case the 4-axis RTCP configuration is incorrect.		
Possible Cause	The 5-axis mechanism parameters, Pr3001, Pr3101, Pr5501 and Pr5601 are configured to 1~3 in case the specific 4-axis tool tip control function (option-29) is turned on but the tool tip control function (option-12) is not.		
Solution	1. Configure the 5-axis mechanism parameters to 4 or 5 correctly. 2. Turn on the tool tip control function (option-12).		

8.160 BGND-163 Multi-kinematic chain command Q Argument setting error.

Alarm ID	COR-163 BGND-163	Alarm Title	Multi-kinematic chain command Q Argument setting error.
Description	Command G10 L5000P_ Q_, Q argument range error.		
Possible Cause	Command G10 L5000P_ Q_, Q argument range error.		
Solution	While using G10 L5000P_ Q_, check Q argument to be within 0~4, and is a integer.		

8.161 BGND-164 Multi-kinematic chain command related 5-Axis mechanism setting error.

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Description	Command G10 L5000P_ Q_ specified the 5-Axis kinematic chain, and the 5-Axis mechanism parameter setting error.		
Possible Cause	While executing G10 L5000P_ Q_, Q argument is given, but the 5-Axis mechanism parameter of the designated 5-Axis kinematic-chain is not a spindle-type 5-Axis machine.		

Alarm ID	COR-164 BGND-164	Alarm Title	Multi-kinematic chain command related 5-Axis mechanism setting error.
Solution	Please check the designated 5-Axis kinematic-chain. The 5-Axis mechanism configuration parameter must be a spindle-type 5-Axis machine. 1. The first group : Pr3001 is 1. 2. The second group : Pr3101 is 1. 3. The third group : Pr5501 is 1. 4. The fourth group : Pr5601 is 1.		

8.162 BGND-165 Multi-kinematic chain command not illegal.

Alarm ID	COR-165 BGND-165	Alarm Title	Multi-kinematic chain command not illegal.
Description	Command G10 L5000P_ Q_ is used for switching 5-Axis kinematic chain, and only provides partial 5-Axis mechanism function command.		
Possible Cause	G10 L5000 P_ Q_ command, the Q argument is set to 2~4 (not the first group of sub-kinematic chain), and only supports the following 5-Axis machine function command. 1. RTCP: G43.4. 2. RTCP: G43.5. 3. Tilted working plane : G68.2 + Tool alignment functions. 4. Tilted working plane : G68.3. Notice: Tool alignment functions include G53.1, G53.3, G53.6, ...		
Solution	When using the G10 L5000 P_ Q_ command to switch multi-kinematic chains, please use the supported 5-Axis machine function command.		

8.163 BGND-166 Characteristic Coordinate System Option not supported

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Description	Option13 (Characteristic Coordinate System Option) was not active, therefore the CNC couldn't execute relevant commands.		

Alarm ID	COR-166 BGND-166	Alarm Title	Characteristic Coordinate System Option not supported
Possible Cause	One or more commands below were given while the Option13 (Characteristic Coordinate System Option) was inactive: 1. G68.2, G68.3 2. G53.1, G53.3, G53.6 3. Other commands that are relevant to Option13 (Characteristic Coordinate System Option)		
Solution	1. Activate Option13 (Characteristic Coordinate System Option) 2. Avoid using the commands listed above		

8.164 BGND-167 File name conflicts in NcFiles

Alarm ID	COR-167 BGND-167	Alarm Title	Program File name conflicts in NcFiles
Description	Program File name conflicts in NcFiles folder.		
Possible Cause	When Pr3220 is set to 1, "non-main system", "PLC axis", "pre- and post-processing auxiliary programs" restrict calling files from the Macro folder, and prohibit the NcFiles folder from having the same file name.		
Solution	1. Set Pr3220 to 0. 2. Remove the conflicting files in the NcFiles folder or change the file name.		

8.165 BGND-168 Illegal tool compensation

Alarm ID	COR-168 BGND-168	Alarm Title	Illegal tool compensation.
Description	Illegal tool compensation		
Possible Cause	1. Trying to use non-zero tool length or tool radius compensation value on a machine type which doesn't support tool compensation		

Alarm ID	COR-168 BGND-168	Alarm Title	Illegal tool compensation.
Solution	1. Set tool compensation value as zero 2. Turn off tool compensation related functions		

8.166 BGND-169 Overlap on the same axis

Alarm ID	COR-169 BGND-169	Alarm Title	Overlap on the same axis
Description	Overlap on the same axis.		
Possible Cause	Using G1.10 with argument Q on the same axis with two consecutive blocks.		
Solution	Please check NC Program and avoid overlapping on the same axis with two consecutive blocks.		

8.167 BGND-170 G43.4 L2 illegal argument value

Alarm ID	COR-170 BGND-170	Alarm 标题	【G43.4 L2 illegal argument value 】
说明	When using G43.4 L2. E_ R_, Illegal argument value is assigned		
可能原因	Illegal argument value 1. R value is negative 2. E value is not in the range of 0.001 to 179.999		
排除方法	Please check the E_ R_ argument values		

8.168 BGND-201 Program file not exist

Alarm ID	COR–201 BGND-201	Alarm title	Program file not exist
Description	The specified program does not exist.		
Possible Cause	The specified program does not exist.		
Solution	Ensure that program file exists.		

8.169 BGND-202 Communication link error

Alarm ID	COR–202 BGND-202	Alarm title	Communication link error
Description	Communication link is dropped.		
Possible Cause	The transmission communication link is dropped.		
Solution	Reconnect a good transmission communication link.		

8.170 BGND-203 Illegal NC file format

Alarm ID	COR–203 BGND-203	Alarm title	Illegal NC file format
Description	NC program format not valid, as a result, the system cannot fully interpret the NC program.		
Possible Cause	1. The NC file is in macro format. 2. Use M98 to call a multi-path subprogram (including $1 and $2), and the size of the subprogram is greater than 60KB(60000bytes).		
Solution	1. Update the controller to 10.114.50I or the later version. 2. For M98 application, reduce the size of subprogram. Or, split it into individual single-path subprograms, then call each of them by coordinate respectively.		

8.171 BGND-204 File size too large

Alarm ID	COR–204 BGND-204	Alarm title	File size too large
Description	Program file is too large.		
Possible Cause	If the size of MACRO program is larger than 60KB(60000bytes) and use the syntax with scope statement. The syntax with scope statement contains: IF, CASE, REPEAT, FOR, WHILE		
Solution	1. Reduce the program size, or split program into two subprograms. 2. Remove all the syntax with scope statement.		

8.172 BGND-205 File content is empty

Alarm ID	COR–205 BGND-205	Alarm title	File content is empty
Description	After controller loads the program, the file content found to be null.		
Possible Cause	Program loading error or CF card damaged.		
Solution	Reload program or replace CF card.		

8.173 BGND-206 Loading page lock failure

Alarm ID	COR–206 BGND-206	Alarm title	Loading page lock failure
Description	New NC program fails to required the system to distribute loading page.		
Possible Cause	Lack of memory when multi-system executes large-size program.		
Solution	Please upgrade to 9.240.27/ 9.242.5/10.104.36/10.111.41/10.113.5 or contact OEM Syntec.		

8.174 BGND-207 Specified sequence number not found

Alarm ID	COR–207 BGND-207	Alarm title	Specified sequence number not found
Description	Specified sequence number is not found.		
Possible Cause	Programming error.		
Solution	Use a sequence number within the NC program range.		

8.175 BGND-208 Cannot use jump statement in sequential file

Alarm ID	COR–208 BGND-208	Alarm title	Cannot use jump statement in sequential file
Description	Using jump command to execute sequential file.		
Possible Cause	Use the jump command when executing the sequence file.		
Solution	Do not use jump command to execute sequential file.		

8.176 BGND-209 File format error

Alarm ID	COR–209 BGND-209	Alarm title	【File format error】
Description	Invalid File Format.		
Possible Cause	1. It is not a MACRO format file. For example, using APP syntax to call MARCO G200, G200 APP "appname", but this G200 file is not declared as a MACRO format file. 2. It is not a ISO format file.		
Solution	Please check the NC program, which should be written according to the file opening requirements.		

8.177 BGND-250 Wrong mechanism axial setting

Alarm ID	COR-250 BGND-250	Alarm title	Wrong mechanism axial setting
Description	The axial setting of the mechanism is wrong.		
Possible Cause	The number of axis and the axis name required for the corresponding mechanism conversion are not set.		
Solution	Set sufficient number of axis (Pr21~), and required axis names (Pr321~).		

8.178 BGND-251 The joint is not set to rotate in the right/left-hand direction

Alarm ID	COR-251 BGND-251	Alarm title	【The joint is not set to rotate in the right/left-hand direction】
Description	The joint is not set to the direction of rotation as the right hand, left hand rule.		
Possible Cause	Pr4141~Pr4150 are not set to the right-hand rule or the left-hand rule.		
Solution	Set the rotation direction of each joint, 0 is not allowed.		

8.179 BGND-252 The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position

Alarm ID	COR-252 BGND-252	Alarm title	【The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position】
Description	The 2D mechanism is converted and the target position cannot be reached. Please enter a reasonable target position.		
Possible Cause	The program coordinate position exceeds the limit of the machine.		

Alarm ID	COR-252 BGND-252	Alarm title	【The target position cannot be reached due to 2D mechanism transformation. Please enter a reasonable target position】
Solution	The position is not reasonable. Please re-check the NC program.		

8.180 BGND-253 Unworkable parallel mechanism posture

Alarm ID	COR -253 BGND-253	Alarm title	【Unworkable parallel mechanism posture】
Description	Parallel mechanism posture that is not feasible.		
Possible Cause	Currently at a parallel mechanism posture that is not feasible.		
Solution	After Reset, use the axial movement mode to leave this posture, or modify the target point.		

8.181 BGND-301 OPEN command format error

Alarm ID	COR-301 BGND-301	Alarm title	【OPEN command format error】
Description	OPEN command format error [command format] OPEN ("file name", "write file mode") (i) Designated as '**a**': Represent **retains** the original file content and **continuously** outputs the new data in the text file. (ii) Not specified or designated as '**w**': Represents the **emptying** of the original content and **re-outputs** the new material in the text file.		
Possible Cause	"Write mode" in the OPEN command format specifies an error. This alarm is issued if the string is specified as a string other than 'a' or 'w'.		
Solution	Set "FileWrite mode" to "a" or "w" according to the requirement of FileWrite.		

8.182 BGND-302 G10 L1150 command format error

Alarm ID	COR-302 BGND-302	Alarm title	【G10 L1150 command format error】
Description	G10 L1150 command format error.		
Possible Cause	G10 L1150 parameter numbering (P argument) is a non-integer or the input is out of range or not entered. G10 L1150 parameter value (R argument) is out of range or not entered.		
Solution	Set the G10 L1150 parameter numbering and values (P and R arguments) correctly.		

8.183 BGND-303 The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)

Alarm ID	COR-303 BGND-303	Alarm title	【The basic spindle cannot use the Spindle Synchronization function in position control mode (C63)】
Description	The basic spindle uses the spindle synchronization (G114.1, G51.2) commands in the position control mode.		
Possible Cause	When using this function, the basic spindle cannot be in position control mode.		
Solution	Write the NC program or PLC correctly so that the principal axis leaves the position control mode before the same period (C63).		

8.184 BGND-304 Thread cutting geometrical axis name error

Alarm ID	COR-304 BGND-304	Alarm title	【Thread cutting geometrical axis name error】
Description	Thread cutting (G33, G34) command format is incorrect. Only the geometric axis names such as X, Y, Z, X1, Y1, Z1, etc. are supported.		
Possible Cause	When the thread cutting (G33, G34) does not have X, Y, Z, X1, Y1, Z1... and other geometric axis arguments, this alarm is issued.		

Alarm ID	COR-304 BGND-304	Alarm title	【Thread cutting geometrical axis name error】
Solution	Adjust the axis names (Pr321~) of the geometrical axis to XYZ, X1/Y1/Z1...		

8.185 BGND-305 Relative position input method is forbidden in current mode

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Description	The mode in which the system is currently running cannot be used with the G91 incremental command.		
Possible Cause	G43.5 cannot be used with the G91 incremental command function: G43.5 determines the tool attitude through the tool vectors I, J, and K. The tool attitude is expressed only in absolute quantities.		
Solution	Check the NC program to confirm that G91 was not executed in G43.5 mode and G43.5 was not executed in G91 mode		

8.186 BGND-306 Advanced look-ahead function without the specified M code

Alarm ID	COR-306 BGND-306	Alarm title	【Advanced look-ahead function without the specified M code】
Description	The advanced look-ahead function is enabled, and the specified M code is not found after the complete NC program is interpreted.		
Possible Cause	1. Pr3599 is not set. 2. The NC programming error.		
Solution	1. Confirm whether the setting of Pr3599 is correct. 2. Check the NC program to confirm that an command has been given to speedily predecode M code (Pr3599).		

8.187 BGND-307 Advanced look-ahead of M code, wrong designation of argument P

Alarm ID	COR-307 BGND-307	Alarm title	【Advanced look-ahead of M code, wrong designation of argument P】
Description	The advanced look-ahead M code (Pr3599) must specify the P argument.		
Possible Cause	When using the advanced look-ahead function, the P argument is not specified or the P argument is not an integer		
Solution	Check the NC program and confirm that there is a P argument and specify a 4-digit code subprogram.		

8.188 BGND-308 Insufficient system memory, increase a single step displacement for advanced look-ahead function

Alarm ID	COR-308 BGND-308	Alarm title	【Insufficient system memory, increase a single step displacement for advanced look-ahead function】
Description	The system look-ahead memory is insufficient, and the subsequent move command cannot be interpreted and is stuck.		
Possible Cause	There are too many very short blocks in the program, so the interpretation conditions cannot be met.		
Solution	Check the NC program and increase a single step displacement amount appropriately.		

8.189 BGND-309 Advanced look-ahead function failed

Alarm ID	COR-309 BGND-309	Alarm title	【Advanced look-ahead function failed】
Description	The advanced look-ahead function cannot be executed correctly.		
Possible Cause	1. The interpolation mode is incorrect during advanced look-ahead. 2. The contents of the look-ahead NC program cannot be stored correctly.		

Alarm ID	COR-309 BGND-309	Alarm title	【Advanced look-ahead function failed】
Solution	1. Check if the subprogram of the look-ahead NC programs contains commands other than G01, G02, and G03. 2. Check if disk space is adequate.		

8.190 BGND-310 Look-ahead NC programs failed

Alarm ID	COR-310 BGND-310	Alarm title	【Look-ahead NC programs failed】
Description	Look-ahead NC program cannot be executed correctly.		
Possible Cause	1. The condition of the look-ahead NC program does not match: a. The version of the advanced look-ahead, the interpolation time, the number of axes, do not match. b. There is an abnormal interruption in the look-ahead, resulting in incomplete content of the look-ahead. c. To look-ahead the axis of movement in the subprogram, enter the advanced look-ahead M code in the main program. The coordinates before (i.e. M298) should be the same. (i.e. if the subprogram has X, Y, and Z three-axis movement commands, if the X, Y, and Z coordinates before look-ahead M code are modified after the look-ahead is completed, the look-ahead NC program will be executed. Issue this alarm.) 2. The contents of the look-ahead NC program cannot be read correctly.		
Solution	After advanced look-ahead under correct conditions, execute the look-ahead NC programs.		

8.191 BGND-311 Hard disk storage space is insufficient to place look-ahead NC programs

Alarm ID	COR-311 BGND-311	Alarm title	【Hard disk storage space is insufficient to place look-ahead NC programs】
Description	The hard disk storage space is not enough to place the look-ahead NC program.		
Possible Cause	There is not enough storage space on the hard drive.		

Alarm ID	COR-311 BGND-311	Alarm title	【Hard disk storage space is insufficient to place look-ahead NC programs】
Solution	Release hard disk storage space.		

8.192 BGND-320 Friction Compensation Adjustment setting is illegal

Alarm ID	COR-320 BGND-320	Alarm title	【Friction Compensation Adjustment setting is illegal】
Description	Friction compensation adjustment setting is illegal.		
Possible Cause	1. Selected adjustment mode is not supported. 2. No axis is assigned for adjustment. 3. Radius or feedrate is not configured in Comp ON/Comp OFF mode . 4. No. of circular setting is not configured in Comp Learning mode.		
Solution	1. Select other compensation mode and try again. 2. Please set a set of available axes. 3. Please set correct radius or feedrate. 4. Please set correct No. of circular setting. If the alarm still exists, please contact Syntec OEM.		

8.193 BGND-321 The arguments (P,Q,R) retrieved from the I point position must be integers

Alarm ID	COR-321 BGND-321	Alarm title	【The arguments (P,Q,R) retrieved from the I point position must be integers】
Description	The I value position taken argument (G10 L1010 P_ Q_ R_) is not an integer.		
Possible Cause	NC programming error.		
Solution	Modify the NC program to specify correct P, Q, and R arguments.		

8.194 BGND-322 I point position capture failed

Alarm ID	COR-322 BGND-322	Alarm title	【I point position capture failed】
Description	I value position capture function (G10 L1010 P_ Q_ R_), startup failed.		
Possible Cause	1. The specified P argument (axis number) does not exist. 2. The specified Q argument (I value) is not supported. 3. Exceeding the limit of the number of used groups, currently support to start three I value location captures at the same time.		
Solution	1. Modify the program to specify correct P, Q, and R arguments. 2. Turn off settings that are not in use.		

8.195 BGND-323 The arguments (P,R,I,J,K) captured from the driver signal position must be integers

Alarm ID	COR-323 BGND-323	Alarm title	【The arguments (P,R,I,J,K) captured from the driver signal position must be integers】
Description	The argument of the drive signal position (G10 L1011 P_ R_ I_ J_ K_) is not an integer.		
Possible Cause	The NC programming error.		
Solution	Modify the program to specify correct P, R, I, J, and K arguments.		

8.196 BGND-324 Driver signal position captured failed to activate

Alarm ID	COR-324 BGND-324	Alarm title	【Driver signal position captured failed to activate】
Description	The drive signal position capture function (G10 L1011 P_ R_ I_ J_ K_) failed to activate.		

Alarm ID	COR-324 BGND-324	Alarm title	【Driver signal position captured failed to activate】
Possible Cause	1. The range of R values set by the specified P and R arguments is invalid (occupied by the system or out of range). 2. The specified I argument (axis number) does not exist. 3. The specified J argument (which set of latches to use) is out of range. 4. The specified K argument (select signal source) is out of range.		
Solution	Modify the NC program to specify correct arguments.		

8.197 BGND-325 In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited

Alarm ID	COR-325 BGND-325	Alarm title	【In the polar coordinate interpolation mode, the use of the diameter & radius axis programming command is prohibited】
Description	In the polar coordinate interpolation (G12.1) mode, the diameter & radius axis programming switching command (G10.9) cannot be used.		
Possible Cause	In polar coordinate interpolation (G12.1) mode, if switch the diameter & radius axis program with the diameter & radius axis programming switching command (G10.9), this alarm is issued.		
Solution	In the polar coordinate interpolation (G12.1) mode, do not use the diameter & radius axis to program the switching command (G10.9). Please follow Pr281~Axis radius axis or diameter axis set value to move the command. [Remarks] For milling machine G12.1, the X axis is programmed in radius axis. For lathe machine G12.1, and the X axis programming mode can be set using Pr4020 (G12.1 X axis programming).		

SYNTEC

8.198 BGND-326 Diameter & radius axis programming command argument error

Alarm ID	**COR-326** **BGND-326**	**Alarm title**	**【Diameter & radius axis programming command argument error】**
Description	Diameter and Radius axis programming (G10.9) switching, command arguments are written incorrectly.		
Possible Cause	No axial programming is specified, or programming is specified as a value other than 0 and 1.		
Solution	1. Specify any axial direction after diameter & radius axis programming (G10.9). 2. Specify the argument value (programming mode) as 0 or 1.		

8.199 BGND-327 Skip function argument input error

Alarm ID	**COR-327** **BGND-327**	**Alarm title**	**【Skip function argument input error】**
Description	Skip function (G31) input argument is incorrect.		
Possible Cause	Skip function (G31) specifies both P and R arguments.		
Solution	Please modify the Skip function (G31), no specified both P and R.		

8.200 BGND-328 G10 L1800 command argument number, out of specification limit

Alarm ID	**COR-328** **BGND-328**	**Alarm title**	**【G10 L1800 command argument number, out of specification limit】**
Description	The command argument number is outside the specification limit. i.e. G10 L1800 I514 P10 R1, I argument is out of range.		
Possible Cause	The NC programming error.		
Solution	Please confirm the argument value of G10 L1800.		

P.S. Valid version of COR-328 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

8.201 BGND-329 G10 L1800 command number has exceeded the single block limit

Alarm ID	COR-329 BGND-329	Alarm title	【G10 L1800 command number has exceeded the single block limit】
Description	The number of G10 L1800 commands with different arguments exceeds the upper limit allowed by a single block, up to 5, or too many G10 L1800 commands.		
Possible Cause	The NC programming error.		
Solution	Please reduce the number of G10 L1800 commands with different arguments to less than 5 in a block, or reduce the number of G10 L1800s as a whole.		

P.S. Valid version of COR-329 : range from 10.118.12B, 10.118.13 to 10.118.28D, 10.118.31 (included).

8.202 BGND-330 Illegal interrupt signal format

Alarm ID	COR-330 BGND-330	Alarm title	【Illegal interrupt signal format】
Description	The command argument number is outside the specification limit. i.e. M96 P5566 I4 Q100 R1 L1000; I argument specifies the error signal source.		
Possible Cause	The NC programming error..		
Solution	Please check the NC program to ensure that the command arguments are correct.		

8.203 BGND-331 This product limits T Code function

Alarm ID	COR-331 BGND-331	Alarm title	【This product limits T Code function】
Description	This product only supports T0~T4.		
Possible Cause	This product only supports T0~T4.		
Solution	Use a products that support the full T-code function.		

8.204 BGND-332 Interrupt type subprogram (M96, M97) execution failure

Alarm ID	**COR-332** **BGND-332**	**Alarm title**	**【Interrupt type subprogram (M96, M97) execution failure】**
Description	When the interrupt type subprogram function is used, the execution fails when the interrupt signal is triggered. Note: Due to the pre-solved relationship when issuing this alarm, there is no guarantee that it will stop at the problem line.		
Possible Cause	Only support to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). Triggering interrupt signals in other subprograms is not supported.		
Solution	1. Make sure to trigger interrupt where in the interrupt type subprogram enable or cancel (M96, M97). 2. Set #1510 to 4 (the second bit is set to on) and only display the line number of the main program. This can be regarded as triggering the interrupt in the main program, but this setting will re-execute the subprogram when it returns.		

8.205 BGND-333 Single end point exceeds hardware stroke limit

Alarm ID	**COR-333** **BGND-333**	**Alarm title**	**【Single end point exceeds hardware stroke limit】**
Description	The coordinate position in the program exceeds the hardware stroke limit set by the machine.		
Possible Cause	1. The NC program is wrong. 2. The hardware stroke limit signal is abnormal.		
Solution	1. Check the NC program and correct the coordinate position. 2. First confirm whether it is occasionally happen during machining, and MOT-25, 26 is no longer exists after the system enters the feedhold. If so, it may be that an axis hardware stroke limit signal is triggered, but then disappears; please check the limit switch wiring or the hardware itself for abnormality.		

8.206 BGND-334 G10 L1810 illegal signal condition command format

Alarm ID	COR-334 BGND-334	Alarm title	【G10 L1810 illegal signal condition command format】
Description	G10 L1810 command format error.		
Possible Cause	1. One of the I, Q, and R arguments is not set. 2. The I, Q, R, and J arguments are out of range, set to a negative, or a decimal value.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-334 : before 10.118.34 (included).

8.207 BGND-335 G10 L1810 signal waiting condition is too many or repeat the same signal

Alarm ID	COR-335 BGND-335	Alarm title	【G10 L1810 signal waiting condition is too many or repeat the same signal】
Description	The G10 L1810 signal waiting condition exceeds the allowable number, or the G10 L1810 repeatedly sets the same signal.		
Possible Cause	1. G10 L1810 continuously sets more than 5 commands. 2. The same G10 L1810 signal command has been set.		
Solution	Please combine multiple signals into one signal to wait, or reduce the signal waiting condition.		

8.208 BGND-336 G10 L1820 illegal waiting signal condition command format

Alarm ID	COR-336 BGND-336	Alarm title	【G10 L1820 illegal waiting signal condition command format】
Description	G10 L1820 command format error.		
Possible Cause	P, K arguments are out of range, set negative or decimal.		
Solution	Please refer to the manual for troubleshooting.		

P.S. Valid version of COR-336 : before 10.118.34 (included).

8.209 BGND-337 G10 L1820 without G10 L1810 command in the front

Alarm ID	**COR-337** **BGND-337**	**Alarm title**	**【G10 L1820 without G10 L1810 command in the front】**
Description	The G10 L1820 command needs to wait for the G10 L1810 signal wait condition. At least one G10 L1810 command must be programmed first.		
Possible Cause	The G10 L1820 command was not written before the G10 L1820 command.		
Solution	Modify the NC program and write the G10 L1810 command before the G10 L1820 command.		

8.210 BGND-338 G10 L1820 signal condition waiting timeout

Alarm ID	**COR-338** **BGND-338**	**Alarm title**	**【G10 L1820 signal condition waiting timeout】**
Description	G10 L1820 waiting signal condition exceeds the waiting time.		
Possible Cause	The signal status condition was not reached within the waiting time.		
Solution	1. Check if the G10 L1810 setting signal condition is correct. 2. Check if the PLC wait signal is processed correctly. 3. Check the hardware device to confirm why the signal status is not reached.		

8.211 BGND-339 Chopping axis prohibition movement command

Alarm ID	**COR-339** **BGND-339**	**Alarm title**	**【Chopping axis prohibition movement command】**
Description	The chopping axial direction does not accept any movement commands.		
Possible Cause	After using the chopping function (G81.1, G81.2, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.		

Alarm ID	COR-339 BGND-339	Alarm title	【Chopping axis prohibition movement command】
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.		

8.212 BGND-340 Chopping axis prohibits changing coordinate system

Alarm ID	COR-340 BGND-340	Alarm title	【Chopping axis prohibits changing coordinate system】
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.		
Possible Cause	1. After using the chopping function (G81.1, G81.2, C86), switch the coordinate system before closing and affect the chopping axis. 2. Simultaneously use of chopping function (G81.1, G81.2, C86) and tilted work plane machining function (G68.2, G68.3). 3. Simultaneously use the chopping function (G81.1, G81.2, C86) and the axis exchange function (C133~C136). Note: C86 support version: 10.118.19 and earlier.		
Solution	1. Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. 2. Check if the NC program uses the chopping function (G81.1, G81.2,C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). 3. Check if the NC program has the chopping function (G81.1, G81.2, C86) and the shaft exchange of the chopping shaft. (C133~C136). Note 1: C86 support version: 10.118.19 and earlier. Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658). Note 3: Coordinate system related operations: external coordinate offset, MPG offset.		

8.213 BGND-341 Chopping axial switching error

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Description	The specified axis cannot be switched to the chopping axis.		

Alarm ID	COR-341 BGND-341	Alarm title	【Chopping axial switching error】
Possible Cause	1. This axis has been designated as the PLC axis. 2. The axis has been designated as the spindle. 3. This axis has been designated as a chopping axis by other paths.		
Solution	1. Do not specify the PLC axis as the chopping axis. 2. Do not specify the spindle as the chopping axis. 3. Check if the multi-path repeats the chopping function for the same axis (G81.1, G81.2, C86). Note: C86 support version: 10.118.19 and earlier version.		

8.214 BGND-342 Chopping axis prohibits RTCP mode

Alarm ID	COR-342 BGND-342	Alarm title	【Chopping axis prohibits non-linear kinematic transform】
Description	Non-linear kinematic transform is prohibited for the chopping axis.		
Possible Cause	1. Using non-linear kinematic transform when chopping function is enabled. 2. The machine type used is two-dimensional kinemetic (special model).		
Solution	1. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the range of the tangential control (G41.1, G42.1). 3. Check the NC program to make sure that the chopping function (G81.1, G81.2, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 4. The machine configuration used is the two-dimensional kinemetic (special model), and the chopping function (G81.1, G81.2, C86) cannot be used. Note: C86 support version: 10.118.19 and earlier.		

8.215 BGND-343 The main program does not use the ending M code

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Description	M02/M30/M99 is not used at the end of the main program.		

Alarm ID	COR-343 BGND-343	Alarm title	【The main program does not use the ending M code】
Possible Cause	1. Programming error. 2. NC program is damaged. 3. Some machining information was lost due to file transmission errors.		
Solution	1. Check the NC program to reset the alarm and clear it. 2. Close the Pr3853-NC program error check function.		

8.216 BGND-344 The NC program under invalid paragraph

Alarm ID	COR-344 BGND-344	Alarm title	【The NC program under invalid paragraph】
Description	1. There is continuous blank/annotation/garbled in the NC program, and the size is more than 8KB. 2. There are no valid commands in the NC program.		
Possible Cause	1. Programmning error. 2. NC program is damaged.		
Solution	1. Avoid writing lots of consecutive blanks or annotations. 2. Please check if the NC program is damaged. After the problem is solved, reset to remove the alarm. 3. Close the Pr3853 - NC program error check function.		

8.217 BGND-345 Illegally format input to programmable data

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Description	Programmable data input argument format error.		

Alarm ID	COR-345 BGND-345	Alarm title	【Illegally format input to programmable data】
Possible Cause	1. Wobble command a. One of the I, Q, R arguments is not set. b. I, Q, R arguments out of range. c. I argument not integer. 2. G10 L1803 command a. One of the I, Q, P, R arguments is not set. b. I, Q, P, R, J, E arguments out of range. c. I, Q, R, J, K, E arguments not integer. 3. G10 L1805 、G10 L1810 command a. One of the I, Q, R arguments is not set. b. I, Q, R, J arguments out of range. c. I, Q, R, J arguments not integer. 4. G10 L1820 command a. P, K arguments out of range. b. P, K arguments not positive integer. 5. G10 L4500 command a. One of the P, Q, R arguments is not set. b. P argument is not a string. c. Q, R arguments out of range. d. Q, R arguments not integer. 6. G10 L1030 command a. P、Q、R、X、Y、Z、I、J、K、U、V arguments out of range. b. Q、X、Y、Z、I、J、K、U、V arguments not integer. 7. G10 L1031 command a. R argument is not set. b. R, V arguments out of range. c. R, V arguments not integer. 8. G10 L1500 command a. One of the P、Q arguments is not set. b. One of the P、Q arguments is out of range. c. One of the P、Q arguments is not an integer. 9. G10 L1350 command a. P, E argument is not set. b. P, E argument is out of range. c. P argument not integer.		
Solution	Please refer to command description to troubleshoot.		

8.218 BGND-346 Illegally use of programmable data entry

Alarm ID	COR-346 BGND-346	Alarm title	【Illegally use of programmable data entry】
Description	Programmable data input conflicts with specifications.		
Possible Cause	1. Between wobble command, only one block is allowed to wobble. 2. G10 L1803 commands with different arguments exceeds the upper limit allowed by a single block, up to 12. 3. Too many G10 L1803 and G10 L1805 commands. 4. The feature as L-argument specified is unsupported for current version (from 10.118.41P, 10.118.49). 5. Any axis belongs to multiple coordinates.		
Solution	1. Remove the exceeding number of wobble moving blocks. 2. Reduce the number of G10 L1803 commands in a single movement block to less than 12. 3. Reduce G10 L1803 and G10 L1805 commands. 4. Update to the version which supports the feature as L-argument specified. 5. Please set Pr701~ of the axis to only belong to a single axis group.		

8.219 BGND-347 This function only supports Syntec axis

Alarm ID	COR-347 BGND-347	Alarm title	【This function only supports Syntec axis】
Description	Only Syntec axis can use this function.		
Possible Cause	The weave function only supports Syntec drives.		
Solution	Please use Syntec driver.		

8.220 BGND-348 APP command error

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error】
Description	There is an error in the APP command.		

Alarm ID	COR–348 BGND-348	Alarm title	【APP command error】
Possible Cause	1. APP name is not a string. 2. The length of APP name is over 20 characters. 3. No APP name is specified.		
Solution	Please make sure that APP commands are used correctly in the NC program.		

8.221 BGND-349 This function only supports Syntec encoder

Alarm ID	COR-349 BGND-349	Alarm title	【This function only supports Syntec magnetic gear or ring gear encoder】
Description	Limited to use with Syntec magnetic gear or ring gear encoders.		
Possible Cause	When complex threading cycle(G78 or G78.2) enables chip removal function, the active spindle must use Syntec magnetic gear or ring gear encoders. In dual-feedback system, it is allowed to use chip removal function when either of the encoders is Syntec magnetic gear or ring gear encoder.		
Solution	1. Set the argument of D to 0. 2. Please use Syntec magnetic gear or ring gear encoders.		

8.222 BGND-350 Invalid EnIP Macro

Alarm ID	COR-350 BGND-350	Alarm title	【Invalid EnIP Macro】
Description	EnIP macro syntax error.		
Possible Cause	The format or range of the input macro arguments is incorrect.		
Solution	Refer to the EnIP macro syntax description.		

8.223 BGND-351 Communication command abnormal

Alarm ID	COR-351 BGND-351	Alarm title	【Communication command abnormal】
Description	1. The execution of the macro communication command failed. 2. The macro communication to driver failed.		
Possible Cause	1. The controller does not support EnIP communication. 2. The EnIP function input request data value range is incorrect. 3. An error occurred while sending or receiving a communication command.		
Solution	1. Please contact the Syntec OEM. 2. Each request data of EnIP represents a 1 value ranging from 0 to 255. 3. This alarm is accompanied by a more detailed error alarm, refer to the alarm description for troubleshooting.		

8.224 BGND-352 Communication command response data is too long

Alarm ID	COR-352 BGND-352	Alarm title	【Communication command response data is too long】
Description	The device return data length is too long.		
Possible Cause	The storage space provided by the user for returning data is insufficient.		
Solution	Make sure that the storage space for the returned data is sufficient.		

8.225 BGND-353 Invalid argument of CHKINF

Alarm ID	COR–353 BGND-353	Alarm title	【Invalid argument of CHKINF】
Description	The type of argument of CHKINF() is incorrect, or the category number is out of range.		
Reason	Programming error.		

Solution	1. Please check the NC program. 2. Check the type of first argument is integer. 3. Check the type of second argument is string. 4. Ensure the category number is in the range, please refer to OpenCNC_Macro Development Manual.

8.226 BGND-354 With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.

Alarm ID	**COR-354** **BGND-354**	**Alarm title**	**【With RTCP enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.】**
Description	With Rotate Tool Center Point function(G43.4, G43.5) enabled, the usage of G10 L1150 to set Five-Axis mechanism parameter is prohibited.		
Reason	The NC file was written incorrectly.		
Solution	Please modify the NC file, cancel Rotate Tool Center Point function(G49), and then use G10 L1150 to set the Five-Axis mechanism parameter.		

8.227 BGND-355 Need geometric axes positioning after tool length compensation changed

Alarm ID	**COR–355** **BGND-355**	**Alarm title**	**Need geometric axes positioning after tool length compensation changed**
Description	Positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) is required after executing tool length compensation commands, if tool radius compensation commands needed.		
Reason	No positioning of the axes on work plane(G17: XY, G18: ZX, G19: YZ) between executing tool length compensation commands and tool radius compensation commands. Tool length compensation commands: H, T(Lathe), G43, G44, G43.4, G43.5, G49, G10 L1050, G10 L1051 Tool radius compensation commands: G41, G42		
Solution	Please modify the NC file. Add positioning command of the axes on work plane(G17: XY, G18: ZX, G19: YZ) after tool length compensation commands and before tool radius compensation commands. Positioning commands: G90 G00, G90 G01, G92		

8.228 BGND-356 Invalid APP command

Alarm ID	COR–356 BGND-356	Alarm title	【Invalid APP command】
Description	There is a syntax error on the Modbus command(G10 L1900/1901/1910/1911).		
Possible Cause	The format or range of the input argument is incorrect.		
Solution	Please refer to the Industry Machine Application Manual.		

8.229 BGND-357 Illegal Modbus packet contents

Alarm ID	COR–357 BGND-357	Alarm title	【Illegal Modbus packet contents】
Description	The contents of the Modbus command packet is illegal.		
Possible Cause	The contents of the customized package is incorrect.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

8.230 BGND-358 Modbus communication timeout

Alarm ID	COR–358 BGND-358	Alarm title	【Modbus communication timeout】
Description	The Modbus communication experienced a timeout.		
Possible Cause	Abnormal communication or unestablished communication causes the connection timeout.		
Solution	Please refer to R5039, as this register provides the error code for troubleshooting.		

8.231 BGND-359 Modbus communication failure

Alarm ID	COR–359 BGND-359	Alarm title	【Modbus communication failure】
Description	The Modbus communication failed.		
Possible Cause	1. There is a communication error or device setting error.		
Solution	1. Please refer to R5039, as this register provides the error code for troubleshooting		

8.232 BGND-361 The number of non-moving blocks exceeds the permissible value under tool radius compensation

Alarm ID	COR-361 BGND-361	Alarm title	【The number of non-moving blocks exceeds the permissible value under tool radius compensation】
Description	The machining program has programmed too many non-moving blocks when tool radius compensation is enable.		
Possible Cause	Programming error.		
Solution	1. Modify the machining program and issue the non-moving blocks when tool radius compensation is disable. 2. Modify the machining program and reduce the number of non-moving blocks when tool radius compensation is enable.		

8.233 BGND-362 Multi-axis multi-signal skip function (G31.10, G31.11) command error

Alarm ID	COR-362 BGND-362	Alarm 标题	【Multi-axis multi-signal skip function (G31.10, G31.11) command error】
说明	Multi-axis multi-signal skip function (G31.10, G31.11) command error		

可能原因	1. Setting command (G31.10) or execution command (G31.11) cannot be issued alone, and other commands cannot be issued between them. Always set first then execute. 2. A set of multi-axis multi-signal skip function is composed of one G31.10 command at least and one G31.11 command. Set first then execute. Both commands should be issued in a set of multi-axis multi-signal skip function, and it is not allowed to insert other commands in between. 3. Supports up to six G31.10 commands in a set of multi-axis multi-signal skip function. 4. Same axis is used in different G31.10 commands in a set of multi-axis multi-signal skip function. E.g., a. Same axis name is used in different G31.10 commands. b. Assign same axis by using virtual axis function in different G31.10 commands. (E.g., Virtual axis Z is corresponding to axis Z1 and Z2. Using axis Z in one G31.10 command, and Z1 in another G31.10 command.) c. Assign same axis by misusing alias of axis. (E.g., There are axis X, Y, Z1, Z2, Z3, Z4, the alias of Z1 is Z.)
排除方法	1. Make sure there is always an execution command (G31.11) behind setting command (G31.10), and there is no execution command (G31.11) left alone. 2. Make sure there are six G31.10 commands at most in each set of multi-axis multi-signal skip function. 3. Make sure there is no axis assigned repeatedly in each set of multi-axis multi-signal skip function.

8.234 BGND-366 The parameters of rotary axis auxiliary brake are set incorrectly

Alarm ID	**COR-366** **BGND-366**	**Alarm Title**	**【The parameters of rotary axis auxiliary brake are set incorrectly】**
Description	Using the rotary axis auxiliary brake, the parameters mapped to the enable group are set incorrectly.		
Possible Cause	Parameters are incorrect. Enable G10 L1500 rotary axis auxiliary brake: 1. The axis ID of the rotary axis auxiliary brake is not specified. 2. The axis type of the specified axis is linear. 3. The M code of the rotary axis auxiliary brake is not specified.		
Solution	Modify parameters Pr3741~Pr3744.		

8.235 BGND-367 Prohibited commands in rotary axis auxiliary brake mode

Alarm ID	COR-367 BGND-367	Alarm Title	【Prohibited commands in rotary axis auxiliary brake mode】
Description	When the rotary axis auxiliary brake is enabled, prohibited commands are used in the nc program.		
Possible Cause	1. The axis, which is in rotary axis auxiliary brake mode, is specified by skip function(G28.1, G31, G31.11). 2. The axis, which is in rotary axis auxiliary brake mode, is specified by chopping function(G81.1, G81.2). 3. Enable the rotary axis auxiliary brake in RTCP mode(G43.4, G43.5). 4. Enable both the rotary axis auxiliary brake mode and the polar coordinate mode(G12.1).		
Solution	Check the NC program.		

8.236 BGND-370 Failed to enable chopping function

Alarm ID	COR-370 BGND-370	Alarm title	【Failed to enable chopping function】
Description	Failed to enable the chopping function .		
Possible Cause	1. G81.2 or G81.1 command repeatedly a. G81.1 command repeatedly. b. G81.2 and G81.1 command simultaneously. 2. Using G81.2 to make 2 axis in the same axis group do advanced chopping simultaneously.		
Solution	Please activate or deactivate the chopping function correctly.		

8.237 BGND-401 Path planning plug-in error

Alarm ID	COR–401 BGND-401	Alarm title	Path planning plug-in error
Description	Path planning plug-in error.		

Reason	Path planning plug-in error leads to system abnormality.
Solution	Please contact OEM Syntec.

8.238 BGND-501 Register not enough, program execution failed

Alarm ID	**BGND-501**	**Alarm title**	**Register not enough, program execution failed**
Description	Insufficient controller remaining registers cause the background execution unit to fail to activate.		
Possible Cause	1. Use too many background execution units in ladder. 2. The system remaining register is too low.		
Solution	Contact machinery manufacturer.		

8.239 BGND-502 Invalid program command

Alarm ID	**BGND-502**	**Alarm title**	**Invalid program command**
Description	Incorrect G code is used in program or use M, T code to call MACRO.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program to confirm that the unsupported G/M/T codes are not used.		

8.240 BGND-503 Illegal G code argument

Alarm ID	**BGND-503**	**Alarm title**	**Illegal G code argument**
Description	Enter incorrect G10 argument format or range in the program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and refer to manual to input correct argument.		

8.241 BGND-504 S code not supported in background execute mode

Alarm ID	BGND-504	Alarm title	S code not supported in background execute mode
Description	S code command is used in program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm S code is not used.		

8.242 BGND-505 F code not supported in background execute mode

Alarm ID	BGND-505	Alarm title	F code not supported in background execute mode
Description	F code command is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program and confirm F code is not used.		

8.243 BGND-506 H code not supported in background execute mode

Alarm ID	BGND-506	Alarm title	H code not supported in background execute mode
Description	H code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm H codes is not used.		

8.244 BGND-507 D code not supported in background execute mode

Alarm ID	BGND-507	Alarm title	D code not supported in background execute mode
Description	D ode is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm D codes is not used.		

8.245 BGND-508 T code not supported in background execute mode

Alarm ID	BGND-508	Alarm title	T code not supported in background execute mode
Description	T code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm T codes is not used.		

8.246 BGND-509 M code not supported in background execute mode

Alarm ID	BGND-509	Alarm title	M code not supported in background execute mode
Description	M code is used in MACRO program.		
Possible Cause	Program editing error.		
Solution	Please check the MACRO program, and confirm M codes is not used.		

8.247 BGND-510 Quantities of Background execute unit exceeds capacity

Alarm ID	BGND-510	Alarm title	【Quantities of Background execute unit exceeds capacity】
Description	The number of enabled BGND components exceeds the upper limit (20), and can not enable more BGND components.		
Possible Cause	Use too many background execute unit in ladder.		
Solution	Contact machinery manufacturer.		

SYNTEC

9 ROT Turret Alarm - ROT

Alarm ID	ROT–001	Alarm title	【Absolute origin setting is not completed】
Description	The ROT function only supports serial bus absolute motors. Therefore, the absolute origin setting of the ROT must be completed before starting the positioning.		
Possible Cause	The ROT origin is not set.		
Solution	After reset the origin, reset to clear the alarm.		

Alarm ID	ROT–002	Alarm title	【Restart the machine and reset the origin due to absolute relevant parameter variations】
Description	The user has modified the relevant important parameters affecting theoriginposition setting, and must power off the controller and the drive, and set the ROT origin.		
Possible Cause	Modify any of the following parameters: position sensor resolution, number of teeth on the side of the screw, and number of teeth on the motor side.		
Solution	Reboot the controller and driver.		

Alarm ID	ROT–003	Alarm title	【Incremental motors are not supported】
Description	The ROT function only supports serial bus absolute motors. If it is non-absolute motor, after enable positioning, the system will enter not ready state.		
Possible Cause	1. Use a non-absolute motor. 2. There is no driver corresponding to the ROT station number.		
Solution	1. Using an absolute motor. 2. Set the correct station number. Reboot the controller and drive to clear alarm.		

Alarm ID	ROT–004	Alarm title	【Target tool numbers are out of range】
Description	Enter an unreasonable target tool number.		

Alarm ID	**ROT–004**	**Alarm title**	**【Target tool numbers are out of range】**
Possible Cause	The target tool number is greater than the total tool number, or less than -2.		
Solution	Set the target tool numbers correctly, and reset to clear the alarm.		

Alarm ID	**ROT–005**	**Alarm title**	**【Driver station numbers do not exist】**
Description	There is no driver corresponding to the ROT station number on the communication network, system enters not ready state.		
Possible Cause	There is no driver corresponding to the ROT station number on the communication network. (Example: set the driver station number to 999)		
Solution	Set the correct ROT station number then reboot the controller and driver.		

Alarm ID	**ROT–006**	**Alarm title**	**【ROT/system axis/serial PLC axis driver station number repetition】**
Description	The ROT axis station number is repeated with other axis station numbers and the system enters a not ready state.		
Possible Cause	The ROT axis station number is repeated with other axis station number settings. Please check if the following axis station number setting is repeated; 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	With correct parameter settings, reboot controller and driver.		

Alarm ID	**ROT–007**	**Alarm title**	**【Registers are located in the system protection area or cannot be bitwise】**
Description	The initial register is not set properly and is located in the system protection area or does not support bit access.		

Alarm ID	ROT–007	Alarm title	【Registers are located in the system protection area or cannot be bitwise】
Possible Cause	1. The data base address R(n) or the command status base address R(m) is set in the CNC system interface area. 2. The command status base address R(m) is set to a register that does not support bit access.		
Solution	Specify the correct initial Register (Set R(m) to between R50~R80 or R256~R511.)		

Alarm ID	ROT–008	Alarm title	【Excessive positioning deviation】
Description	The current ROT feedback angle does not correspond to any of the tools on the ROT. The system determines that the current position deviation of the ROT is too large. Have to perform tool sequence error removal procedure to reposition the ROT to the correct position.		
Possible Cause	After the ROT starts positioning, the positioning action is not completed due to the triggering of the emergency stop, or the controller is turned off and the system detects that the current position deviation is too large. (The difference between the "feedbackangle" and the "positioning angle" exceeds the "position check window angle" setting, which is default is 0.1 degrees). For example: if there are 10 tools on the ROT, the position feedback is 40 degrees after booting, and the positioning angle is 36 * N (N=0~9)The difference is greater than the position check window setting angle (default is 0.1 degrees).		
Solution	1. Complete the tool no. sequence error removal procedure (release the emergency stop and set the mode to manual clockwise/counterclockwise) to release the alarm and enable positioning. 2. Set ROT absolute origin to remove alarm, for versions 10.116.36D, 10.116.38, 10.117.38 and above.		

Alarm ID	ROT-009	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		

Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.

Alarm ID	**ROT–010**	**Alarm title**	**【Acceleration/deceleration or speed setting error】**
Description	Acceleration/deceleration or speed set value is out of range.		
Possible Cause	1. The acceleration/deceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the driver. 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration/deceleration time setting, exceeding the range of driver parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" driver, the parameter setting range is 0~60,000) Acceleration check is considered successful if the following formula is satisfied: M3 Syntec、M3 Yaskawa Sigma7：(Resolution * Screw Gear) / (Acceleration Time * Motor Gear) < 32,212 * 10^(Acceleration Base Unit)。 M3 Yaskawa Sigma5：(Resolution * Screw Gear) / (Acceleration Time * Motor Gear) < 786 * 10^(Acceleration Base Unit)。 Velocity check is considered successful if the following formula is satisfied: M3 Syntec、M3 Yaskawa：(Auto Velocity * Resolution * Screw Gear) / Motor Gear < 32,212,254,705。		

SYNTEC

Alarm ID	ROT–010	Alarm title	【Acceleration/deceleration or speed setting error】
Solution	1. Gradually increase the acceleration/deceleration time until the alarm is removed. 2. Decrease the resolution and increase the electronic gear ratio (Pn-20E) until the alarm is removed. 3. Increase the acceleration base unit (Syntec M3: Pn-86C, Yaskawa M3: Pn-A8C) until the alarm is removed. 4. * When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot controller an driver. 5. Properly reduce the target speed and maximum movement speed to within a reasonable range to remove the alarm. 6. When using the Syntec 2nd generation M2 drivers, set the acceleration/deceleration time between 0 and 60,000. 7. Reset to clear the alarms. Note: Yaskawa Sigma series M2, Pn833=1 are valid only for version 10.116.36R, 10.116.46, and 10.117.46 and above, otherwise, the following parameters modified on the controller are invalid: 1. ROT tool changing acceleration time. 2. Acceleration/deceleration time R(n+3) and R(n+4) of serial PLC axis positioning/MPG mode. 3. Deceleration time R(n+4) of serial PLC axis speed mode.		

Alarm ID	ROT-011	Alarm title	【Driver power abnormality】
Description	The controller detects an error in the power state of the drive (servo is not on or powered off)		
Possible Cause	1. The power supply module of driver voltage unstable. 2. Drive failure.		
Solution	1. Check whether the power supply status of the driver is abnormal. 2. Check whether the power cable of the driver is loose or broken. 3. Replace the driver. 4. After the above driver problems are eliminated, reboot the controller and driver to remove alarm.		

Alarm ID	ROT-012	Alarm title	【This driver type is not supported】
Description	The drive type is not supported by ROT.		

Alarm ID	ROT-012	Alarm title	【This driver type is not supported 】
Possible Cause	The ROT function is supported only for the following drive types. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, * Syntec 2nd gerneration driver. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, * Syntecdriver.		
Solution	Replace the driver with a supported driver type and reboot the controller and driver to remove the alarm. Notes: • The Syntec M3 drivers supports ROT function with version 10.116.38D, 10.116.54B, 10.116.0A, and above. • The Syntec 2nd generation M2 drivers supports ROT with version 10.116.54H, 10.116.0E, and above.		

Alarm ID	ROT-013	Alarm title	【Parameter setting causes positioning abnormality】
Description	The gear ratio and resolution settings may cause the tool magazine positioning function abnormal.		
Possible Cause	When the gear ratio is too large, and the positioning angle is large, the command will be greater than 2,147,483,647 and the overflow. The motor will not operate as expected. For example, the gear ratio is 1440:1, the resolution is 4194304 (set value 1048576), and the pulse number per degree is (1440 * 4194304) / (1 * 360) = 16,777,216, Use the maximum positioning angle of 180 degrees to check, 16777216 * 180 = 3,019,898,880, greater than 2,147,483,647, with overflow the clockwise and counterclockwise fail to positioned at the target position.		
Solution	1. Reduce the driver electronic gear ratio. 2. Reduce the resolution setting on the controller side. For example: the gear ratio is 1440:1, the resolution is 4194304 (set value 1048576), the drive electronic gear numerator is 32, the resolution is reduced to 131072 (set value 32768). The pulse number per degree is (1440 * 131072) / (1 * 360) = 524, 288, and the pulse number of 180 degrees is 94,371,840. There will be no command overflow problem.		

Alarm ID	ROT-014	Alarm title	【Syntec encoder firmware upgrade is over, please restart the drive】
Description	Syntec encoder firmware upgrade is over and the drive needs to be reboot.		

Alarm ID	**ROT-014**	**Alarm title**	**【Syntec encoder firmware upgrade is over, please restart the drive】**
Possible Cause	Perform axial encoder firmware upgrade.		
Solution	Reboot the controller and driver.		

Alarm ID	**ROT-015**	**Alarm title**	**【Syntec encoder information reading timeout】**
Description	Syntec encoder information reading timeout.		
Possible Cause	1. The communication between the controller and the driver is abnormal. 2. The driver status is abnormal.		
Solution	1. Confirm whether the communication cable is loose. 2. Reboot controller and driver.		

Alarm ID	**ROT-017**	**Alarm title**	**【User-defined tool position setting error】**
Description	When using user-defined tool position, the tool position table is set incorrectly.		
Possible Cause	The user-defined tool position is not set in ascending order.		
Solution	1. Set the user-defined tool position in ascending order. 2. Check the parameter "Total tool number" that it is matched with the user-defined tool position table. 3. Reset ROT axis or fix the setting should be able to clear this alarm.		

Alarm ID	**ROT-018**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

Alarm ID	ROT-019	Alarm title	The hardware doesn't receive communication packet
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

Alarm ID	ROT-020	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

Alarm ID	ROT-021	Alarm title	Driver software misses communication packet
Description	Driveresponse packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

9.1 ROT–001 Absolute origin setting is not completed

Alarm ID	ROT–001	Alarm title	【Absolute origin setting is not completed】
Description	The ROT function only supports serial bus absolute motors. Therefore, the absolute origin setting of the ROT must be completed before starting the positioning.		
Possible Cause	The ROT origin is not set.		
Solution	After reset the origin, reset to clear the alarm.		

9.2 ROT–002 Restart the machine and reset the origin due to absolute relevant parameter variations

Alarm ID	ROT–002	Alarm title	【Restart the machine and reset the origin due to absolute relevant parameter variations】
Description	The user has modified the relevant important parameters affecting theoriginposition setting, and must power off the controller and the drive, and set the ROT origin.		
Possible Cause	Modify any of the following parameters: position sensor resolution, number of teeth on the side of the screw, and number of teeth on the motor side.		
Solution	Reboot the controller and driver.		

9.3 ROT–003 Incremental motors are not supported

Alarm ID	ROT–003	Alarm title	【Incremental motors are not supported】
Description	The ROT function only supports serial bus absolute motors. If it is non-absolute motor, after enable positioning, the system will enter not ready state.		
Possible Cause	1. Use a non-absolute motor. 2. There is no driver corresponding to the ROT station number.		

Alarm ID	ROT–003	Alarm title	【Incremental motors are not supported】
Solution	1. Using an absolute motor. 2. Set the correct station number. Reboot the controller and drive to clear alarm.		

9.4 ROT–004 Target tool numbers are out of range

Alarm ID	ROT–004	Alarm title	【Target tool numbers are out of range】
Description	Enter an unreasonable target tool number.		
Possible Cause	The target tool number is greater than the total tool number, or less than -2.		
Solution	Set the target tool numbers correctly, and reset to clear the alarm.		

9.5 ROT–005 Driver station numbers do not exist

Alarm ID	ROT–005	Alarm title	【Driver station numbers do not exist】
Description	There is no driver corresponding to the ROT station number on the communication network, system enters not ready state.		
Possible Cause	There is no driver corresponding to the ROT station number on the communication network. (Example: set the driver station number to 999)		
Solution	Set the correct ROT station number then reboot the controller and driver.		

9.6 ROT–006 ROT/system axis/serial PLC axis driver station number repetition

Alarm ID	ROT–006	Alarm title	【ROT/system axis/serial PLC axis driver station number repetition】
Description	The ROT axis station number is repeated with other axis station numbers and the system enters a not ready state.		

Alarm ID	ROT–006	Alarm title	【ROT/system axis/serial PLC axis driver station number repetition】
Possible Cause	The ROT axis station number is repeated with other axis station number settings. Please check if the following axis station number setting is repeated; 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	With correct parameter settings, reboot controller and driver.		

9.7 ROT–007 Registers are located in the system protection area or cannot be bitwise

Alarm ID	ROT–007	Alarm title	【Registers are located in the system protection area or cannot be bitwise】
Description	The initial register is not set properly and is located in the system protection area or does not support bit access.		
Possible Cause	1. The data base address R(n) or the command status base address R(m) is set in the CNC system interface area. 2. The command status base address R(m) is set to a register that does not support bit access.		
Solution	Specify the correct initial Register (Set R(m) to between R50~R80 or R256~R511.)		

9.8 ROT–008 Excessive positioning deviation

Alarm ID	ROT–008	Alarm title	【Excessive positioning deviation】
Description	The current ROT feedback angle does not correspond to any of the tools on the ROT. The system determines that the current position deviation of the ROT is too large. Have to perform tool sequence error removal procedure to reposition the ROT to the correct position.		

Alarm ID	ROT–008	Alarm title	【Excessive positioning deviation 】
Possible Cause	After the ROT starts positioning, the positioning action is not completed due to the triggering of the emergency stop, or the controller is turned off and the system detects that the current position deviation is too large. (The difference between the "feedbackangle" and the "positioning angle" exceeds the "position check window angle" setting, which is default is 0.1 degrees). For example: if there are 10 tools on the ROT, the position feedback is 40 degrees after booting, and the positioning angle is 36 * N (N=0~9)The difference is greater than the position check window setting angle (default is 0.1 degrees).		
Solution	1. Complete the tool no. sequence error removal procedure (release the emergency stop and set the mode to manual clockwise/counterclockwise) to release the alarm and enable positioning. 2. Set ROT absolute origin to remove alarm, for versions 10.116.36D, 10.116.38, 10.117.38 and above.		

9.9 ROT–009 Drive communication error

Alarm ID	ROT-009	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

9.10 ROT–010 Acceleration/deceleration or speed setting error

Alarm ID	ROT–010	Alarm title	【Acceleration/deceleration or speed setting error】
Description	Acceleration/deceleration or speed set value is out of range.		
Possible Cause	1. The acceleration/deceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the driver. 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration/deceleration time setting, exceeding the range of driver parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" driver, the parameter setting range is 0~60,000) Acceleration check is considered successful if the following formula is satisfied: M3 Syntec、M3 Yaskawa Sigma7：(Resolution * Screw Gear) / (Acceleration Time * Motor Gear) < 32,212 * 10^(Acceleration Base Unit)。 M3 Yaskawa Sigma5：(Resolution * Screw Gear) / (Acceleration Time * Motor Gear) < 786 * 10^(Acceleration Base Unit)。 Velocity check is considered successful if the following formula is satisfied: M3 Syntec、M3 Yaskawa：(Auto Velocity * Resolution * Screw Gear) / Motor Gear < 32,212,254,705。		

SYNTEC

Alarm ID	ROT–010	Alarm title	【Acceleration/deceleration or speed setting error】
Solution	1. Gradually increase the acceleration/deceleration time until the alarm is removed. 2. Decrease the resolution and increase the electronic gear ratio (Pn-20E) until the alarm is removed. 3. Increase the acceleration base unit (Syntec M3: Pn-86C, Yaskawa M3: Pn-A8C) until the alarm is removed. 4. * When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot controller an driver. 5. Properly reduce the target speed and maximum movement speed to within a reasonable range to remove the alarm. 6. When using the Syntec 2nd generation M2 drivers, set the acceleration/deceleration time between 0 and 60,000. 7. Reset to clear the alarms. Note: Yaskawa Sigma series M2, Pn833=1 are valid only for version 10.116.36R, 10.116.46, and 10.117.46 and above, otherwise, the following parameters modified on the controller are invalid: 1. ROT tool changing acceleration time. 2. Acceleration/deceleration time R(n+3) and R(n+4) of serial PLC axis positioning/MPG mode. 3. Deceleration time R(n+4) of serial PLC axis speed mode.		

9.11 ROT-011 Driver power abnormality

Alarm ID	ROT-011	Alarm title	【Driver power abnormality】
Description	The controller detects an error in the power state of the drive (servo is not on or powered off)		
Possible Cause	1. The power supply module of driver voltage unstable. 2. Drive failure.		
Solution	1. Check whether the power supply status of the driver is abnormal. 2. Check whether the power cable of the driver is loose or broken. 3. Replace the driver. 4. After the above driver problems are eliminated, reboot the controller and driver to remove alarm.		

9.12 ROT-012 This driver type is not supported

Alarm ID	ROT-012	Alarm title	【This driver type is not supported 】
Description	The drive type is not supported by ROT.		
Possible Cause	The ROT function is supported only for the following drive types. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, * Syntec 2nd gerneration driver. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, * Syntecdriver.		
Solution	Replace the driver with a supported driver type and reboot the controller and driver to remove the alarm. Notes: • The Syntec M3 drivers supports ROT function with version 10.116.38D, 10.116.54B, 10.116.0A, and above. • The Syntec 2nd generation M2 drivers supports ROT with version 10.116.54H, 10.116.0E, and above.		

9.13 ROT-013 Parameter setting causes positioning abnormality

Alarm ID	ROT-013	Alarm title	【Parameter setting causes positioning abnormality】
Description	The gear ratio and resolution settings may cause the tool magazine positioning function abnormal.		
Possible Cause	When the gear ratio is too large, and the positioning angle is large, the command will be greater than 2,147,483,647 and the overflow. The motor will not operate as expected. For example, the gear ratio is 1440:1, the resolution is 4194304 (set value 1048576), and the pulse number per degree is (1440 * 4194304) / (1 * 360) = 16,777,216, Use the maximum positioning angle of 180 degrees to check, 16777216 * 180 = 3,019,898,880, greater than 2,147,483,647, with overflow the clockwise and counterclockwise fail to positioned at the target position.		

Alarm ID	ROT-013	Alarm title	【Parameter setting causes positioning abnormality】
Solution	1. Reduce the driver electronic gear ratio. 2. Reduce the resolution setting on the controller side. For example: the gear ratio is 1440:1, the resolution is 4194304 (set value 1048576), the drive electronic gear numerator is 32, the resolution is reduced to 131072 (set value 32768). The pulse number per degree is (1440 * 131072) / (1 * 360) = 524, 288, and the pulse number of 180 degrees is 94,371,840. There will be no command overflow problem.		

9.14 ROT-014 Syntec encoder firmware upgrade is over, please restart the drive

Alarm ID	ROT-014	Alarm title	【Syntec encoder firmware upgrade is over, please restart the drive】
Description	Syntec encoder firmware upgrade is over and the drive needs to be reboot.		
Possible Cause	Perform axial encoder firmware upgrade.		
Solution	Reboot the controller and driver.		

9.15 ROT-015 Syntec encoder information reading timeout

Alarm ID	ROT-015	Alarm title	【Syntec encoder information reading timeout】
Description	Syntec encoder information reading timeout.		
Possible Cause	1. The communication between the controller and the driver is abnormal. 2. The driver status is abnormal.		
Solution	1. Confirm whether the communication cable is loose. 2. Reboot controller and driver.		

9.16 ROT-017 User-defined tool position setting error

Alarm ID	ROT-017	Alarm title	【User-defined tool position setting error】
Description	When using user-defined tool position, the tool position table is set incorrectly.		
Possible Cause	The user-defined tool position is not set in ascending order.		
Solution	1. Set the user-defined tool position in ascending order. 2. Check the parameter "Total tool number" that it is matched with the user-defined tool position table. 3. Reset ROT axis or fix the setting should be able to clear this alarm.		

9.17 ROT-018 Poor contact of communication wire

Alarm ID	ROT-018	Alarm title	Poor contact of communication wire
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

9.18 ROT-019 The hardware doesn't receive communication packet

Alarm ID	ROT-019	Alarm title	The hardware doesn't receive communication packet
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		

Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.

9.19 ROT-020 The communication packet is disturbed by noise

Alarm ID	**ROT-020**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

9.20 ROT-021 Driver software misses communication packet

Alarm ID	**ROT-021**	**Alarm title**	**Driver software misses communication packet**
Description	Driveresponse packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

10 ATC Alarm - ATC

Alarm ID	**ATC-001**	**Alarm Title**	**ATC axis driver station number is repeated with other axis**
Description	ATC axis station number is repeated with other axis station number, and the system enters Not Ready state.		
Possible Cause	ATC axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	After correctly set the station number, reboot the controller.		

Alarm ID	**ATC-002**	**Alarm Title**	**Driver station number not exist**
Description	The ATC station number corresponded driver does not exist on the communication network, and the system enters Not Ready state.		
Possible Cause	1. There is no driver corresponding to the ATC station number on the communication network. 2. Use pulse axis or virtual axis.		
Solution	After setting the correct ATC station number, reboot controller and driver.		

Alarm ID	**ATC-003**	**Alarm Title**	**【Register is in the system protected area or non-bita**
Description	The start register is set incorrectly, which is in the system protected area or bit access is not supported.		
Possible Cause	1. The data base address R(n) or command status base address R(m) is set in the CNC system interface 2. The command status base address R(m) is set in a register that does not support bit access.		
Solution	1. Check the target start registers (Rn)~(Rn+2), do not repeat with the system registers. The recommen 2. Check the command and status start registers (Rm)~(Rm+1). Do not repeat the system registers, and recommended interval is [50,80]. 3. After set correctly, reboot the controller and can remove the alarm.		

Alarm ID	**ATC-004**	**Alarm Title**	**incremental motor is not supported**

Description	ATC axis not support incremental motor, only support absolute motor.
Possible Cause	1. Incremental motor is used. 2. Use absolute motor, but driver parameter is set as incremental type.
Solution	1. Change to absolute motor. 2. Modify the driver parameter to absolute, and reboot driver. 3. After troubleshoot completed, reboot controller and can remove alarm.

Alarm ID	**ATC-005**	**Alarm Title**	**Absolute home position is not set**
Description	ATC function only supports serial bus absolute motors. Therefore, the absolute home position setting of must be completed before cycle start.		
Possible Cause	ATC axis home position is not set.		
Solution	Use MPG mode to turn the ATC axis to the position you want to set as home (zero degree angle), set Rm.6 home position, and can remove the alarm.		

Alarm ID	**ATC-006**	**Alarm Title**	**Home related parameters changed, please reboot and**
Description	The user has modified the important parameters that affect the home position setting. Need to reboot co driver and re-set ATC axis home.		
Possible Cause	修改以下任一参数：位置感应器分辨率、螺杆侧齿数、电机侧齿数、电机运动方向。 Modify any of the following parameters: the position sensor resolution, the number of teeth on the ballsc number of teeth on the motor side, motor movement direction.		
Solution	Reboot controller and driver.		

Alarm ID	**ATC-007**	**Alarm Title**	**Motion plan calculation error**
Description	Controller cannot calculate the motion plan track.		
Possible Cause	1. The tool clamp or unclamp range set by the parameter is too small. 2. The advance time of tool clamp or unclamp interval set by the parameter is too large, causing the tool pre-unclamp or pre-clamp position exceed the previous interval.		
Solution	1. Please set a reasonable tool clamp and unclamp interval angle according to the specifications of the cam. 2. Please set the advance time of tool clamp or unclamp smaller.		

Alarm ID	ATC-008	Alarm Title	Angle setting error
Description	The angle of each interval of the ATC axis is set incorrectly.		
Possible Cause	The angle-related setting is wrong, and not following the process of first tool unclamp then clamp, to set the angle from small to large.		
Solution	1. Please follow the first tool unclamp then clamp process to set the angle. 2. The angle needs to be from small to large.		

Alarm ID	ATC-009	Alarm Title	Tool clamp or unclamp signal timeout not triggered
Description	When the ATC axis enters the tool clamp or unclamp interval system starts to check corresponding signal. It is necessary to use this inspection mechanism to confirm the completion of the tool clamp and unclamp action, to avoid the machine mechanism crash.		
Possible Cause	After the ATC axis enters the 70% position of tool clamp and unclamp interval, the corresponding check signal has not been triggered (not On) when set waiting time is exceed.		
Solution	1. Check PLC and confirm that the tool clamp and unclamp check signals are sent to Rm.8 and Rm.9. 2. Check the tool clamp and unclamp sensors and wiring, and confirm whether the signal is sent normally. 3. Check the mechanism, and confirm whether the ATC axis is performing the tool clamp and unclamp action normally.		

Alarm ID	ATC-010	Alarm Title	When activating not at zero-degree angle
Description	When the ATC axis is activated, it must be in the check window of zero-degree angle, otherwise alarm will be triggered.		
Possible Cause	1. The position check window is set too small. 2. Unexpected ATC axial movement. 3. ATC moves axially after power failure.		
Solution	1. Please adjust the zero-degree position check window within a reasonable range. 2. Please check whether the ATC activating process is correct in according to the current angle (Rn+1), and eliminate the reason that the angle is not at zero degrees. 3. Use MPG mode to turn the ATC axis back to zero degrees. 4. Reset can remove the alarm.		

Alarm ID	ATC-011	Alarm Title	Exceed the tool clamp and unclamp angle when activating
Description	The ATC axis exceed the tool clamp and unclamp angle when activating, and there may be a risk of damaging the mechanism.		
Possible Cause	1. ATC axis has unexpected movement. 2. ATC axis has moved after power off.		
Solution	1. Use MPG mode to rotate ATC axis back to zero-degree angle. 2. Reset can remove the alarm.		

Alarm ID	ATC-012	Alarm Title	ATC axis driver station number is repeated with other axis
Description	When the ATC axis is activated, the tool axis must be within the check window of the check position, otherwise alarm will be issued.		
Possible Cause	1. The ATC tool change is performed before the tool axis has moved to the check position. 2. The tool axis has reached the check position, but the axis position check window (Pr481~) is set too small.		
Solution	1. If you do not need to check the tool axis, set the tool axis number to 0 (off). 2. Before cycle start (Rm.0 is set to On), make sure the tool axis has moved to the set check position (machine coordinate). 3. Please properly adjust the tool axis position check window (Pr481~) within a reasonable range.		

Alarm ID	ATC-013	Alarm Title	Tool axis not exist
Description	The set tool axis is not exist.		
Possible Cause	The set tool axis is not activated.		
Solution	1. Please check whether the tool axis corresponded ATC parameter "Station ID" is activated. 2. Reboot controller.		

Alarm ID	ATC-014	Alarm title	Drive communication error

Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.

Alarm ID	**ATC-015**	**Alarm Title**	**Driver parameter write failed**
Description	When ATC axis failed to write parameter into driver.		
Possible Cause	The communication between the controller and the drive is unstable, causing the drive parameter write timeout.		
Solution	After eliminating the cause of communication error, reboot the controller and driver.		

Alarm ID	**ATC-016**	**Alarm Title**	**Drive power abnormal**
Description	The controller detected the driver power status error (the servo off or is powered off).		
Possible Cause	1. The driver power-amp back power supply is unstable. 2. Drive failure.		
Solution	1. Check if the driver power supply status is normal. 2. Check if the drive power cable is loose or damaged. 3. Replace the drive. 4. After the above driver problems are eliminated, reboot the controller and drive, then can remove the alarm.		

Alarm ID	**ATC-017**	**Alarm Title**	**Syntec encoder firmware update is over, please reboot the driver**
Description	The firmware update of the Syntec encoder is over, and the driver needs to be rebooted.		

Possible Cause	Perform an axial encoder firmware upgrade.
Solution	Reboot controller and driver.

Alarm ID	**ATC-018**	**Alarm Title**	**Syntec encoder information read timeout**
Description	Syntec encoder information read timeout		
Possible Cause	1. The communication between the controller and the driver is abnormal. 2. The drive status is abnormal.		
Solution	1. Check if the communication cable is loose. 2. Reboot the controller and driver.		

Alarm ID	**ATC-019**	**Alarm Title**	**Can't find HomeDog**
Description	After the controller reached the HomeDog, it cannot find the HomeDog after moving in the reverse direction.		
Possible Cause	1. HomeDog poor contact. 2. noise interference.		
Solution	1. Check if the HomeDog signal is interfered by noise. 2. Check if the HomeDog contact is well. 3. Reset can remove the alarm. **Advanced description** The system first moves towards the direction of HomeDog, after the HomeDog is reached, then decelerate to stop with planned deceleration command. If the deceleration distance is too long when stopping, causing the machine to run over stop block, when the system continues to move in the reverse direction, system need to wait for the HomeDog signal to again appear and disappear before starting the home setting. If the HomeDog signal does not appear when moving in the reverse direction, when "distance moved 2.5 times of acceleration/deceleration time at home searching speed" is exceeded, this alarm will be issued.		

Alarm ID	**ATC-020**	**Alarm Title**	**Home searching zero speed check failed**
Description	The HomeDog was reached during home searching but the motor could not stop completely.		

Possible Cause	1. The driver gain setting is not good, causing the motor shaking. 2. Resonance occurs when the motor is running.
Solution	1. Check the position loop gain and speed loop gain setting of the driver. 2. Activate the drive resonance frequency suppression function. 3. Reset can remove the alarm. 4. If it cannot be resolved, please contact Syntec OEM. **Advanced description** When home searching, machine will move towards the HomeDog at the home searching speed until it stops after hitting the HomeDog, and then find the HomeDog at the speed of 5 (rpm) in the opposite direction. When the machine starts to decelerate when it reached the HomeDog at the home searching speed, if 0.1 seconds after the deceleration command is send, the machine still cannot decelerate to less than the zero speed check window, this alarm will be issued.

Alarm ID	**ATC-021**	**Alarm Title**	**Can't leave HomeDog**
Description	When home searching, after stop and return, axis has moved more than 20 degrees and still can't leave HomeDog.		
Possible Cause	1. Home Dog poor contact. 2. noise interference.		
Solution	1. Check if the HomeDog signal is interfered by noise. 2. Check if the HomeDog is not in good contact. 3. Reset can remove the alarm. **Advanced description** When home searching, machine will move towards the HomeDog at the home searching speed until it stops after hitting the HomeDog, and then find the HomeDog at the speed of 5 (rpm) in the opposite direction. The controller calculates based on the sensor resolution. If the motor moves more than 20 degrees and still not leaving the home signal, this alarm will be issued.		

Alarm ID	**ATC-023**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		

Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		
Alarm ID	**ATC-024**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		
Alarm ID	**ATC-025**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		
Alarm ID	**ATC-026**	**Alarm title**	**Driver software misses communication packet**
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

10.1 ATC-001 ATC axis driver station number is repeated with other axis

Alarm ID	ATC-001	Alarm Title	ATC axis driver station number is repeated with other axis
Description	ATC axis station number is repeated with other axis station number, and the system enters Not Ready state.		
Possible Cause	ATC axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	After correctly set the station number, reboot the controller.		

10.2 ATC-002 Driver station number not exist

Alarm ID	ATC-002	Alarm Title	Driver station number not exist
Description	The ATC station number corresponded driver does not exist on the communication network, and the system enters Not Ready state.		
Possible Cause	1. There is no driver corresponding to the ATC station number on the communication network. 2. Use pulse axis or virtual axis.		
Solution	After setting the correct ATC station number, reboot controller and driver.		

10.3 ATC-003 Register is in the system protected area or non-bitable

Alarm ID	ATC-003	Alarm Title	【Register is in the system protected area or non-bita
Description	The start register is set incorrectly, which is in the system protected area or bit access is not supported.		
Possible Cause	1. The data base address R(n) or command status base address R(m) is set in the CNC system interface 2. The command status base address R(m) is set in a register that does not support bit access.		

Solution	1. Check the target start registers (Rn)~(Rn+2), do not repeat with the system registers. The recommen 2. Check the command and status start registers (Rm)~(Rm+1). Do not repeat the system registers, and recommended interval is [50,80]. 3. After set correctly, reboot the controller and can remove the alarm.

10.4 ATC-004 Incremental motor is not supported

Alarm ID	**ATC-004**	**Alarm Title**	**incremental motor is not supported**
Description	ATC axis not support incremental motor, only support absolute motor.		
Possible Cause	1. Incremental motor is used. 2. Use absolute motor, but driver parameter is set as incremental type.		
Solution	1. Change to absolute motor. 2. Modify the driver parameter to absolute, and reboot driver. 3. After troubleshoot completed, reboot controller and can remove alarm.		

10.5 ATC-005 Absolute home position is not set

Alarm ID	**ATC-005**	**Alarm Title**	**Absolute home position is not set**
Description	ATC function only supports serial bus absolute motors. Therefore, the absolute home position setting of must be completed before cycle start.		
Possible Cause	ATC axis home position is not set.		
Solution	Use MPG mode to turn the ATC axis to the position you want to set as home (zero degree angle), set Rm.6 home position, and can remove the alarm.		

10.6 ATC-006 Home related parameters changed, please reboot and reset home

Alarm ID	**ATC-006**	**Alarm Title**	**Home related parameters changed, please reboot and**
Description	The user has modified the important parameters that affect the home position setting. Need to reboot co driver and re-set ATC axis home.		

Possible Cause	修改以下任一参数：位置感应器分辨率、螺杆侧齿数、电机侧齿数、电机运动方向。 Modify any of the following parameters: the position sensor resolution, the number of teeth on the ballsc number of teeth on the motor side, motor movement direction.
Solution	Reboot controller and driver.

10.7 ATC-007 Motion plan calculation error

Alarm ID	**ATC-007**	**Alarm Title**	**Motion plan calculation error**
Description	Controller cannot calculate the motion plan track.		
Possible Cause	1. The tool clamp or unclamp range set by the parameter is too small. 2. The advance time of tool clamp or unclamp interval set by the parameter is too large, causing the tool pre-unclamp or pre-clamp position exceed the previous interval.		
Solution	1. Please set a reasonable tool clamp and unclamp interval angle according to the specifications of the cam. 2. Please set the advance time of tool clamp or unclamp smaller.		

10.8 ATC-008 Angle setting error

Alarm ID	**ATC-008**	**Alarm Title**	**Angle setting error**
Description	The angle of each interval of the ATC axis is set incorrectly.		
Possible Cause	The angle-related setting is wrong, and not following the process of first tool unclamp then clamp, to set the angle from small to large.		
Solution	1. Please follow the first tool unclamp then clamp process to set the angle. 2. The angle needs to be from small to large.		

10.9 ATC-009 Tool clamp or unclamp signal timeout not triggered

Alarm ID	**ATC-009**	**Alarm Title**	**Tool clamp or unclamp signal timeout not triggered**

Description	When the ATC axis enters the tool clamp or unclamp interval system starts to check corresponding signal. It is necessary to use this inspection mechanism to confirm the completion of the tool clamp and unclamp action, to avoid the machine mechanism crash.
Possible Cause	After the ATC axis enters the 70% position of tool clamp and unclamp interval, the corresponding check signal has not been triggered (not On) when set waiting time is exceed.
Solution	1. Check PLC and confirm that the tool clamp and unclamp check signals are sent to Rm.8 and Rm.9. 2. Check the tool clamp and unclamp sensors and wiring, and confirm whether the signal is sent normally. 3. Check the mechanism, and confirm whether the ATC axis is performing the tool clamp and unclamp action normally.

10.10 ATC-010 When activating not at zero-degree angle

Alarm ID	**ATC-010**	**Alarm Title**	**When activating not at zero-degree angle**
Description	When the ATC axis is activated, it must be in the check window of zero-degree angle, otherwise alarm will be triggered.		
Possible Cause	1. The position check window is set too small. 2. Unexpected ATC axial movement. 3. ATC moves axially after power failure.		
Solution	1. Please adjust the zero-degree position check window within a reasonable range. 2. Please check whether the ATC activating process is correct in according to the current angle (Rn+1), and eliminate the reason that the angle is not at zero degrees. 3. Use MPG mode to turn the ATC axis back to zero degrees. 4. Reset can remove the alarm.		

10.11 ATC-011 Exceed the tool clamp and unclamp angle when activating

Alarm ID	**ATC-011**	**Alarm Title**	**Exceed the tool clamp and unclamp angle when activating**
Description	The ATC axis exceed the tool clamp and unclamp angle when activating, and there may be a risk of damaging the mechanism.		

Possible Cause	1. ATC axis has unexpected movement. 2. ATC axis has moved after power off.
Solution	1. Use MPG mode to rotate ATC axis back to zero-degree angle. 2. Reset can remove the alarm.

10.12 ATC-012 Tool axis not moved to check position

Alarm ID	**ATC-012**	**Alarm Title**	**ATC axis driver station number is repeated with other axis**
Description	When the ATC axis is activated, the tool axis must be within the check window of the check position, otherwise alarm will be issued.		
Possible Cause	1. The ATC tool change is performed before the tool axis has moved to the check position. 2. The tool axis has reached the check position, but the axis position check window (Pr481~) is set too small.		
Solution	1. If you do not need to check the tool axis, set the tool axis number to 0 (off). 2. Before cycle start (Rm.0 is set to On), make sure the tool axis has moved to the set check position (machine coordinate). 3. Please properly adjust the tool axis position check window (Pr481~) within a reasonable range.		

10.13 ATC-013 Tool axis not exist

Alarm ID	**ATC-013**	**Alarm Title**	**Tool axis not exist**
Description	The set tool axis is not exist.		
Possible Cause	The set tool axis is not activated.		
Solution	1. Please check whether the tool axis corresponded ATC parameter "Station ID" is activated. 2. Reboot controller.		

10.14 ATC-014 Drive communication error

Alarm ID	**ATC-014**	**Alarm title**	**Drive communication error**

Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.

10.15 ATC-015 Driver parameter write failed

Alarm ID	**ATC-015**	**Alarm Title**	**Driver parameter write failed**
Description	When ATC axis failed to write parameter into driver.		
Possible Cause	The communication between the controller and the drive is unstable, causing the drive parameter write timeout.		
Solution	After eliminating the cause of communication error, reboot the controller and driver.		

10.16 ATC-016 Drive power abnormal

Alarm ID	**ATC-016**	**Alarm Title**	**Drive power abnormal**
Description	The controller detected the driver power status error (the servo off or is powered off).		
Possible Cause	1. The driver power-amp back power supply is unstable. 2. Drive failure.		
Solution	1. Check if the driver power supply status is normal. 2. Check if the drive power cable is loose or damaged. 3. Replace the drive. 4. After the above driver problems are eliminated, reboot the controller and drive, then can remove the alarm.		

10.17 ATC-017 Syntec encoder firmware update is over, please reboot the driver

Alarm ID	ATC-017	Alarm Title	Syntec encoder firmware update is over, please reboot the driver
Description	The firmware update of the Syntec encoder is over, and the driver needs to be rebooted.		
Possible Cause	Perform an axial encoder firmware upgrade.		
Solution	Reboot controller and driver.		

10.18 ATC-018 Syntec encoder information read timeout

Alarm ID	ATC-018	Alarm Title	Syntec encoder information read timeout
Description	Syntec encoder information read timeout		
Possible Cause	1. The communication between the controller and the driver is abnormal. 2. The drive status is abnormal.		
Solution	1. Check if the communication cable is loose. 2. Reboot the controller and driver.		

10.19 ATC-019 Can't find HomeDog

Alarm ID	ATC-019	Alarm Title	Can't find HomeDog
Description	After the controller reached the HomeDog, it cannot find the HomeDog after moving in the reverse direction.		
Possible Cause	1. HomeDog poor contact. 2. noise interference.		

Solution	1. Check if the HomeDog signal is interfered by noise. 2. Check if the HomeDog contact is well. 3. Reset can remove the alarm. **Advanced description** The system first moves towards the direction of HomeDog, after the HomeDog is reached, then decelerate to stop with planned deceleration command. If the deceleration distance is too long when stopping, causing the machine to run over stop block, when the system continues to move in the reverse direction, system need to wait for the HomeDog signal to again appear and disappear before starting the home setting. If the HomeDog signal does not appear when moving in the reverse direction, when "distance moved 2.5 times of acceleration/deceleration time at home searching speed" is exceeded, this alarm will be issued.

10.20 ATC-020 Home searching zero speed check failed

Alarm ID	**ATC-020**	**Alarm Title**	**Home searching zero speed check failed**
Description	The HomeDog was reached during home searching but the motor could not stop completely.		
Possible Cause	1. The driver gain setting is not good, causing the motor shaking. 2. Resonance occurs when the motor is running.		
Solution	1. Check the position loop gain and speed loop gain setting of the driver. 2. Activate the drive resonance frequency suppression function. 3. Reset can remove the alarm. 4. If it cannot be resolved, please contact Syntec OEM. **Advanced description** When home searching, machine will move towards the HomeDog at the home searching speed until it stops after hitting the HomeDog, and then find the HomeDog at the speed of 5 (rpm) in the opposite direction. When the machine starts to decelerate when it reached the HomeDog at the home searching speed, if 0.1 seconds after the deceleration command is send, the machine still cannot decelerate to less than the zero speed check window, this alarm will be issued.		

10.21 ATC-021 Can't leave HomeDog

Alarm ID	**ATC-021**	**Alarm Title**	**Can't leave HomeDog**

Description	When home searching, after stop and return, axis has moved more than 20 degrees and still can't leave HomeDog.
Possible Cause	1. Home Dog poor contact. 2. noise interference.
Solution	1. Check if the HomeDog signal is interfered by noise. 2. Check if the HomeDog is not in good contact. 3. Reset can remove the alarm. **Advanced description** When home searching, machine will move towards the HomeDog at the home searching speed until it stops after hitting the HomeDog, and then find the HomeDog at the speed of 5 (rpm) in the opposite direction. The controller calculates based on the sensor resolution. If the motor moves more than 20 degrees and still not leaving the home signal, this alarm will be issued.

10.22 ATC-023 Poor contact of communication wire

Alarm ID	**ATC-023**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

10.23 ATC-024 The hardware doesn't receive communication packet

Alarm ID	**ATC-024**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		

Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.

10.24 ATC-025 The communication packet is disturbed by noise

Alarm ID	**ATC-025**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

10.25 ATC-026 Driver software misses communication packet

Alarm ID	**ATC-026**	**Alarm title**	**Driver software misses communication packet**
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

11 Serial PLC Axis Alarm – SERIALPLCAXIS

Alarm ID	**SERIALPLCAXIS-001**	**Alarm title**	**【ROT/system axis/serial PLC axis driver station number repetition】**
Description	The serial PLC axis station number is repeated with other axis station numbers, and the system enters the not ready state.		
Reason	The serial PLC station number setting to repeated with other axis. Please check if the following axis station number settings are repeated: 1. System axis. 2. Serial PLC axis. 3. ROT axis. 4. ATC axis. 5. Setpoint axis		
Solution	With correct parameter settings, reboot controller and driver to remove the alarm.		

Alarm ID	**SERIALPLCAXIS-002**	**Alarm title**	**【R value registers are located in the system protection area or cannot be bitwise】**
Description	The initial register is not set properly and is located in the system protection area or does not support bit access.		
Reason	1. The data base address R(n) or the command status base address R(m) is set in the CNC system interface area. 2. The command status base address R(m) is set to a register that does not support bit access.		
Solution	Specify the initial R register correctly (Set Rm to between R50~R80 or R256~R511.)		

Alarm ID	**SERIALPLCAXIS-003**	**Alarm title**	**【Acceleration/deceleration or speed setting error】**
Description	Acceleration/deceleration or speed set value is out of range.		

Reason	1. The acceleration/deceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the driver. (The acceleration/deceleration is based on the pulse wave per square sec. of resolution, gear ratio, and the screw width, if it is the "Yakawa" drive, it must fall within the range of 1~655,350,000.) 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration/deceleration time setting, exceeding the range of driver parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" driver, the parameter setting range is 0~60,000)
Solution	1. Properly enlarge acceleration/deceleration time to within a reasonable range, can remove alarm, 2. * When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot controller an driver. 3. Properly reduce the target speed and maximum movement speed to within a reasonable range to remove the alarm. 4. When using the Syntec 2nd generation M2 drivers, set the acceleration/deceleration time between 0 and 60,000. 5. Reset to clear the alarms. Note: Yaskawa Sigma series M2, Pn833=1 are valid only for version 10.116.36R, 10.116.46, and 10.117.46 and above, otherwise, the following parameters modified on the controller are invalid: 1. ROT tool changing acceleration time. 2. Acceleration/deceleration time R(n+3) and R(n+4) of serial PLC axis positioning/MPG mode. 3. Deceleration time R(n+4) of serial PLC axis speed mode.

Alarm ID	**SERIALPLCAXIS-004**	**Alarm title**	**【Inspection window setting error】**
Description	The serial PLC axis position and speed checking window settings are incorrect. Please re-specify.		
Reason	The serial PLC axis position and speed check window settings are out of range. (The number of pulse converted by the position check window according to resolution, gear ratio, and ballscrew width must fall within the range of 0~1,073,741,824) (The RPM number converted by the speed check window according to resolution, gear ratio, and ballscrew width must fall within the range of 0 to 100 RPM)		
Solution	Set the PLC axis position and speed inspection windows to be within a reasonable range.		

Alarm ID	SERIALPLCAXIS-005	Alarm title	【The first phase of homing speed set is too high】
Description	The first phase homing speed of the serial PLC axis is set too high. Please re-specify.		
Reason	The first phase homing speed of the serial PLC axis is set too high. (The pulse per second converted by first-phase homing speed according to the resolution, gear ratio, and ballscrew width, must be fall within 0~6,553,500 pulses per sec.		
Solution	Lower the first phase of homing speed on the serial PLC axis for it to fall within a reasonable range to remove the alarm. Note: This alarm supports only versions 10.116.38M, 10.116.54K, 10.118.0F, 10.118.6 and before, the later versions do not support driver homing.		
Alarm ID	SERIALPLCAXIS-006	Alarm title	【The second phase of homing speed set is too high】
Description	The second phase homing speed setting of the serial PLC axis is too large. Please re-specify.		
Reason	The second phase homing speed setting of the serial PLC axis is too large. (The pulse per second of the second phase homing speed depends on the resolution, gear ratio, and ballscrew width, and must fall within the range of 0~6,553,500.)		
Solution	Lower the second phase of homing speed on the serial PLC axis to be within a reasonable range. Note: This alarm only supports versions 10.116.38M, 10.116.54K, 10.118.0F, and 10.118.6 and before, the later version do not support the driver homing.		
Alarm ID	SERIALPLCAXIS-007	Alarm title	【The offset of homing is set too high】
Description	The serial PLC axis homing offset setting is too large, please re-specify.		
Possible causes	The offset of serial PLC axis homing is set out of range. (The number of pulse converted from the homing offset based on resolution, gear ratio, and ballscrew width must fall within the range of -1073741823~1073741823 pulse.)		
Solution	Adjust the offset of serial PLC axis homing for it to fall within a reasonable range.		
Alarm ID	SERIALPLCAXIS-008	Alarm title	【Absolute motors do not support incremental homing】

Description	Absolute motor cannot use incremental homing function
Possible causes	When using an absolute motor, the incremental homing action is triggered.
Solution	1. When using absolute motors, trigger mechanical origin settings (Rm.6) rather than the incremental homing (Rm.5, Rm.8). 2. Reset to remove the alarm.

Alarm ID	**SERIALPLCAXIS-009**	**Alarm title**	**【Absolute origin setting is not completed】**
Description	Absolute encoder needs to set the origin at first time.		
Possible causes	Absolute encoder is used, but the machine origin has not been set.		
Solution	Use R(m).6 to set the mechanical origin, when the origin setting is completed (R(m+1).7 On), the alarm can be removed.		

Alarm ID	**SERIALPLCAXIS-010**	**Alarm title**	**【Restart the machine and reset the origin due to absolute relevant parameter variations】**
Description	The user has modified the relevant important parameters which affects the home position setting. Must reboot controller and driver, and perform the serial PLC axis origin setting.		
Possible causes	Modify any of the following parameters: position sensor resolution, number of gear on the side of the ballscrew, and number of gear on the side of the motor.		
Solution	Reboot the controller and driver.		

Alarm ID	**SERIALPLCAXIS-011**	**Alarm title**	**【Driver station numbers do not exist】**
Description	There is no driver corresponding to the serial PLC axis station number on the communication network, and the system enters not ready state.		
Possible causes	There is no drive on the communication network that corresponds to the serial PLC axis station number. (Example: set the drive station number to 999)		
Solution	Set the correct serial PLC axis station number, and reboot the controller and driver.		

Alarm ID	SERIALPLCAXIS-012	Alarm title	【Exceeding the positive software stroke limit】
Description	The axial mechanical coordinate exceeds the positive software stroke limit.		
Possible causes	The axial movement exceeds the setting value.		
Solution	1. Reset can remove the alarms. 2. Move the axial in negative direction to be out of the software stroke protection range, and can remove alarm.		

Alarm ID	SERIALPLCAXIS-013	Alarm title	【Exceeding the negative software stroke limit】
Description	Axial mechanical coordinates exceed the negative software stroke limit.		
Possible causes	The axial movement exceeds the set value.		
Solution	1. Reset to remove the alarm. 2. Move the axial in positive direction to be out of the software stroke protection range to remove the alarm.		

Alarm ID	SERIALPLCAXIS-014	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in M3 communication. 1. M3 response packet not received. 2. M3 response packet CRC error. 3. M3 response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. M3 response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

Alarm ID	SERIALPLCAXIS-015	Alarm title	【Driver parameters writing failed】

Description	Error occurred while writing the serial PLC axis parameters to the driver.
Possible causes	The communication between the controller and the drive is unstable, causing the driver parameter writing timeout.
Solution	After the cause of communication instability is eliminated, reboot the controller and driver.

Alarm ID	**SERIALPLCAXIS-016**	**Alarm title**	**【The origin index signal could not be found】**
Description	When searching for the origin, after leaving the home switch, the motor index signal cannot be found after moving more than 5 pitches.		
Possible causes	1. The index signal cannot be read. 2. The second phase speed setting of the homing is too large. 3. Motor reduction ratio is set too high. 4. The index signal is more than 5 pitches from the origin stroke switch.		
Solution	1. Check if the motor index wiring is normal. 2. Reduce the value of the second phase of homing speed setting. 3. Reset to remove the alarm. **Advanced Instructions** 1. When homing, the machine will move towards the origin switch with the first phase speed setting, and stop until it hits the origin switch, then move in the opposite direction at the second phase homing speed setting. 2. After leaving the origin switch, start looking for the nearest motor index signal. 3. When returning at the second phase speed, with the single feedback of the motor, the controller calculates the resolution of the encoder per revolution. If the index signal is not found within 5 turns of the motor, controller issues the alarm immediately.		

Alarm ID	**SERIALPLCAXIS-017**	**Alarm title**	**【Homing zero speed check failed】**
Description	Controller will issue this alarm while homing when home switch is encountered but the motor cannot be completely stopped.		
Possible causes	1. Driver gain setting is poor, causing the motor to shake. 2. Resonance occurs when the motor is running.		

Solution	1. Check the driver's position loop gain and speed loop gain settings. 2. Start the driver resonance frequency suppression function. 3. Reset to remove alarm. 4. If it cannot be solved, please contact the Vendor. **Advanced Instructions** When homing, the machine will move towards the origin switch with the first phase speed setting, and stops until it hits the origin switch, then move in the opposite direction to search motor index signal at the second phase homing speed setting. After hitting the origin switch at the first phase of speed machine starts to decelerate, if after 0.1 sec from deceleration command is given the machine still cannot decelerate to smaller than the "zero speed inspection window" speed, this alarm will be issued.

Alarm ID	**SERIALPLCAXIS-018**	**Alarm title**	**【Inseparable from the origin switch】**
Description	While homing, after hit the origin switch and return, the system exceed the second phase protection encoder revolution but still cannot leave the origin switch.		
Possible causes	Use a multimeter to measure whether the stroke switch is shorted circuit or malfunction.		
Solution	Resetting can remove the alarms. **Advanced Instructions** When homing, machine will move towards the origin switch with first phase speed until it hits the origin switch, and then move in the opposite direction at the speed of origin searching in the second phase. The controller will calculate the resolution of the encoder per revolution. If the motor exceeds "the second phase protection encoder revolution but still cannot leave the origin switch signal, this alarm is issued.		

Alarm ID	**SERIALPLCAXIS-019**	**Alarm title**	**【Driver power abnormality】**
Description	When the controller sends a motion command, it detects that the power state of the drive is incorrect (servo is not on or powered off).		
Possible causes	1. Drive power supply module voltage abnormal. 2. Drive defect.		
Solution	1. Check whether the driver power supply status is abnormal. 2. Check whether the driver power cable is loose or broken. 3. Replace the driver. 4. After the above driver problems are eliminated, reboot controller and driver to remove the alarm.		

Alarm ID	SERIALPLCAXIS-020	Alarm title	【This driver type is not supported】
Description	This drive type is not supported by the serial PLC axis.		
Possible causes	Serial PLC axis function is supported only by the following driver types. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, * Syntec 2nd generation driver. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, Syntec driver.		
Solution	Replace the driver with a supported driver type and reboot the driver and controller to remove the alarm. Note: The Syntec 2nd Generation M2 drivers only support the serial PLC axis with version 10.116.54H, 10.116.0E, and above.		

Alarm ID	SERIALPLCAXIS-021	Alarm title	【Positioning moving distance exceeds upper limit】
Description	The serial PLC axis positioning movement distance exceeds the upper limit.		
Possible causes	When the moving distance calculated by the current position and the target position is converted into the pulse wave number, is greater than the upper limit of the moving pulse wave number 2,147,483,647. (Avoid to set target position too far, the user misjudge the problem of clockwise and counterclockwise failure and the inability to positioned to the target position.) [upper limit of the positioning moving distance calculation method] **Upper limit of moving distance (um) = upper limit of moving pulse number x (number of gear on the motor side x screw width) / (number of gear on the side of the screw x position sensor resolution x 4)** For example: Number of teeth on the motor side = 1, number of gear on the side of the ballscrew = 1, position sensor resolution = 262,144 (pulse/rev), ballscrew width = 5,000 (um/rev) The upper limit of the moving distance is 2147483647 x (1 x 5000) / (1 x 262144 x 4) = 10239999 (um) =10239.999 (mm). In other words, if the calculated distance between the current position and the target position exceeds 10239.999 mm, this alarm will be issued; If it does not exceed 10239.999 mm, it is a reasonable moving distance.		

Solution	1. Change the target position to avoid the current position and the target position exceed the moving distance of upper limit. 2. Adjust the gear number ration between of motor side and ballscrew side, and reduce the resolution setting on the controller side. i.e. According to the above setting, the drive electronic gear numerator is 8, the position sensor resolution is reduced to 32,768, and the upper limit of the moving distance is enlarged to 81919.999mm. 3. Adjust the gear number on motor side, and ballscrew side. 4. Adjust the screw width. 5. Once the above adjustments are made, the alarm can be removed. 6. Reset to remove the alarms.

Alarm ID	**SERIALPLCAXIS-022**	**Alarm title**	**【This driver type does not support hand wheel function】**
Description	The MPG function is not available for this drive type.		
Possible causes	Currently only the following drive types support the MPG function. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, Syntec 2nd generation drive. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, Syntec drive.		
Solution	1. Replace the driver to which supports MPG functions. 2. Switch to other modes other than the MPG mode (i.e. positioning mode, speed mode) to remove the alarm. Notes: • The Syntec M3 drivers, supported version 10.116.38D, 10.116.54B, and 10.116.0A. • The Syntec 2nd generation M2 drivers, supported versions 10.116.54H and 10.116.0E.		

Alarm ID	**SERIALPLCAXIS-023**	**Alarm title**	**【Syntec encoder firmware upgrade is over, please restart the drive】**
Description	The Syntec of encoder firmware upgrade is over and the drive needs to be reboot.		
Possible causes	Perform axial encoder firmware upgrade.		
Solution	Reboot the controller and driver.		

Alarm ID	**SERIALPLCAXIS-024**	**Alarm title**	**【Syntec encoder information reading timeout】**
Description	Syntec encoder information reading timeout.		

Possible causes	1. The communication between the controller and the drive is abnormal. 2. The driver status is abnormal.
Solution	1. Confirm whetherthe communication cable is loose. 2. Reboot the controller and driver.

Alarm ID	**SERIALPLCAXIS-025**	**Alarm title**	**【Gap control mode switching failure】**
Description	Axial switching to gap control mode failed.		
Possible causes	1. The axis is not in the positioning mode. 2. The target information is in relative position.		
Solution	1. Set the axis to the positioning mode. 2. Set target information to absolute position.		

Alarm ID	**SERIALPLCAXIS-026**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

Alarm ID	**SERIALPLCAXIS-027**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

Alarm ID	SERIALPLCAXIS-028	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		
Alarm ID	SERIALPLCAXIS-029	Alarm title	Driver software misses communication packet
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

11.1 SERIALPLCAXIS-001 ROT/system axis/serial PLC axis driver station number repetition

Alarm ID	SERIALPLCAXIS-001	Alarm title	【ROT/system axis/serial PLC axis driver station number repetition】
Description	The serial PLC axis station number is repeated with other axis station numbers, and the system enters the not ready state.		
Reason	The serial PLC station number setting to repeated with other axis. Please check if the following axis station number settings are repeated: 1. System axis. 2. Serial PLC axis. 3. ROT axis. 4. ATC axis. 5. Setpoint axis		

Alarm ID	SERIALPLCAXIS-001	Alarm title	【ROT/system axis/serial PLC axis driver station number repetition】
Solution	With correct parameter settings, reboot controller and driver to remove the alarm.		

11.2 SERIALPLCAXIS-002 R value registers are located in the system protection area or cannot be bitwise

Alarm ID	SERIALPLCAXIS-002	Alarm title	【R value registers are located in the system protection area or cannot be bitwise】
Description	The initial register is not set properly and is located in the system protection area or does not support bit access.		
Reason	1. The data base address R(n) or the command status base address R(m) is set in the CNC system interface area. 2. The command status base address R(m) is set to a register that does not support bit access.		
Solution	Specify the initial R register correctly (Set Rm to between R50~R80 or R256~R511.)		

11.3 SERIALPLCAXIS-003 Acceleration/deceleration or speed setting error

Alarm ID	SERIALPLCAXIS-003	Alarm title	【Acceleration/deceleration or speed setting error】
Description	Acceleration/deceleration or speed set value is out of range.		
Reason	1. The acceleration/deceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the driver. (The acceleration/deceleration is based on the pulse wave per square sec. of resolution, gear ratio, and the screw width, if it is the "Yakawa" drive, it must fall within the range of 1~655,350,000.) 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration/deceleration time setting, exceeding the range of driver parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" driver, the parameter setting range is 0~60,000)		

Solution	1. Properly enlarge acceleration/deceleration time to within a reasonable range, can remove alarm, 2. * When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot controller an driver. 3. Properly reduce the target speed and maximum movement speed to within a reasonable range to remove the alarm. 4. When using the Syntec 2nd generation M2 drivers, set the acceleration/deceleration time between 0 and 60,000. 5. Reset to clear the alarms. Note: Yaskawa Sigma series M2, Pn833=1 are valid only for version 10.116.36R, 10.116.46, and 10.117.46 and above, otherwise, the following parameters modified on the controller are invalid: 1. ROT tool changing acceleration time. 2. Acceleration/deceleration time R(n+3) and R(n+4) of serial PLC axis positioning/MPG mode. 3. Deceleration time R(n+4) of serial PLC axis speed mode.

11.4 SERIALPLCAXIS-004 Inspection window setting error

Alarm ID	**SERIALPLCAXIS-004**	**Alarm title**	**【Inspection window setting error】**
Description	The serial PLC axis position and speed checking window settings are incorrect. Please re-specify.		
Reason	The serial PLC axis position and speed check window settings are out of range. (The number of pulse converted by the position check window according to resolution, gear ratio, and ballscrew width must fall within the range of 0~1,073,741,824) (The RPM number converted by the speed check window according to resolution, gear ratio, and ballscrew width must fall within the range of 0 to 100 RPM)		
Solution	Set the PLC axis position and speed inspection windows to be within a reasonable range.		

11.5 SERIALPLCAXIS-005 The first phase of homing speed set is too high

Alarm ID	**SERIALPLCAXIS-005**	**Alarm title**	**【The first phase of homing speed set is too high】**
Description	The first phase homing speed of the serial PLC axis is set too high. Please re-specify.		

Reason	The first phase homing speed of the serial PLC axis is set too high. (The pulse per second converted by first-phase homing speed according to the resolution, gear ratio, and ballscrew width, must be fall within 0~6,553,500 pulses per sec.
Solution	Lower the first phase of homing speed on the serial PLC axis for it to fall within a reasonable range to remove the alarm. Note: This alarm supports only versions 10.116.38M, 10.116.54K, 10.118.0F, 10.118.6 and before, the later versions do not support driver homing.

11.6 SERIALPLCAXIS-006 The second phase of homing speed set is too high

Alarm ID	**SERIALPLCAXIS-006**	**Alarm title**	**【The second phase of homing speed set is too high】**
Description	The second phase homing speed setting of the serial PLC axis is too large. Please re-specify.		
Reason	The second phase homing speed setting of the serial PLC axis is too large. (The pulse per second of the second phase homing speed depends on the resolution, gear ratio, and ballscrew width, and must fall within the range of 0~6,553,500.)		
Solution	Lower the second phase of homing speed on the serial PLC axis to be within a reasonable range. Note: This alarm only supports versions 10.116.38M, 10.116.54K, 10.118.0F, and 10.118.6 and before, the later version do not support the driver homing.		

11.7 SERIALPLCAXIS-007 The offset of homing is set too high

Alarm ID	**SERIALPLCAXIS-007**	**Alarm title**	**【The offset of homing is set too high】**
Description	The serial PLC axis homing offset setting is too large, please re-specify.		
Possible causes	The offset of serial PLC axis homing is set out of range. (The number of pulse converted from the homing offset based on resolution, gear ratio, and ballscrew width must fall within the range of -1073741823~1073741823 pulse.)		
Solution	Adjust the offset of serial PLC axis homing for it to fall within a reasonable range.		

11.8 SERIALPLCAXIS-008 Absolute motors do not support incremental homing

Alarm ID	SERIALPLCAXIS-008	Alarm title	【Absolute motors do not support incremental homing】
Description	Absolute motor cannot use incremental homing function		
Possible causes	When using an absolute motor, the incremental homing action is triggered.		
Solution	1. When using absolute motors, trigger mechanical origin settings (Rm.6) rather than the incremental homing (Rm.5, Rm.8). 2. Reset to remove the alarm.		

11.9 SERIALPLCAXIS-009 Absolute origin setting is not completed

Alarm ID	SERIALPLCAXIS-009	Alarm title	【Absolute origin setting is not completed】
Description	Absolute encoder needs to set the origin at first time.		
Possible causes	Absolute encoder is used, but the machine origin has not been set.		
Solution	Use R(m).6 to set the mechanical origin, when the origin setting is completed (R(m+1).7 On), the alarm can be removed.		

11.10 SERIALPLCAXIS-010 Restart the machine and reset the origin due to absolute relevant parameter variations

Alarm ID	SERIALPLCAXIS-010	Alarm title	【Restart the machine and reset the origin due to absolute relevant parameter variations】
Description	The user has modified the relevant important parameters which affects the home position setting. Must reboot controller and driver, and perform the serial PLC axis origin setting.		
Possible causes	Modify any of the following parameters: position sensor resolution, number of gear on the side of the ballscrew, and number of gear on the side of the motor.		
Solution	Reboot the controller and driver.		

11.11 SERIALPLCAXIS-011 Driver station numbers do not exist

Alarm ID	SERIALPLCAXIS-011	Alarm title	【Driver station numbers do not exist】
Description	There is no driver corresponding to the serial PLC axis station number on the communication network, and the system enters not ready state.		
Possible causes	There is no drive on the communication network that corresponds to the serial PLC axis station number. (Example: set the drive station number to 999)		
Solution	Set the correct serial PLC axis station number, and reboot the controller and driver.		

11.12 SERIALPLCAXIS-012 Exceeding the positive software stroke limit

Alarm ID	SERIALPLCAXIS-012	Alarm title	【Exceeding the positive software stroke limit】
Description	The axial mechanical coordinate exceeds the positive software stroke limit.		
Possible causes	The axial movement exceeds the setting value.		
Solution	1. Reset can remove the alarms. 2. Move the axial in negative direction to be out of the software stroke protection range, and can remove alarm.		

11.13 SERIALPLCAXIS-013 Exceeding the negative software stroke limit

Alarm ID	SERIALPLCAXIS-013	Alarm title	【Exceeding the negative software stroke limit】
Description	Axial mechanical coordinates exceed the negative software stroke limit.		
Possible causes	The axial movement exceeds the set value.		
Solution	1. Reset to remove the alarm. 2. Move the axial in positive direction to be out of the software stroke protection range to remove the alarm.		

11.14 SERIALPLCAXIS-014 Drive communication error

Alarm ID	SERIALPLCAXIS-014	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in M3 communication. 1. M3 response packet not received. 2. M3 response packet CRC error. 3. M3 response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. M3 response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

11.15 SERIALPLCAXIS-015 Driver parameters writing failed

Alarm ID	SERIALPLCAXIS-015	Alarm title	【Driver parameters writing failed】
Description	Error occurred while writing the serial PLC axis parameters to the driver.		
Possible causes	The communication between the controller and the drive is unstable, causing the driver parameter writing timeout.		
Solution	After the cause of communication instability is eliminated, reboot the controller and driver.		

11.16 SERIALPLCAXIS-016 The origin index signal could not be found

Alarm ID	SERIALPLCAXIS-016	Alarm title	【The origin index signal could not be found】
Description	When searching for the origin, after leaving the home switch, the motor index signal cannot be found after moving more than 5 pitches.		

Possible causes	1. The index signal cannot be read. 2. The second phase speed setting of the homing is too large. 3. Motor reduction ratio is set too high. 4. The index signal is more than 5 pitches from the origin stroke switch.
Solution	1. Check if the motor index wiring is normal. 2. Reduce the value of the second phase of homing speed setting. 3. Reset to remove the alarm. **Advanced Instructions** 1. When homing, the machine will move towards the origin switch with the first phase speed setting, and stop until it hits the origin switch, then move in the opposite direction at the second phase homing speed setting. 2. After leaving the origin switch, start looking for the nearest motor index signal. 3. When returning at the second phase speed, with the single feedback of the motor, the controller calculates the resolution of the encoder per revolution. If the index signal is not found within 5 turns of the motor, controller issues the alarm immediately.

11.17 SERIALPLCAXIS-017 Homing zero speed check failed

Alarm ID	**SERIALPLCAXIS-017**	**Alarm title**	**【Homing zero speed check failed】**
Description	Controller will issue this alarm while homing when home switch is encountered but the motor cannot be completely stopped.		
Possible causes	1. Driver gain setting is poor, causing the motor to shake. 2. Resonance occurs when the motor is running.		
Solution	1. Check the driver's position loop gain and speed loop gain settings. 2. Start the driver resonance frequency suppression function. 3. Reset to remove alarm. 4. If it cannot be solved, please contact the Vendor. **Advanced Instructions** When homing, the machine will move towards the origin switch with the first phase speed setting, and stops until it hits the origin switch, then move in the opposite direction to search motor index signal at the second phase homing speed setting. After hitting the origin switch at the first phase of speed machine starts to decelerate, if after 0.1 sec from deceleration command is given the machine still cannot decelerate to smaller than the "zero speed inspection window" speed, this alarm will be issued.		

11.18 SERIALPLCAXIS-018 Inseparable from the origin switch

Alarm ID	SERIALPLCAXIS-018	Alarm title	【Inseparable from the origin switch】
Description	While homing, after hit the origin switch and return, the system exceed the second phase protection encoder revolution but still cannot leave the origin switch.		
Possible causes	Use a multimeter to measure whether the stroke switch is shorted circuit or malfunction.		
Solution	Resetting can remove the alarms. **Advanced Instructions** When homing, machine will move towards the origin switch with first phase speed until it hits the origin switch, and then move in the opposite direction at the speed of origin searching in the second phase. The controller will calculate the resolution of the encoder per revolution. If the motor exceeds “the second phase protection encoder revolution but still cannot leave the origin switch signal, this alarm is issued.		

11.19 SERIALPLCAXIS-019 Driver power abnormality

Alarm ID	SERIALPLCAXIS-019	Alarm title	【Driver power abnormality】
Description	When the controller sends a motion command, it detects that the power state of the drive is incorrect (servo is not on or powered off).		
Possible causes	1. Drive power supply module voltage abnormal. 2. Drive defect.		
Solution	1. Check whether the driver power supply status is abnormal. 2. Check whether the driver power cable is loose or broken. 3. Replace the driver. 4. After the above driver problems are eliminated, reboot controller and driver to remove the alarm.		

11.20 SERIALPLCAXIS-020 This driver type is not supported

Alarm ID	SERIALPLCAXIS-020	Alarm title	【This driver type is not supported】

Description	This drive type is not supported by the serial PLC axis.
Possible causes	Serial PLC axis function is supported only by the following driver types. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, * Syntec 2nd generation driver. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, Syntec driver.
Solution	Replace the driver with a supported driver type and reboot the driver and controller to remove the alarm. Note: The Syntec 2nd Generation M2 drivers only support the serial PLC axis with version 10.116.54H, 10.116.0E, and above.

11.21 SERIALPLCAXIS-021 Positioning moving distance exceeds upper limit

Alarm ID	**SERIALPLCAXIS-021**	**Alarm title**	**【Positioning moving distance exceeds upper limit】**
Description	The serial PLC axis positioning movement distance exceeds the upper limit.		
Possible causes	When the moving distance calculated by the current position and the target position is converted into the pulse wave number, is greater than the upper limit of the moving pulse wave number 2,147,483,647. (Avoid to set target position too far, the user misjudge the problem of clockwise and counterclockwise failure and the inability to positioned to the target position.) [upper limit of the positioning moving distance calculation method] **Upper limit of moving distance (um) = upper limit of moving pulse number x (number of gear on the motor side x screw width) / (number of gear on the side of the screw x position sensor resolution x 4)** For example: Number of teeth on the motor side = 1, number of gear on the side of the ballscrew = 1, position sensor resolution = 262,144 (pulse/rev), ballscrew width = 5,000 (um/rev) The upper limit of the moving distance is 2147483647 x (1 x 5000) / (1 x 262144 x 4) = 10239999 (um) =10239.999 (mm). In other words, if the calculated distance between the current position and the target position exceeds 10239.999 mm, this alarm will be issued; If it does not exceed 10239.999 mm, it is a reasonable moving distance.		

Solution	1. Change the target position to avoid the current position and the target position exceed the moving distance of upper limit. 2. Adjust the gear number ration between of motor side and ballscrew side, and reduce the resolution setting on the controller side. i.e. According to the above setting, the drive electronic gear numerator is 8, the position sensor resolution is reduced to 32,768, and the upper limit of the moving distance is enlarged to 81919.999mm. 3. Adjust the gear number on motor side, and ballscrew side. 4. Adjust the screw width. 5. Once the above adjustments are made, the alarm can be removed. 6. Reset to remove the alarms.

11.22 SERIALPLCAXIS-022 This driver type does not support hand wheel function

Alarm ID	**SERIALPLCAXIS-022**	**Alarm title**	**【This driver type does not support hand wheel function】**
Description	The MPG function is not available for this drive type.		
Possible causes	Currently only the following drive types support the MPG function. M2: Yaskawa sigma5, Yaskawa sigma7, Yaskawa sigmaM, Syntec 2nd generation drive. M3: Yaskawa sigma5, Yaskawa sigma7S, Yaskawa sigmaM, Syntec drive.		
Solution	1. Replace the driver to which supports MPG functions. 2. Switch to other modes other than the MPG mode (i.e. positioning mode, speed mode) to remove the alarm. Notes: • The Syntec M3 drivers, supported version 10.116.38D, 10.116.54B, and 10.116.0A. • The Syntec 2nd generation M2 drivers, supported versions 10.116.54H and 10.116.0E.		

11.23 SERIALPLCAXIS-023 Syntec encoder firmware upgrade is over, please restart the drive

Alarm ID	**SERIALPLCAXIS-023**	**Alarm title**	**【Syntec encoder firmware upgrade is over, please restart the drive】**
Description	The Syntec of encoder firmware upgrade is over and the drive needs to be reboot.		
Possible causes	Perform axial encoder firmware upgrade.		

Solution	Reboot the controller and driver.

11.24 SERIALPLCAXIS-024 Syntec encoder information reading timeout

Alarm ID	**SERIALPLCAXIS-024**	**Alarm title**	**【Syntec encoder information reading timeout】**
Description	Syntec encoder information reading timeout.		
Possible causes	1. The communication between the controller and the drive is abnormal. 2. The driver status is abnormal.		
Solution	1. Confirm whetherthe communication cable is loose. 2. Reboot the controller and driver.		

11.25 SERIALPLCAXIS-025 Cruise mode switching failure

Alarm ID	**SERIALPLCAXIS-025**	**Alarm title**	**【Gap control mode switching failure】**
Description	Axial switching to gap control mode failed.		
Possible causes	1. The axis is not in the positioning mode. 2. The target information is in relative position.		
Solution	1. Set the axis to the positioning mode. 2. Set target information to absolute position.		

11.26 SERIALPLCAXIS-026 Poor contact of communication wire

Alarm ID	**SERIALPLCAXIS-026**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		

Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.

11.27 SERIALPLCAXIS-027 The hardware doesn't receive communication packet

Alarm ID	**SERIALPLCAXIS-027**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

11.28 SERIALPLCAXIS-028 The communication packet is disturbed by noise

Alarm ID	**SERIALPLCAXIS-028**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

11.29 SERIALPLCAXIS-029 Driver software misses communication packet

Alarm ID	SERIALPLCAXIS-029	Alarm title	Driver software misses communication packet
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

SYNTEC

12 SRI Alarm -SRI

Alarm ID	**SRI-001**	**Alarm Title**	**【Scan time setting is too short】**
Description	The actual scan time of the controller exceeds the set scan time.		
Possible Reason	The insufficient performance of the system, the number of scan points is too large and the set scan time is too short.		
Solution	Enlarge the scan time.		

Alarm ID	**SRI-002**	**Alarm Title**	**【Communication Timeout】**
Description	The device has no response.		
Possible Reason	There is a problem with the wiring or the device hardware.		
Solution	Check whether the hardware setting and wiring connection is accurate.		

Alarm ID	**SRI-003**	**Alarm Title**	**【Package CRC error】**
Description	The CRC error has exceeded a certain number of times.		
Possible Reason	The hardware is not properly connected or the cable is defective		
Solution	Check whether the hardware is properly set up and connected, especially the wires.		

Alarm ID	**SRI-004**	**Alarm Title**	**【Module station number or sortingdoes not match the parameters】**
Description	When the parameter setting is different from the actual installation configuration.		
Possible Reason	The external device station number or module sequence has been changed.		
Solution	Please reset the system parameters.		

Alarm ID	**SRI-005**	**Alarm Title**	**【Firmware update failure】**
Description	An error occurred during the firmware update process.		

Alarm ID	**SRI-005**	**Alarm Title**	**【Firmware update failure】**
Possible Reason	1. An error occurred in the communication during the update process. 2. The module is powered abnormally during the update process. 3. The update file is broken. 4. FC-IO module malfunction.		
Solution	1. Check if the hardware is properly set up and connected, especially the wire, and re-update the firmware. 2. Check if the module power is supplied properly and re-update the firmware. 3. If the firmware update continues to fail, please contact Syntec for solution.		

Alarm ID	**SRI-006**	**Alarm Title**	**【Module is in copying mode】**
Description	The module is detected in the copying mode when it is turned on.		
Possible Reason	The firmware has failed to copy, causing the module to stay in the copying mode.		
Solution	Please update the firmware.		

Alarm ID	**SRI-007**	**Alarm Title**	**【Module communication abnormal】**
Description	An error code occurred when the module is communicating.		
Possible Reason	The module can not normally communicate.		
Solution	1. Power off and Reboot. 2. Update the firmware version. 3. This alarm appears after system running for a period of time, indicating that it may be noise interference. Please check whether surrounding has interference source, whether the equipment and wire anti-noise treatment is enough.		

Alarm ID	**SRI-008**	**Alarm Title**	**【Module hardware configuration error】**
Description	Module initialization failed.		
Possible Reason	The HW-ID interpretation of this module is incorrect.		
Solution	Return to Syntec for repair.		

Alarm ID	SRI-009	Alarm Title	【The firmware update is successful, Please power off and reboot】
Description	After the firmware update is successful, the system to be powered off and rebooted.		
Possible Reason	After the firmware update is successful, he system to be powered off and rebooted.		
Solution	Power off and reboot the controller and SRI module.		

12.1 SRI-001 Scan time setting is too short

Alarm ID	SRI-001	Alarm Title	【Scan time setting is too short】
Description	The actual scan time of the controller exceeds the set scan time.		
Possible Reason	The insufficient performance of the system, the number of scan points is too large and the set scan time is too short.		
Solution	Enlarge the scan time.		

12.2 SRI-002 Communication Timeout

Alarm ID	SRI-002	Alarm Title	【Communication Timeout】
Description	The device has no response.		
Possible Reason	There is a problem with the wiring or the device hardware.		
Solution	Check whether the hardware setting and wiring connection is accurate.		

12.3 SRI-003 Package CRC error

Alarm ID	SRI-003	Alarm Title	【Package CRC error】
Description	The CRC error has exceeded a certain number of times.		
Possible Reason	The hardware is not properly connected or the cable is defective		

Alarm ID	SRI-003	Alarm Title	【Package CRC error】
Solution	Check whether the hardware is properly set up and connected, especially the wires.		

12.4 SRI-004 Module station number or sortingdoes not match the parameters

Alarm ID	SRI-004	Alarm Title	【Module station number or sortingdoes not match the parameters】
Description	When the parameter setting is different from the actual installation configuration.		
Possible Reason	The external device station number or module sequence has been changed.		
Solution	Please reset the system parameters.		

12.5 SRI-005 Firmware update failure

Alarm ID	SRI-005	Alarm Title	【Firmware update failure】
Description	An error occurred during the firmware update process.		
Possible Reason	1. An error occurred in the communication during the update process. 2. The module is powered abnormally during the update process. 3. The update file is broken. 4. FC-IO module malfunction.		
Solution	1. Check if the hardware is properly set up and connected, especially the wire, and re-update the firmware. 2. Check if the module power is supplied properly and re-update the firmware. 3. If the firmware update continues to fail, please contact Syntec for solution.		

12.6 SRI-006 Module is in copying mode

Alarm ID	SRI-006	Alarm Title	【Module is in copying mode】
Description	The module is detected in the copying mode when it is turned on.		
Possible Reason	The firmware has failed to copy, causing the module to stay in the copying mode.		

Alarm ID	SRI-006	Alarm Title	【Module is in copying mode】
Solution	Please update the firmware.		

12.7 SRI-007 Module communication abnormal

Alarm ID	SRI-007	Alarm Title	【Module communication abnormal】
Description	An error code occurred when the module is communicating.		
Possible Reason	The module can not normally communicate.		
Solution	1. Power off and Reboot. 2. Update the firmware version. 3. This alarm appears after system running for a period of time, indicating that it may be noise interference. Please check whether surrounding has interference source, whether the equipment and wire anti-noise treatment is enough.		

12.8 SRI-008 Module hardware configuration error

Alarm ID	SRI-008	Alarm Title	【Module hardware configuration error】
Description	Module initialization failed.		
Possible Reason	The HW-ID interpretation of this module is incorrect.		
Solution	Return to Syntec for repair.		

12.9 SRI-009 The firmware update is successful, Please power off and reboot

Alarm ID	SRI-009	Alarm Title	【The firmware update is successful, Please power off and reboot】
Description	After the firmware update is successful, the system to be powered off and rebooted.		
Possible Reason	After the firmware update is successful, he system to be powered off and rebooted.		

Alarm ID	**SRI-009**	**Alarm Title**	**【The firmware update is successful, Please power off and reboot】**
Solution	Power off and reboot the controller and SRI module.		

SYNTEC

13 MACRO Alarm - MACRO

Mill	
001	The lowest point in PROFILE is lower than the platform.
002	The amount of feed each time of the vertical axis or extension axis is less than or equal to 0.
003	The maximum arc chord length is less than or equal to 0.
004	The maximum arc radius is less than or equal to 0.
005	The specified PROFILE path is incorrect.
006	The designation of G02 or G03 in PROFILE is incorrect.
007	The minimum arc radius is less than or equal to 0.
011	The feed plane R of the drilling (boring) cycle is lower than the plane of the hole bottom.
012	The drilling (boring) cycle does not specify the depth of the hole to be drilled (bored).
013	The peck drilling cycle does not specify feed amount Q.
015	The Hole number for multi-holes drilling should not be zero.
016	Fast drilling G81 lifting rate F2 argument must be integer.
017	In G41/G42 mode, G12.1 cannot be activated.
018	In G96 mode, G12.1 cannot be activated.
019	G12.1 axis does not exist.
020	Under G41/G42 mode, G12.1 cannot end.
031	XY plane feed depth is too small.
032	Z feed depth is too small.

Mill	
040	The machining direction is not specified.
041	The tool diameter is too large, exceed the workpiece.
042	The fillet radius is smaller than that of the tool or too large to fit into the corner.
051	The X axis step value exceeds the limit value.
052	The Y axis step value exceeds the limit value.
053	The Z axis step value exceeds the limit value.
054	X axis length is less than 0.
055	Y axis width is less than 0.
056	The maximum depth of axis Z is less than 0.
057	The X axis starting hole of the matrix hole must not be less than or equal to 0.
058	The Y axis starting hole of the matrix hole must not be less than or equal to 0.
059	The number of auto-compensation holes of the matrix hole must not be less than or equal to 0.
405	Use a custom pause point retreat program for non-conventional mills.
406	G53.6 needs to be enabled in G40 mode.
407	The selected tool number of G53.6 cannot be 0.
408	Tool alignment is invalid or failed.
499	Using G84.48 in the wrong time
500	Tapping retract argument error

Lathe	
001	The first block has no net movement on the Z-axis.
002	Turning path X, Z axis allow only unidirectional increase or decrease.
003	The first single segment has no net movement on the X-axis.
004	The Z axis of the turning path allows only unidirectional increase or decrease.
005	The starting point of turning is below the path.
006	Tool numbers are allowed only to be in four digits.
011	The feed plane R of the drilling (boring) cycle is lower than the plane of the hole bottom.
012	The drilling (boring) cycle does not specify the plane of the hole bottom.
013	The peck drilling cycle does not specify feed amount Q.
014	In G41/G42 mode, G12.1 cannot be started.
015	In G96 mode, G12.1 cannot be activated.
016	The G12.1 axis does not exist.
017	Under G41/G42 mode, G12.1 cannot end.
018	G73/G74 H value input error.
019	The thread feed has no specified length or height.
020	The chamfer length of thread feed/retraction exceeds the total movement on the Z axis.
021	The thread feed position along the X axis is higher than the starting point.
022	Z and X half-axis lengths of elliptical interpolation are not input, less than, or equal to zero.
023	The starting and end point distance of ellipse interpolation is greater than the elliptic major axis length.

Lathe	
024	The focal length of parabolic interpolation is not input, less than, or equal to zero.
025	The starting and end point connecting line of parabolic interpolation is parallel to the symmetry axis.
026	The starting and end points of ellipse and parabolic interpolation with Z coordinates cannot be the same point.
027	The parabola end point is not on the parabola.
028	pecking type turning escape direction conflict.
Common	
300	Auto mid-point function starting point error.
301	Auto mid-point function length or safety distance input error.
302	Auto mid-point function width or safe distance input error.
303	Auto mid-point function safety height input error.
304	Auto mid-point function speed input error.
305	Auto mid-point function safety distance input error.
306	Auto mid-point function length or width input error.
307	An error occurred during auto mid-point process.
320	Input error in the distance between the spindle and the tool set probe.
321	Input error in estimated tool length.
322	Input error in the distance between the spindle and the tool set probe or in estimated tool length.
323	The error in the first/second detection exceeds the allowable range.

Common	
324	Error occurred in tool measurement.
330	Z axis lowest point error alarm.
331	Tool measuring error alarm.
332	The error in the second/third detection exceeds the set range, or tool probe signal abnormal.
333	The Z direction starting point error alarm.
334	G37.3 compensation tool number not set.
335	The measuring position setting is wrong and the measuring signal of tool length has been triggered.
336	The measuring position setting is wrong and the measuring signal of tool length is not triggered.
401	Wrong input in workpiece coordinate system settings.
402	Wrong input in tool settings.
403	Wrong input in tool setting mode.
404	Use error reference point settings.
411	Command format error.
412	The argument setting exceeds the range.
451	The spindle tool number is different from the tool breakpoint tool number.
452	Tool retraction function only supports X/Y/Z one axis retraction or retract along tool vector.
453	Breakpoint return program does not support G91 mode.
454	Tool retraction function only supports retraction distance R greater than 0.

Alarm ID	MAR-408	Alarm Title	【Tool alignment is invalid or failed.】
Description	Tool alignment is invalid or failed because system is unable to calculate alignment angle.		
Reason	When system is unable to calculate alignment angle, COR alarms couldn't be normally triggered because there is no tilted working plane command preceding tool alignment command. Note: tilted working plane command includes G68.2, G68.3, ... Note: tool alignment command includes G53.1, G53.3, G53.6, ...		
Solution	Use tilted working plane command before tool alignment command so that COR alarms can be triggered normally. Then troubleshoot according to the COR alarm.		

13.1 Macro alarm - MACRO - Common

Alarm ID	MACRO-7xx	Alarm title	【Txx tool life has expired】
Description	1. Suitable for milling machines and lathes 2. Among them, the "xx" displayed in MACRO-7xx and Txx is the tool number of the corresponding life 3. Inform the user that the usage of the tool has expired the set value, please check or replace the tool		
Possible reason	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found		
Method of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established		

Alarm ID	MACRO-797	Alarm title	【The life of multiple tools has expired, please go to the tool management function to check】
Description	1. Suitable for milling machines and lathes 2. The life of many tools has expired, and none of the group tools can be found		

Alarm ID	MACRO-797	Alarm title	【The life of multiple tools has expired, please go to the tool management function to check】
Possible reason	1. The tool life management has been turned on, the life of multiple tools used has expired, and the group tool cannot be found		
Method of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established		

13.1.1 MACRO-701~796 Txx tool life has expired

Alarm ID	MACRO-7xx	Alarm title	【Txx tool life has expired】
Description	1. Suitable for milling machines and lathes 2. Among them, the "xx" displayed in MACRO-7xx and Txx is the tool number of the corresponding life 3. Inform the user that the usage of the tool has expired the set value, please check or replace the tool		
Possible reason	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found		
Method of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established		

13.1.2 MACRO-797 The life of multiple tools has expired, please go to the tool management function to check

Alarm ID	MACRO-797	Alarm title	【The life of multiple tools has expired, please go to the tool management function to check】
Description	1. Suitable for milling machines and lathes 2. The life of many tools has expired, and none of the group tools can be found		
Possible reason	1. The tool life management has been turned on, the life of multiple tools used has expired, and the group tool cannot be found		

Alarm ID	MACRO-797	Alarm title	【The life of multiple tools has expired, please go to the tool management function to check】
Method of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established		

13.2 Macro Alarm - MACRO - Grinder

13.2.1 Precautions

1. The ID attached to the MSG warning can be found in the alarm manual for the corresponding ID exclusion method. For example, pop out MSG 10115 to query the alarm manual AlarmID 10115.



Alarm ID	MACRO-10001	Alarm title	【File Operation Error】
Description	An error occurred while operating the file.		
Possible reason	1. G302 is not executed before G303 is executed. 2. When using G302 to call the grinding wheel data, the grinding wheel with the specified number does not exist. 3. The grinding wheel characteristics and dressing condition files do not exist. 4. The grinding wheel list file does not exist.		
Method of exclusion	1. Check whether G302 is executed before the program executes G303. 2. Build the tool. 3. Delete the existing grinding wheel data and add new data.		

Alarm ID	MACRO-10002	Alarm title	【Grinding Wheel Tool Number Setting Error】
Description	The tool number of the grinding wheel is set incorrectly.		
Possible reason	1. The tool number format of the program is wrong. 2. The tool number exceeds the set range of 1~15.		

Alarm ID	MACRO-10002	Alarm title	【Grinding Wheel Tool Number Setting Error】
Method of exclusion	1. Refer to the instruction of the T code to confirm whether the format of the tool number is two digits, for example, the first tool is 01, the third tool is 03, and the twelfth tool is 12. 2. Re-set the tool number value to be in the range of 1~15.		

Alarm ID	MACRO-10003	Alarm title	【Grinding Wheel Cutting Edge Setting Error】
Description	The cutting edge of the grinding wheel is set incorrectly.		
Possible reason	1. The format of the cutting edge number of the program is wrong. 2. The cutting edge number exceeds the range of 1~6.		
Method of exclusion	1. Refer to the instruction of the T code to confirm whether the format of the cutting edge number is two digits, for example, cutting edge one is 01, cutting edge three is 03. 2. Re-set the cutting edge number to be in the range of 1~6.		

Alarm ID	MACRO-10004	Alarm title	【The Number of Pieces of Grinding Wheel Compensation is Incorrectly Set】
Description	The number of pieces of grinding wheel compensation is incorrectly set.		
Possible reason	The number of pieces of grinding wheel wearing is set to less than 0。		
Method of exclusion	Reset the grinding wheel compensation piece counting to an integer greater than or equal to 0.		

Alarm ID	MACRO-10005	Alarm title	【The Wheel Rotation Function is Set Incorrectly】
Description	When performing sand dressing or grinding wheel call, the setting of the grinding wheel rotation function condition is incorrect.		
Possible reason	1. The grinding wheel speed unit exceeds the range of 0~1. 2. The wheel speed is not set.		

Alarm ID	MACRO-10005	Alarm title	【The Wheel Rotation Function is Set Incorrectly】
Method of exclusion	1. Check if the wheel speed unit is 0 (M/S) or 1 (RPM). 2. Set the wheel speed.		

Alarm ID	MACRO-10206	Alarm 標題	【超細磨每次進給量設定錯誤】
說明	超細磨每次進給量(U2)設定有誤。		
可能原因	1.執行G201/G205/G206/G207/G208/G211/G212 時有設定細磨每次進給量且參數Pr4003(開啟檢查刀具最大進給深度狀態)設定為1或2。 2.執行G201/G205/G206/G207/G208/G211/G212 時未設定細磨每次進給量(U2)數值。 3.執行G201/G205/G206/G207/G208/G211/G212 時超細磨每次進給量(U2)大於超細研磨量(U1)。		
排除方法	1.修改細磨每次進給量；或修改磨削進給上限；或參閱磨床專用參數手冊，將參數Pr4003設定為0。 2.重新設定超細磨每次進給量(U2)引數欄位。 3.確認超細磨每次進給量(U2)是否小於超細研磨量(U1)。		

Alarm ID	MACRO-10207	Alarm 標題	【超細磨進給速度未設定】
說明	細磨進給速度未設定。		
可能原因	1.執行G201 指令時，未設定超細磨進給速度(U3)數值。 2.執行G202/G203/G209/G210 指令時，未設定超細磨進給速度(U2)數值。		
排除方法	1.重新設定G201 超細磨進給速度(U3)引數欄位。 2.重新設定G202/G203/G209/G210 超細磨進給速度(U2)引數欄位。		

Alarm ID	MACRO-10008	Alarm title	【Incorrect Setting of Each Feed for Coarse Grinding】
Description	The setting of each feed (D) for coarse grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the coarse grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each coarse grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the coarse grinding feed amount (D) per time is larger than the coarse grinding amount (A).		

Alarm ID	MACRO-10008	Alarm title	【Incorrect Setting of Each Feed for Coarse Grinding】
Method of exclusion	1. Modify the coarse grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the coarse grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the coarse grinding amount (A).		

Alarm ID	MACRO-10009	Alarm title	【Incorrect Setting of Each Feed for Middle Grinding】
Description	The setting of each feed (E) of the middle grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the middle grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each middle grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the middle grinding feed amount (D) per time is larger than the coarse grinding amount (A).		
Method of exclusion	1. Modify the middle grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the middle grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the middle grinding amount (A).		

Alarm ID	MACRO-10010	Alarm title	【Incorrect Setting of Each Feed for Fine Grinding】
Description	The setting of each feed (E) of the fine grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the fine grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each fine grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the fine grinding feed amount (D) per time is larger than the coarse grinding amount (A).		

Alarm ID	MACRO-10010	Alarm title	【Incorrect Setting of Each Feed for Fine Grinding】
Method of exclusion	1. Modify the fine grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the fine grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the fine grinding amount (A).		

Alarm ID	MACRO-10011	Alarm title	【Coarse Grinding Feed Rate is Not Set】
Description	The coarse grinding feed rate is not set.		
Possible reason	1. When the G201 command is executed, the coarse grinding feed rate(I) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the coarse grinding feed rate(D) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 coarse grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 coarse grinding feed rate(D).		

Alarm ID	MACRO-10012	Alarm title	【Middle Grinding Feed Rate is Not Set】
Description	The middle grinding feed rate is not set.		
Possible reason	1. When the G201 command is executed, the middle grinding feed rate(P) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the middle grinding feed rate(E) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 middle grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 middle grinding feed rate(D).		

Alarm ID	MACRO-10013	Alarm title	【Fine Grinding Feed Rate is Not Set】
Description	The fine grinding feed rate is not set.		

Alarm ID	MACRO-10013	Alarm title	【Fine Grinding Feed Rate is Not Set】
Possible reason	1. When the G201 command is executed, the fine grinding feed rate(I) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the fine grinding feed rate(D) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 fine grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 fine grinding feed rate(D).		

Alarm ID	MACRO-10015	Alarm title	【G202 Shoulder Coarse Grinding Speed is Not Set】
Description	When using the G202 straight shoulder grinding function, the coarse grinding speed (H) of the shoulder is not set.		
Possible reason	The shoulder coarse grinding speed (H) value was not set when the G202 command was executed.		
Method of exclusion	Reset the shoulder coarse grinding speed (H) argument field.		

Alarm ID	MACRO-10016	Alarm title	【G202 Shoulder Fine Grinding Speed is Not Set】
Description	When using the G202 straight shoulder grinding function, the fine grinding speed (M) of the shoulder is not set.		
Possible reason	The shoulder fine grinding speed (M) value was not set when the G202 command was executed.		
Method of exclusion	Reset the shoulder fine grinding speed (M) argument field.		

Alarm ID	MACRO-10018	Alarm title	【G204 Condition Setting Error】
Description	When G204 is used for radius grinding, the grinding conditions are set incorrectly.		
Possible reason	1. No round radius (R) value specified. 2. An error occurs when the corner rounding process (H) or the value of it is not set.		

Alarm ID	MACRO-10018	Alarm title	【G204 Condition Setting Error】
Method of exclusion	1. Reset the round radius (R) argument field. 2. Rerun the corner rounding process (H) parameter field or delete the operation and add the operation again.		

Alarm ID	MACRO-10019	Alarm title	【G206/G208 Z-direction Infeed Condition Error】
Description	The conditions required for G206/G208 to execute Z-direction infeed are wrong or insufficient.		
Possible reason	1. When G206/208 command is executed, the value of feed overlap amount (I) is not specified. 2. The wheel currently has a width of 0 or is empty. 3. When G206/208 command is executed, the value of infeed overlap (I) is larger than the current width of the grinding wheel.		
Method of exclusion	1. Reset the infeed overlap (I) parameter field. 2. Reset the current width of the grinding wheel. 3. Reset the feed overlap amount or the current width of the grinding wheel		

Alarm ID	MACRO-10018	Alarm title	【G209 Left/right Shoulder Selection is Not Set】
Description	When using the G209 shoulder straight grinding function, the left/right shoulder selection (W) is not set.		
Possible reason	When the G209 command is executed, the left/right shoulder selection (W) value is not set.		
Method of exclusion	Reset the left/right shoulder selection (W) argument field.		

Alarm ID	MACRO-10021	Alarm title	【G209 Arc Radius Setting Error】
Description	When using the G209 shoulder straight grinding function, the arc radius setting (V) is incorrect.		
Possible reason	When G209 command is executed, the arc radius (V) value is not set or the arc radius setting (V) value is incorrect.		

Alarm ID	MACRO-10021	Alarm title	【G209 Arc Radius Setting Error】
Method of exclusion	Check that the arc radius (V) argument fields are all entered with numeric values or modified argument formats.		

Alarm ID	MACRO-10022	Alarm title	【MACRO-10022 G209 X-axis Dimension Second Coordinate Setting Error】
Description	When using the G209 shoulder straight grinding function, the second coordinate (U) of the X-axis dimension is set incorrectly.		
Possible reason	When G209 command is executed, the value of the second coordinate (U) of the X-axis dimension is not set or the value of the second coordinate (U) of the X-axis dimension is set incorrectly.		
Method of exclusion	Check whether the second coordinate (U) argument field of the X-axis dimension has entered a value or corrected the argument format.		

Alarm ID	MACRO-10023	Alarm title	【Swing Speed is Not Set】
Description	When using the G205/G206/G207/G208 function, the swing speed (J) is not set.		
Possible reason	When the G205/G206/G207/G208 command is executed, the swing speed (J) value is not set.		
Method of exclusion	Reset the swing speed (J) argument field.		

Alarm ID	MACRO-10024	Alarm title	【Safe Point Return Method is Not Set】
Description	The grinding safety point return method is not set.		
Possible reason	The settings are not made on the Grinding Safe Points page.		
Method of exclusion	Go to F3 Offset/Setting → F3 Workpiece Coordinate System → F5 Grinding Safety Point to set.		

Alarm ID	MACRO-10025	Alarm title	【The End Surface Measurement Result Exceeds the Error Threshold】
Description	The measurement error of general parts and standard parts exceeds the threshold.		

Alarm ID	MACRO-10025	Alarm title	【The End Surface Measurement Result Exceeds the Error Threshold】
Possible reason	1. The size of the general piece is wrong. 2. The measurement result is different from the actual position of the workpiece end face.		
Method of exclusion	1. Check whether the size of the workpiece is correct or whether the position of the workpiece is correct. 2. Check whether the triggering of the end surface measurement tripping signal is normal, and check whether the C bit of the specified value of the signal source is triggered. For example, fill in 111 to check whether C111 is triggered.		

Alarm ID	MACRO-10026	Alarm title	【The End Surface Measurement Signal is Not Triggered】
Description	No signal is triggered when the measurement action is completed.		
Possible reason	1. The end surface measurement trip signal is not properly triggered. 2. The starting point of measurement is too far from the workpiece, and no signal is triggered after the first stage of measurement is performed.		
Method of exclusion	1. Check whether the C bit of the specified value of the end surface signal source is triggered, for example, fill in 111 to check whether C111 is triggered. 2. Modify the measurement starting point or increase the measurement distance of the first section to ensure that the distance between the end surface of workpieces and the measurement starting point is less than the first section measurement distance.		

Alarm ID	MACRO-10027	Alarm title	【M Code List - Q Code of Jump Signal is Not Set】
Description	M code list - jump signal is not set.		
Possible reason	1. The M code list does not set the Q code of the end surface measurement trip signal. 2. The M code list does not set the Q code for the skip signal of coarse, middle and fine grinding.		
Method of exclusion	1. Reset the Q code of the end surface measurement trip signal. 2. Reset the Q code of the skip signal for coarse, middle and fine grinding.		

Alarm ID	MACRO-10028	Alarm title	【M Code List - M Code of Measuring Equipment is Not Set】
Description	M code list - measuring equipment not set.		
Possible reason	1. The M code list does not set the workpiece end surface measurement to open M code or workpiece end surface measurement to close the M code. 2. The M code list does not set the outer diameter measurement on M code (1 or 2) or the outer diameter measurement off M code (1 or 2).		
Method of exclusion	1. Re-set workpiece end surface measurement to open M code or workpiece end surface measurement to close M code. 2. Re-set the outer diameter measurement to open M code (1 or 2) or to close the outer diameter measurement to M code (1 or 2).		
Alarm ID	MACRO-10029	Alarm title	【Problems Occurred in the Second Stage Measurement】
Description	A problem occurred when reverting to the starting point of the second measurement		
Possible reason	1. The measurement speed of the first stage is too fast, resulting in a large gap between the coordinates recorded when the probe stops and the actual coordinates of the workpiece end surface, and the probe is still in the trigger state when it returns to the starting point of the second stage of measurement. 2. The measuring distance of the second stage is too short, so that the probe is still in the triggering state when returning to the starting point of the second stage.		
Method of exclusion	1. Reduce the measurement speed of the first stage, reduce the continuous moving distance after touching the end surface of the workpiece to the actual stop, so as to avoid the probe still in the triggering state after the second stage of measurement is returned. 2. Increase the measurement distance of the second stage to ensure that the retraction distance of the second stage measurement is greater than the moving distance between the touch end face and the actual stop during the first stage measurement, so as to prevent the probe from being triggered.		
Alarm ID	MACRO-10030	Alarm title	【Trouble Moving to the Measurement Start Point】
Description	There is a problem with the signal in the middle of moving to the measurement start point.		
Possible reason	The measurement signal is triggered midway to the measurement starting point.		

Alarm ID	**MACRO-10030**	**Alarm title**	**【Trouble Moving to the Measurement Start Point】**
Method of exclusion	Check whether there is any obstacle in the way to the measurement starting point, and re-teach the measurement starting point position.		

Alarm ID	**MACRO-10031**	**Alarm title**	**【M Code List - M Code of Workpieces is Not Set】**
Description	M code list - M code of the workpiece of the shaft equipment is not set.		
Possible reason	The M code list does not set the workpiece to start forwarding or reversing M code or to delay stopping M code.		
Method of exclusion	Reset the workpiece to start forwarding or reversing M code or to delay stopping M code.		

Alarm ID	**MACRO-10032**	**Alarm title**	**【M Code List - M Code of Grinding Wheel is Not Set】**
Description	M code list - The grinding wheel of the shaft equipment is not set.		
Possible reason	The M code list does not set the grinding wheel to start rotating M code or the grinding wheel delay stopping m code.		
Method of exclusion	Reset the grinding wheel to start rotating M code or the grinding wheel to delay stopping M code.		

Alarm ID	**MACRO-10033**	**Alarm title**	**【Non-grinder-specific Software, Please Contact the Original Factory for Assistance】**
Description	The software cannot be used without decryption.		
Possible reason	Option2 is not turned on in the system, and the grinder software cannot be used.		
Method of exclusion	Contact the original factory for assistance in decryption.		

Alarm ID	MACRO-10100	Alarm title	【Pr413 G92(G92.1) Coordinate Retention Mode Setting Error】
Description	Pr413 G92 (G92.1) coordinate system error after grind wheel dressing is interrupted due to the current set value of coordinate retention mode.		
Possible reason	Pr413 G92(G92.1) coordinate retention mode is set to 1 or 2.		
Method of exclusion	Pr413 G92 (G92.1) coordinate retention mode is set to 0.		

Alarm ID	MACRO-10101	Alarm title	【The Edge Number Compensation Flag is Out of Range】
Description	The advanced setting of grind wheel dressing conditions provides the user to set whether each cutting edge should be compensated after dressing. The setting value is found to be out of range in macro.		
Possible reason	The tool edge number compensation flag exceeds the range of 0~1.		
Method of exclusion	Check whether the tool edge number compensation flag (@230~@235) contains a value other than 0 and 1.		

Alarm ID	MACRO-10103	Alarm title	【Pr4004 Incorrect Spindle Number Specified】
Description	The specified spindle number in Pr4004 is incorrectly set.		
Possible reason	The spindle number specified by Pr4004 is less than 1 or greater than 10.		
Method of exclusion	Refer to the grinder-specific parameter manual to check whether the spindle number specified by Pr4004 is between 1 and 10.		

Alarm ID	MACRO-10105	Alarm title	【The Trimming Method is Set Incorrectly】
Description	G301 The trimming method is out of range.		
Possible reason	G301 The trimming method exceeds the range of 1, 3, and 4.		
Method of exclusion	Check whether the value of the trimming method exceeds the range of 1, 3, and 4.		

Alarm ID	MACRO-10106	Alarm title	【Fix Interruption Errors】
Description	The setting of the grinding wheel dressing return condition is incorrect.		
Possible reason	1. The flag (@1005) of whether the previous dressing has been interrupted exceeds the range of 0~1 (0: no previous interrupt occurred, 1: previous interrupt occurred) 2. The previous dressing was interrupted, this time using interrupted dressing (with input return amount), the system check found that the grinding wheel data were different. Grinding wheel data includes grinding wheel characteristics and dressing conditions.		
Method of exclusion	1. Go to Common Variables to check if @1005 is outside the range of 0~1. 2. Restore the changed grinding wheel data or clear the interrupted grinding wheel dressing return amount of the processing monitoring.		

Alarm ID	MACRO-10107	Alarm title	【Start Offset is Not Set】
Description	The start X offset of the dresser is not set.		
Possible reason	The start X offset of the grind wheel dressing is not set.		
Method of exclusion	Resets the start X offset of the dresser .		

Alarm ID	MACRO-10108	Alarm title	【M Code List - M Code of the Trimmer is Not Set】
Description	M Code List - Dresser M code for shaft equipment is not set.		
Possible reason	The M code list does not set the dresser to start rotating the M code or the dresser to stop rotating the M code.		
Method of exclusion	Reset the dresser to start rotating M code or the dresser to stop rotating M code.		

Alarm ID	MACRO-10109	Alarm title	【Remaining Diameter or Remaining Width is Negative】
Description	The remaining diameter or remaining width of the grinding wheel is less than zero.		
Possible reason	The current diameter of the grinding wheel is less than the minimum diameter or the current width is less than the minimum width.		

Alarm ID	MACRO-10109	Alarm title	【Remaining Diameter or Remaining Width is Negative】
Method of exclusion	1. Check whether the current diameter or current width of the grinding wheel is correct. 2. Check whether the minimum diameter or minimum width of the grinding wheel is correct.		

Alarm ID	MACRO-10110	Alarm title	【Inclined Grinding Wheel Does Not Need to Be Formed】
Description	The current side length value is wrong.		
Possible reason	The current side length of the inclined grinding wheel is the same as the side length of the target contour, and no shaping is required.		
Method of exclusion	Resets the current side length or the target outline side length.		

Alarm ID	MACRO-10111	Alarm title	【Custom Grinding Wheel Does Not Set Any Infeed】
Description	Processing with a custom grinding wheel which does not set any infeed.		
Possible reason	X single infeed amount and Z single infeed amount are not set.		
Method of exclusion	Re-X single feed amount and Z single feed amount.		

Alarm ID	MACRO-10112	Alarm title	【Single Feed Amount Setting Error】
Description	The single feed amount is set incorrectly.		

SYNTEC

Alarm ID	MACRO-10112	Alarm title	【Single Feed Amount Setting Error】
Possible reason	1. Performing the rapid prototyping of the inclined grinding wheel (feeding the bottom edge) does not set the X single feed amount. 2. Performing the sharpening of flat grinding wheel, side grinding wheel and inclined grinding wheel without setting X single feed amount. 3. The X single feed amount is not set when performing roller grinding wheel dressing. 4. The Z single feed amount is not set when performing side grinding wheel rapid prototyping or sharpening. 5. The Z single feed amount is not set when performing oblique grinding wheel sharpening. 6. When performing trimming, the X/Z single feed amount reaches the upper limit of the feed and the parameter Pr4003 (open to check the single feed status) is 1.		
Method of exclusion	1. Reset the X single feed. 2. Reset the X single feed. 3. Reset the X single feed. 4. Reset the Z single feed. 5. Reset the Z single feed. 6. Reduce the X single feed or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4003 to 0 or 2.		
Alarm ID	MACRO-10114	Alarm title	【Speed Setting Above Limit】
Description	The speed setting is higher than the maximum speed.		
Possible reason	The grinding wheel speed setting exceeds the upper limit of the speed, and the parameter Pr4002 (open to check the maximum speed/circumferential speed status) is set to 1 or 2.		
Method of exclusion	Modify the grinding wheel speed setting for dressing or grinding, or go to the grinding wheel information to confirm the upper limit; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4002 to 0.		
Alarm ID	MACRO-10115	Alarm title	【Grinding Wheel Diameter is Smaller Than the Minimum Diameter】
Description	Performing a pre-compensation/pre-dressing system check found that the wheel diameter will be smaller than the minimum wheel diameter.		

Alarm ID	MACRO-10115	Alarm title	【Grinding Wheel Diameter is Smaller Than the Minimum Diameter】
Possible reason	The current diameter of the grinding wheel is close to the minimum diameter, and the parameter Pr4001 (open check minimum diameter/width status) is set to 1 or 2.		
Method of exclusion	Replace the grinding wheel and update the current diameter of the grinding wheel and the minimum diameter of the grinding wheel; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4001 to 0.		

Alarm ID	MACRO-10116	Alarm title	【Wheel Width is Smaller Than the Minimum Width】
Description	Performing a pre-compensation/pre-dressing system check found that the wheel width will be smaller than the minimum wheel width.		
Possible reason	The current width of the grinding wheel is close to the minimum width, and the parameter Pr4001 (open check minimum diameter/width status) is set to 1 or 2.		
Method of exclusion	Replace the grinding wheel and update the current width of the grinding wheel and the minimum width of the grinding wheel; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4001 to 0.		

Alarm ID	MACRO-10118	Alarm title	【The Number of Rough Dressing is Incorrectly Set】
Description	The number of rough dressings is incorrectly set.		
Possible reason	When the grinding wheel is sharpened or the roller grinding wheel is not set, the number of rough dressings or the number of rough dressings is equal to 0.		
Method of exclusion	Reset the number of rough dressings .		

13.2.2 MACRO-10001 File Operation Error

Alarm ID	MACRO-10001	Alarm title	【File Operation Error】
Description	An error occurred while operating the file.		

Alarm ID	MACRO-10001	Alarm title	【File Operation Error】
Possible reason	1. G302 is not executed before G303 is executed. 2. When using G302 to call the grinding wheel data, the grinding wheel with the specified number does not exist. 3. The grinding wheel characteristics and dressing condition files do not exist. 4. The grinding wheel list file does not exist.		
Method of exclusion	1. Check whether G302 is executed before the program executes G303. 2. Build the tool. 3. Delete the existing grinding wheel data and add new data.		

13.2.3 MACRO-10002 Grinding Wheel Tool Number Setting Error

Alarm ID	MACRO-10002	Alarm title	【Grinding Wheel Tool Number Setting Error】
Description	The tool number of the grinding wheel is set incorrectly.		
Possible reason	1. The tool number format of the program is wrong. 2. The tool number exceeds the set range of 1~15.		
Method of exclusion	1. Refer to the instruction of the T code to confirm whether the format of the tool number is two digits, for example, the first tool is 01, the third tool is 03, and the twelfth tool is 12. 2. Re-set the tool number value to be in the range of 1~15.		

13.2.4 MACRO-10003 Grinding Wheel Cutting Edge Setting Error

Alarm ID	MACRO-10003	Alarm title	【Grinding Wheel Cutting Edge Setting Error】
Description	The cutting edge of the grinding wheel is set incorrectly.		
Possible reason	1. The format of the cutting edge number of the program is wrong. 2. The cutting edge number exceeds the range of 1~6.		

Alarm ID	MACRO-10003	Alarm title	【Grinding Wheel Cutting Edge Setting Error】
Method of exclusion	1. Refer to the instruction of the T code to confirm whether the format of the cutting edge number is two digits, for example, cutting edge one is 01, cutting edge three is 03. 2. Re-set the cutting edge number to be in the range of 1~6.		

13.2.5 MACRO-10004 The Number of Pieces of Grinding Wheel Compensation is Incorrectly Set

Alarm ID	MACRO-10004	Alarm title	【The Number of Pieces of Grinding Wheel Compensation is Incorrectly Set】
Description	The number of pieces of grinding wheel compensation is incorrectly set.		
Possible reason	The number of pieces of grinding wheel wearing is set to less than 0。		
Method of exclusion	Reset the grinding wheel compensation piece counting to an integer greater than or equal to 0.		

13.2.6 MACRO-10005 The Wheel Rotation Function is Set Incorrectly

Alarm ID	MACRO-10005	Alarm title	【The Wheel Rotation Function is Set Incorrectly】
Description	When performing sand dressing or grinding wheel call, the setting of the grinding wheel rotation function condition is incorrect.		
Possible reason	1. The grinding wheel speed unit exceeds the range of 0~1. 2. The wheel speed is not set.		
Method of exclusion	1. Check if the wheel speed unit is 0 (M/S) or 1 (RPM). 2. Set the wheel speed.		

13.2.7 MACRO-10008 Incorrect Setting of Each Feed for Coarse Grinding

Alarm ID	**MACRO-10008**	**Alarm title**	**【Incorrect Setting of Each Feed for Coarse Grinding】**
Description	The setting of each feed (D) for coarse grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the coarse grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each coarse grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the coarse grinding feed amount (D) per time is larger than the coarse grinding amount (A).		
Method of exclusion	1. Modify the coarse grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the coarse grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the coarse grinding amount (A).		

13.2.8 MACRO-10009 Incorrect Setting of Each Feed for Middle Grinding

Alarm ID	**MACRO-10009**	**Alarm title**	**【Incorrect Setting of Each Feed for Middle Grinding】**
Description	The setting of each feed (E) of the middle grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the middle grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each middle grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the middle grinding feed amount (D) per time is larger than the coarse grinding amount (A).		
Method of exclusion	1. Modify the middle grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the middle grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the middle grinding amount (A).		

13.2.9 MACRO-10010 Incorrect Setting of Each Feed for Fine Grinding

Alarm ID	**MACRO-10010**	**Alarm title**	**【Incorrect Setting of Each Feed for Fine Grinding】**
Description	The setting of each feed (E) of the fine grinding is incorrect.		
Possible reason	1. When G201/G205/G207 is executed, the fine grinding feed amount is set and the parameter Pr4003 (open check tool maximum feed depth state) is set to 1 or 2. 2. When G201/G205/G207 is executed, the value of each fine grinding feed (D) is not set. 3. When G201/G205/G207 is executed, the fine grinding feed amount (D) per time is larger than the coarse grinding amount (A).		
Method of exclusion	1. Modify the fine grinding feed per time; or modify the grinding feed upper limit; or refer to the Special Parameter Manual of the Grinding Machine, and set the parameter Pr4003 to 0. 2. Reset the fine grinding feed (D) argument field. 3. Check whether the coarse grinding amount (D) is smaller than the fine grinding amount (A).		

13.2.10 MACRO-10011 Coarse Grinding Feed Rate is Not Set

Alarm ID	**MACRO-10011**	**Alarm title**	**【Coarse Grinding Feed Rate is Not Set】**
Description	The coarse grinding feed rate is not set.		
Possible reason	1. When the G201 command is executed, the coarse grinding feed rate(I) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the coarse grinding feed rate(D) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 coarse grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 coarse grinding feed rate(D).		

13.2.11 MACRO-10012 Middle Grinding Feed Rate is Not Set

Alarm ID	**MACRO-10012**	**Alarm title**	**【Middle Grinding Feed Rate is Not Set】**
Description	The middle grinding feed rate is not set.		

Alarm ID	MACRO-10012	Alarm title	【Middle Grinding Feed Rate is Not Set】
Possible reason	1. When the G201 command is executed, the middle grinding feed rate(P) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the middle grinding feed rate(E) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 middle grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 middle grinding feed rate(D).		

13.2.12 MACRO-10013 Fine Grinding Feed Rate is Not Set

Alarm ID	MACRO-10013	Alarm title	【Fine Grinding Feed Rate is Not Set】
Description	The fine grinding feed rate is not set.		
Possible reason	1. When the G201 command is executed, the fine grinding feed rate(I) value is not set. 2. When G202/G203/G206/G208/G209 commands are executed, the fine grinding feed rate(D) value is not set.		
Method of exclusion	1. Reset the parameter field of G201 fine grinding feed rate(I). 2. Reset the parameter field of G202/G203/G206/G208/G209 fine grinding feed rate(D).		

13.2.13 MACRO-10015 G202 Shoulder Coarse Grinding Speed is Not Set

Alarm ID	MACRO-10015	Alarm title	【G202 Shoulder Coarse Grinding Speed is Not Set】
Description	When using the G202 straight shoulder grinding function, the coarse grinding speed (H) of the shoulder is not set.		
Possible reason	The shoulder coarse grinding speed (H) value was not set when the G202 command was executed.		
Method of exclusion	Reset the shoulder coarse grinding speed (H) argument field.		

13.2.14 MACRO-10016 G202 Shoulder Fine Grinding Speed is Not Set

Alarm ID	**MACRO-10016**	**Alarm title**	**【G202 Shoulder Fine Grinding Speed is Not Set】**
Description	When using the G202 straight shoulder grinding function, the fine grinding speed (M) of the shoulder is not set.		
Possible reason	The shoulder fine grinding speed (M) value was not set when the G202 command was executed.		
Method of exclusion	Reset the shoulder fine grinding speed (M) argument field.		

13.2.15 MACRO-10018 G204 Condition Setting Error

Alarm ID	**MACRO-10018**	**Alarm title**	**【G204 Condition Setting Error】**
Description	When G204 is used for radius grinding, the grinding conditions are set incorrectly.		
Possible reason	1. No round radius (R) value specified. 2. An error occurs when the corner rounding process (H) or the value of it is not set.		
Method of exclusion	1. Reset the round radius (R) argument field. 2. Rerun the corner rounding process (H) parameter field or delete the operation and add the operation again.		

13.2.16 MACRO-10019 G206/G208 Z-direction Infeed Condition Error

Alarm ID	**MACRO-10019**	**Alarm title**	**【G206/G208 Z-direction Infeed Condition Error】**
Description	The conditions required for G206/G208 to execute Z-direction infeed are wrong or insufficient.		
Possible reason	1. When G206/208 command is executed, the value of feed overlap amount (I) is not specified. 2. The wheel currently has a width of 0 or is empty. 3. When G206/208 command is executed, the value of infeed overlap (I) is larger than the current width of the grinding wheel.		

Alarm ID	MACRO-10019	Alarm title	【G206/G208 Z-direction Infeed Condition Error】
Method of exclusion	1. Reset the infeed overlap (I) parameter field. 2. Reset the current width of the grinding wheel. 3. Reset the feed overlap amount or the current width of the grinding wheel		

13.2.17 MACRO-10020 G209 Left/right Shoulder Selection is Not Set

Alarm ID	MACRO-10018	Alarm title	【G209 Left/right Shoulder Selection is Not Set】
Description	When using the G209 shoulder straight grinding function, the left/right shoulder selection (W) is not set.		
Possible reason	When the G209 command is executed, the left/right shoulder selection (W) value is not set.		
Method of exclusion	Reset the left/right shoulder selection (W) argument field.		

13.2.18 MACRO-10021 G209 Arc Radius Setting Error

Alarm ID	MACRO-10021	Alarm title	【G209 Arc Radius Setting Error】
Description	When using the G209 shoulder straight grinding function, the arc radius setting (V) is incorrect.		
Possible reason	When G209 command is executed, the arc radius (V) value is not set or the arc radius setting (V) value is incorrect.		
Method of exclusion	Check that the arc radius (V) argument fields are all entered with numeric values or modified argument formats.		

13.2.19 MACRO-10022 G209 X-axis Dimension Second Coordinate Setting Error

Alarm ID	MACRO-10022	Alarm title	【MACRO-10022 G209 X-axis Dimension Second Coordinate Setting Error】
Description	When using the G209 shoulder straight grinding function, the second coordinate (U) of the X-axis dimension is set incorrectly.		

Alarm ID	MACRO-10022	Alarm title	【MACRO-10022 G209 X-axis Dimension Second Coordinate Setting Error】
Possible reason	When G209 command is executed, the value of the second coordinate (U) of the X-axis dimension is not set or the value of the second coordinate (U) of the X-axis dimension is set incorrectly.		
Method of exclusion	Check whether the second coordinate (U) argument field of the X-axis dimension has entered a value or corrected the argument format.		

13.2.20 MACRO-10023 Swing Speed is Not Set

Alarm ID	MACRO-10023	Alarm title	【Swing Speed is Not Set】
Description	When using the G205/G206/G207/G208 function, the swing speed (J) is not set.		
Possible reason	When the G205/G206/G207/G208 command is executed, the swing speed (J) value is not set.		
Method of exclusion	Reset the swing speed (J) argument field.		

13.2.21 MACRO-10024 Safe Point Return Method is Not Set

Alarm ID	MACRO-10024	Alarm title	【Safe Point Return Method is Not Set】
Description	The grinding safety point return method is not set.		
Possible reason	The settings are not made on the Grinding Safe Points page.		
Method of exclusion	Go to F3 Offset/Setting → F3 Workpiece Coordinate System → F5 Grinding Safety Point to set.		

13.2.22 MACRO-10025 The End Surface Measurement Result Exceeds the Error Threshold

Alarm ID	MACRO-10025	Alarm title	【The End Surface Measurement Result Exceeds the Error Threshold】
Description	The measurement error of general parts and standard parts exceeds the threshold.		
Possible reason	1. The size of the general piece is wrong. 2. The measurement result is different from the actual position of the workpiece end face.		
Method of exclusion	1. Check whether the size of the workpiece is correct or whether the position of the workpiece is correct. 2. Check whether the triggering of the end surface measurement tripping signal is normal, and check whether the C bit of the specified value of the signal source is triggered. For example, fill in 111 to check whether C111 is triggered.		

13.2.23 MACRO-10026 The End Surface Measurement Signal is Not Triggered

Alarm ID	MACRO-10026	Alarm title	【The End Surface Measurement Signal is Not Triggered】
Description	No signal is triggered when the measurement action is completed.		
Possible reason	1. The end surface measurement trip signal is not properly triggered. 2. The starting point of measurement is too far from the workpiece, and no signal is triggered after the first stage of measurement is performed.		
Method of exclusion	1. Check whether the C bit of the specified value of the end surface signal source is triggered, for example, fill in 111 to check whether C111 is triggered. 2. Modify the measurement starting point or increase the measurement distance of the first section to ensure that the distance between the end surface of workpieces and the measurement starting point is less than the first section measurement distance.		

13.2.24 MACRO-10027 M Code List - Q Code of Jump Signal is Not Set

Alarm ID	MACRO-10027	Alarm title	【M Code List - Q Code of Jump Signal is Not Set】
Description	M code list - jump signal is not set.		

Alarm ID	MACRO-10027	Alarm title	【M Code List - Q Code of Jump Signal is Not Set】
Possible reason	1. The M code list does not set the Q code of the end surface measurement trip signal. 2. The M code list does not set the Q code for the skip signal of coarse, middle and fine grinding.		
Method of exclusion	1. Reset the Q code of the end surface measurement trip signal. 2. Reset the Q code of the skip signal for coarse, middle and fine grinding.		

13.2.25 MACRO-10028 M Code List - M Code of Measuring Equipment is Not Set

Alarm ID	MACRO-10028	Alarm title	【M Code List - M Code of Measuring Equipment is Not Set】
Description	M code list - measuring equipment not set.		
Possible reason	1. The M code list does not set the workpiece end surface measurement to open M code or workpiece end surface measurement to close the M code. 2. The M code list does not set the outer diameter measurement on M code (1 or 2) or the outer diameter measurement off M code (1 or 2).		
Method of exclusion	1. Re-set workpiece end surface measurement to open M code or workpiece end surface measurement to close M code. 2. Re-set the outer diameter measurement to open M code (1 or 2) or to close the outer diameter measurement to M code (1 or 2).		

13.2.26 MACRO-10029 Problems Occurred in the Second Stage Measurement

Alarm ID	MACRO-10029	Alarm title	【Problems Occurred in the Second Stage Measurement】
Description	A problem occurred when reverting to the starting point of the second measurement		
Possible reason	1. The measurement speed of the first stage is too fast, resulting in a large gap between the coordinates recorded when the probe stops and the actual coordinates of the workpiece end surface, and the probe is still in the trigger state when it returns to the starting point of the second stage of measurement. 2. The measuring distance of the second stage is too short, so that the probe is still in the triggering state when returning to the starting point of the second stage.		

Alarm ID	MACRO-10029	Alarm title	【Problems Occurred in the Second Stage Measurement】
Method of exclusion	1. Reduce the measurement speed of the first stage, reduce the continuous moving distance after touching the end surface of the workpiece to the actual stop, so as to avoid the probe still in the triggering state after the second stage of measurement is returned. 2. Increase the measurement distance of the second stage to ensure that the retraction distance of the second stage measurement is greater than the moving distance between the touch end face and the actual stop during the first stage measurement, so as to prevent the probe from being triggered.		

13.2.27 MACRO-10030 Trouble Moving to the Measurement Start Point

Alarm ID	MACRO-10030	Alarm title	【Trouble Moving to the Measurement Start Point】
Description	There is a problem with the signal in the middle of moving to the measurement start point.		
Possible reason	The measurement signal is triggered midway to the measurement starting point.		
Method of exclusion	Check whether there is any obstacle in the way to the measurement starting point, and re-teach the measurement starting point position.		

13.2.28 MACRO-10031 M Code List - M Code of the Workpiece is Not Set

Alarm ID	MACRO-10031	Alarm title	【M Code List - M Code of Workpieces is Not Set】
Description	M code list - M code of the workpiece of the shaft equipment is not set.		
Possible reason	The M code list does not set the workpiece to start forwarding or reversing M code or to delay stopping M code.		
Method of exclusion	Reset the workpiece to start forwarding or reversing M code or to delay stopping M code.		

13.2.29 MACRO-10032 M Code List - M Code of Grinding Wheel is Not Set

Alarm ID	**MACRO-10032**	**Alarm title**	**【M Code List - M Code of Grinding Wheel is Not Set】**
Description	M code list - The grinding wheel of the shaft equipment is not set.		
Possible reason	The M code list does not set the grinding wheel to start rotating M code or the grinding wheel delay stopping m code.		
Method of exclusion	Reset the grinding wheel to start rotating M code or the grinding wheel to delay stopping M code.		

13.2.30 MACRO-10033 Non-grinder-specific Software, Please Contact the Original Factory for Assistance

Alarm ID	**MACRO-10033**	**Alarm title**	**【Non-grinder-specific Software, Please Contact the Original Factory for Assistance】**
Description	The software cannot be used without decryption.		
Possible reason	Option2 is not turned on in the system, and the grinder software cannot be used.		
Method of exclusion	Contact the original factory for assistance in decryption.		

13.2.31 MACRO-10100 Pr413 G92(G92.1) Coordinate Retention Mode Setting Error

Alarm ID	**MACRO-10100**	**Alarm title**	**【Pr413 G92(G92.1) Coordinate Retention Mode Setting Error】**
Description	Pr413 G92 (G92.1) coordinate system error after grind wheel dressing is interrupted due to the current set value of coordinate retention mode.		
Possible reason	Pr413 G92(G92.1) coordinate retention mode is set to 1 or 2.		
Method of exclusion	Pr413 G92 (G92.1) coordinate retention mode is set to 0.		

13.2.32 MACRO-10101 The Edge Number Compensation Flag is Out of Range

Alarm ID	MACRO-10101	Alarm title	【The Edge Number Compensation Flag is Out of Range】
Description	The advanced setting of grind wheel dressing conditions provides the user to set whether each cutting edge should be compensated after dressing. The setting value is found to be out of range in macro.		
Possible reason	The tool edge number compensation flag exceeds the range of 0~1.		
Method of exclusion	Check whether the tool edge number compensation flag (@230~@235) contains a value other than 0 and 1.		

13.2.33 MACRO-10103 Pr4004 Incorrect Spindle Number Specified

Alarm ID	MACRO-10103	Alarm title	【Pr4004 Incorrect Spindle Number Specified】
Description	The specified spindle number in Pr4004 is incorrectly set.		
Possible reason	The spindle number specified by Pr4004 is less than 1 or greater than 10.		
Method of exclusion	Refer to the grinder-specific parameter manual to check whether the spindle number specified by Pr4004 is between 1 and 10.		

13.2.34 MACRO-10105 The Trimming Method is Set Incorrectly

Alarm ID	MACRO-10105	Alarm title	【The Trimming Method is Set Incorrectly】
Description	G301 The trimming method is out of range.		
Possible reason	G301 The trimming method exceeds the range of 1, 3, and 4.		
Method of exclusion	Check whether the value of the trimming method exceeds the range of 1, 3, and 4.		

13.2.35 MACRO-10106 Grind Wheel Dressing Interruption Error

Alarm ID	MACRO-10106	Alarm title	【Fix Interruption Errors】
Description	The setting of the grinding wheel dressing return condition is incorrect.		
Possible reason	1. The flag (@1005) of whether the previous dressing has been interrupted exceeds the range of 0~1 (0: no previous interrupt occurred, 1: previous interrupt occurred) 2. The previous dressing was interrupted, this time using interrupted dressing (with input return amount), the system check found that the grinding wheel data were different. Grinding wheel data includes grinding wheel characteristics and dressing conditions.		
Method of exclusion	1. Go to Common Variables to check if @1005 is outside the range of 0~1. 2. Restore the changed grinding wheel data or clear the interrupted grinding wheel dressing return amount of the processing monitoring.		

13.2.36 MACRO-10107 Start Offset is Not Set

Alarm ID	MACRO-10107	Alarm title	【Start Offset is Not Set】
Description	The start X offset of the dresser is not set.		
Possible reason	The start X offset of the grind wheel dressing is not set.		
Method of exclusion	Resets the start X offset of the dresser .		

13.2.37 MACRO-10108 M Code List - M Code of the Trimmer is Not Set

Alarm ID	MACRO-10108	Alarm title	【M Code List - M Code of the Trimmer is Not Set】
Description	M Code List - Dresser M code for shaft equipment is not set.		
Possible reason	The M code list does not set the dresser to start rotating the M code or the dresser to stop rotating the M code.		
Method of exclusion	Reset the dresser to start rotating M code or the dresser to stop rotating M code.		

13.2.38 MACRO-10109 Remaining Diameter or Remaining Width is Negative

Alarm ID	**MACRO-10109**	**Alarm title**	**【Remaining Diameter or Remaining Width is Negative】**
Description	The remaining diameter or remaining width of the grinding wheel is less than zero.		
Possible reason	The current diameter of the grinding wheel is less than the minimum diameter or the current width is less than the minimum width.		
Method of exclusion	1. Check whether the current diameter or current width of the grinding wheel is correct. 2. Check whether the minimum diameter or minimum width of the grinding wheel is correct.		

13.2.39 MACRO-10110 Inclined Grinding Wheel Does Not Need to Be Formed

Alarm ID	**MACRO-10110**	**Alarm title**	**【Inclined Grinding Wheel Does Not Need to Be Formed】**
Description	The current side length value is wrong.		
Possible reason	The current side length of the inclined grinding wheel is the same as the side length of the target contour, and no shaping is required.		
Method of exclusion	Resets the current side length or the target outline side length.		

13.2.40 MACRO-10111 Custom Grinding Wheel Does Not Set Any Infeed

Alarm ID	**MACRO-10111**	**Alarm title**	**【Custom Grinding Wheel Does Not Set Any Infeed】**
Description	Processing with a custom grinding wheel which does not set any infeed.		
Possible reason	X single infeed amount and Z single infeed amount are not set.		
Method of exclusion	Re-X single feed amount and Z single feed amount.		

13.2.41 MACRO-10112 Single Feed Amount Setting Error

Alarm ID	MACRO-10112	Alarm title	【Single Feed Amount Setting Error】
Description	The single feed amount is set incorrectly.		
Possible reason	1. Performing the rapid prototyping of the inclined grinding wheel (feeding the bottom edge) does not set the X single feed amount. 2. Performing the sharpening of flat grinding wheel, side grinding wheel and inclined grinding wheel without setting X single feed amount. 3. The X single feed amount is not set when performing roller grinding wheel dressing. 4. The Z single feed amount is not set when performing side grinding wheel rapid prototyping or sharpening. 5. The Z single feed amount is not set when performing oblique grinding wheel sharpening. 6. When performing trimming, the X/Z single feed amount reaches the upper limit of the feed and the parameter Pr4003 (open to check the single feed status) is 1.		
Method of exclusion	1. Reset the X single feed. 2. Reset the X single feed. 3. Reset the X single feed. 4. Reset the Z single feed. 5. Reset the Z single feed. 6. Reduce the X single feed or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4003 to 0 or 2.		

13.2.42 MACRO-10114 Speed Setting Above Limit

Alarm ID	MACRO-10114	Alarm title	【Speed Setting Above Limit】
Description	The speed setting is higher than the maximum speed.		
Possible reason	The grinding wheel speed setting exceeds the upper limit of the speed, and the parameter Pr4002 (open to check the maximum speed/circumferential speed status) is set to 1 or 2.		
Method of exclusion	Modify the grinding wheel speed setting for dressing or grinding, or go to the grinding wheel information to confirm the upper limit; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4002 to 0.		

13.2.43 MACRO-10115 Grinding Wheel Diameter is Smaller Than the Minimum Diameter

Alarm ID	MACRO-10115	Alarm title	【Grinding Wheel Diameter is Smaller Than the Minimum Diameter】
Description	Performing a pre-compensation/pre-dressing system check found that the wheel diameter will be smaller than the minimum wheel diameter.		
Possible reason	The current diameter of the grinding wheel is close to the minimum diameter, and the parameter Pr4001 (open check minimum diameter/width status) is set to 1 or 2.		
Method of exclusion	Replace the grinding wheel and update the current diameter of the grinding wheel and the minimum diameter of the grinding wheel; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4001 to 0.		

13.2.44 MACRO-10116 Wheel Width is Smaller Than the Minimum Width

Alarm ID	MACRO-10116	Alarm title	【Wheel Width is Smaller Than the Minimum Width】
Description	Performing a pre-compensation/pre-dressing system check found that the wheel width will be smaller than the minimum wheel width.		
Possible reason	The current width of the grinding wheel is close to the minimum width, and the parameter Pr4001 (open check minimum diameter/width status) is set to 1 or 2.		
Method of exclusion	Replace the grinding wheel and update the current width of the grinding wheel and the minimum width of the grinding wheel; or refer to the Grinder-specific Parameter Manual, and set the parameter Pr4001 to 0.		

13.2.45 MACRO-10118 The Number of Rough Dressings is Incorrectly Set

Alarm ID	MACRO-10118	Alarm title	【The Number of Rough Dressing is Incorrectly Set】
Description	The number of rough dressings is incorrectly set.		
Possible reason	When the grinding wheel is sharpened or the roller grinding wheel is not set, the number of rough dressings or the number of rough dressings is equal to 0.		

Alarm ID	MACRO-10118	Alarm title	【The Number of Rough Dressing is Incorrectly Set】
Method of exclusion	Reset the number of rough dressings .		

13.3 Macro Program Alarm - MACRO - High Gloss Glass

Alarm ID	MACRO-5xx	Alarm title	【Zx-Txx Tool Life Expired】
Description	1. Suitable for high gloss glass. 2. The tool life has expired and the group tool cannot be found in the tool magazine. 3. MACRO-501~524 corresponds to Z1-T1~T24 **tool life expired**; MACRO-525~548 corresponds to Z2-T1~T24 **tool life expired**; So on and so forth.		
Possible reasons	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found.		
Methods of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero. 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established.		

Alarm ID	MACRO-645	Alarm title	【the Life of Multiple Tools Has Expired, Please Go to the Tool Management Function to Check】
Description	1. Suitable for high gloss glass. 2. The tool life of two or more tools with the same tool number on different axes has expired, and the group tool cannot be found in the tool magazine.		
Possible reasons	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found.		
Methods of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero. 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established.		

Alarm ID	MACRO-20001	Alarm title	【Tool Number Setting Cannot Be Zero】
Description	Input wrong tool number		
Possible reasons	Tool number is set to 0		
Methods of exclusion	Set the correct tool number		

Alarm ID	MACRO-20002	Alarm title	【Z1 Coupling Flag Cannot Be Zero】
Description	Input error Z1 coupling flag		
Possible reasons	The Z1 coupling flag is set to 0		
Methods of exclusion	Set the correct Z1 coupling flag		

Alarm ID	MACRO-20003	Alarm title	【The Quick Tool Setting Function Cannot Be Used if the Tool Setting Action Has Not Been Performed】
Description	Because the quick tool setting needs to have the tool surface position information (tool length), the tool needs to perform the tool setting action (automatic tool setting or tool breakage detection)		
Possible reasons	Knife setting has not been performed		
Methods of exclusion	Perform automatic tool setting		

Alarm ID	MACRO-20004	Alarm title	【the Measurement Speed Cannot Be Zero】
Description	Input error measuring speed		
Possible reasons	The measurement speed is 0		
Methods of exclusion	Set the correct measurement speed		

Alarm ID	MACRO-20014	Alarm title	【the Multi-z Tool Table is Not Turned on, So the Multi-z Automatic Tool Setting Function Cannot Be Executed】
Description	You need to open the tool table to get the tool length information		
Possible reasons	Multi-Z tool table is not turned on		
Methods of exclusion	Set Pr3816 to 3		

Alarm ID	MACRO-20021~20026	Alarm title	【Zx Tool Length Measurement Error Value is Too Large】
Description	Check "2nd Tool Setting" in advanced adjustment, if the difference between the two measured tool lengths is greater than 0.01mm, it means that the measurement error is too large		
Possible reasons	1. Tool setting speed is too fast 2. There are obstacles		
Methods of exclusion	1. Slow down the tool setting speed 2. Remove obstacles		

Alarm ID	MACRO-21008~21504	Alarm title	【Zx Starting Point Position Setting Error】
Description	The starting point position is set incorrectly, that is, the tool setting action cannot be performed		
Possible reasons	1. The position of the starting point is lower than the surface position of the tool detector 2. There are obstacles		
Methods of exclusion	1. Set the correct starting point position 2. Check if #1608 is expected 3. Remove obstacles		

Alarm ID	MACRO-23008~23504	Alarm title	【Zx Lowest Point Position Setting Error】
Description	The lowest point position is set incorrectly, that is, the tool setting action cannot be performed		
Possible reasons	1. Z1 trip signal source setting error 2. The lowest point position is higher than the surface position of the tool detector 3. There are obstacles 4. Hardware wiring error		
Methods of exclusion	1. Reset Z1 Trip Source 2. Set the correct starting point position 3. Check if #1608 is expected 4. Remove obstacles 5. Check hardware for wiring errors		

Alarm ID	MACRO-25008~25504	Alarm title	【There is a Tool on the Zx Axis, and the Return of the Tool Fails】
Description	There is a tool on the spindle, that is, the return of the tool fails		
Possible reasons	The tool return action is not confirmed		

Alarm ID	MACRO-25008~25504	Alarm title	【There is a Tool on the Zx Axis, and the Return of the Tool Fails】
Methods of exclusion	Perform the tool return action		

Alarm ID	MACRO-27008~27504	Alarm title	【Zx Current Tool is Seriously Worn or Broken (Tool Needs to Be Changed)】
Description	After the tool breakage detection, it is found that the Z1~Z6 tool is broken		
Possible reasons	1. Z1~Z6 tool breakage 2. Tool setting speed is too fast 3. There are obstacles 4. The allowable value of tool length error is improperly set (ex: not set, the default value is 0)		
Methods of exclusion	1. Replace Z1~Z6 tools 2. Slow down the tool setting speed 3. Remove obstacles 4. Correct the allowable value of tool length error		

13.3.1 Macro-645 the Life of Multiple Tools Has Expired, Please Go to the Tool Management Function to Check

Alarm ID	MACRO-645	Alarm title	**【the Life of Multiple Tools Has Expired, Please Go to the Tool Management Function to Check】**
Description	1. Suitable for high gloss glass. 2. The tool life of two or more tools with the same tool number on different axes has expired, and the group tool cannot be found in the tool magazine.		
Possible reasons	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found.		

Alarm ID	MACRO-645	Alarm title	【the Life of Multiple Tools Has Expired, Please Go to the Tool Management Function to Check】
Methods of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero. 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established.		

13.3.2 MACRO-501~644 Zx-Txx Tool Life Expired

Alarm ID	MACRO-5xx	Alarm title	【Zx-Txx Tool Life Expired】
Description	1. Suitable for high gloss glass. 2. The tool life has expired and the group tool cannot be found in the tool magazine. 3. MACRO-501~524 corresponds to Z1-T1~T24 **tool life expired**; MACRO-525~548 corresponds to Z2-T1~T24 **tool life expired**; So on and so forth.		
Possible reasons	1. The tool life management has been turned on, the used tool life has expired, and the group tool cannot be found.		
Methods of exclusion	1. Enter the tool life management page and reset the current life of the tool life management to zero. 2. Or enter the tool life management page to make the "current life < maximum life" condition of the tool established.		

13.3.3 MACRO-20001 Tool Number Setting Cannot Be Zero

Alarm ID	MACRO-20001	Alarm title	【Tool Number Setting Cannot Be Zero】
Description	Input wrong tool number		
Possible reasons	Tool number is set to 0		
Methods of exclusion	Set the correct tool number		

13.3.4 MACRO-20002 Z1 Coupling Flag Cannot Be Zero

Alarm ID	**MACRO-20002**	**Alarm title**	【Z1 Coupling Flag Cannot Be Zero】
Description	Input error Z1 coupling flag		
Possible reasons	The Z1 coupling flag is set to 0		
Methods of exclusion	Set the correct Z1 coupling flag		

13.3.5 MACRO-20003 The Quick Tool Setting Function Cannot Be Used if the Tool Setting Action Has Not Been Performed

Alarm ID	**MACRO-20003**	**Alarm title**	【The Quick Tool Setting Function Cannot Be Used if the Tool Setting Action Has Not Been Performed】
Description	Because the quick tool setting needs to have the tool surface position information (tool length), the tool needs to perform the tool setting action (automatic tool setting or tool breakage detection)		
Possible reasons	Knife setting has not been performed		
Methods of exclusion	Perform automatic tool setting		

13.3.6 MACRO-20004 The Measurement Speed Cannot Be Zero

Alarm ID	**MACRO-20004**	**Alarm title**	【the Measurement Speed Cannot Be Zero】
Description	Input error measuring speed		
Possible reasons	The measurement speed is 0		

Alarm ID	MACRO-20004	Alarm title	【the Measurement Speed Cannot Be Zero】
Methods of exclusion	Set the correct measurement speed		

13.3.7 MACRO-20014 The Multi-z Tool Table is Not Turned on, So the Multi-z Automatic Tool Setting Function Cannot Be Executed

Alarm ID	MACRO-20014	Alarm title	【the Multi-z Tool Table is Not Turned on, So the Multi-z Automatic Tool Setting Function Cannot Be Executed】
Description	You need to open the tool table to get the tool length information		
Possible reasons	Multi-Z tool table is not turned on		
Methods of exclusion	Set Pr3816 to 3		

13.3.8 MACRO-20021-20026 Zx Tool Length Measurement Error Value is Too Large

Alarm ID	MACRO-20021~20026	Alarm title	【Zx Tool Length Measurement Error Value is Too Large】
Description	Check "2nd Tool Setting" in advanced adjustment, if the difference between the two measured tool lengths is greater than 0.01mm, it means that the measurement error is too large		
Possible reasons	1. Tool setting speed is too fast 2. There are obstacles		
Methods of exclusion	1. Slow down the tool setting speed 2. Remove obstacles		

13.3.9 MACRO-21008-21504 Zx Starting Point Position Setting Error

Alarm ID	**MACRO-21008~21504**	**Alarm title**	【Zx Starting Point Position Setting Error】
Description	The starting point position is set incorrectly, that is, the tool setting action cannot be performed		
Possible reasons	1. The position of the starting point is lower than the surface position of the tool detector 2. There are obstacles		
Methods of exclusion	1. Set the correct starting point position 2. Check if #1608 is expected 3. Remove obstacles		

13.3.10 MACRO-23008-23504 Zx Lowest Point Position Setting Error

Alarm ID	**MACRO-23008~23504**	**Alarm title**	【Zx Lowest Point Position Setting Error】
Description	The lowest point position is set incorrectly, that is, the tool setting action cannot be performed		
Possible reasons	1. Z1 trip signal source setting error 2. The lowest point position is higher than the surface position of the tool detector 3. There are obstacles 4. Hardware wiring error		
Methods of exclusion	1. Reset Z1 Trip Source 2. Set the correct starting point position 3. Check if #1608 is expected 4. Remove obstacles 5. Check hardware for wiring errors		

13.3.11 MACRO-25008-25504 There is a Tool on the Zx Axis, and the Return of the Tool Fails

Alarm ID	**MACRO-25008~25504**	**Alarm title**	【There is a Tool on the Zx Axis, and the Return of the Tool Fails】
Description	There is a tool on the spindle, that is, the return of the tool fails		
Possible reasons	The tool return action is not confirmed		
Methods of exclusion	Perform the tool return action		

13.3.12 MACRO-27008-27504 Zx Current Tool is Seriously Worn or Broken (Tool Needs to Be Changed)

Alarm ID	**MACRO-27008~27504**	**Alarm title**	【Zx Current Tool is Seriously Worn or Broken (Tool Needs to Be Changed)】
Description	After the tool breakage detection, it is found that the Z1~Z6 tool is broken		
Possible reasons	1. Z1~Z6 tool breakage 2. Tool setting speed is too fast 3. There are obstacles 4. The allowable value of tool length error is improperly set (ex: not set, the default value is 0)		
Methods of exclusion	1. Replace Z1~Z6 tools 2. Slow down the tool setting speed 3. Remove obstacles 4. Correct the allowable value of tool length error		

14 Device Alarm - STATION

Alarm ID	STATION-01h	Alarm title	【M3 Information abnormal: function command error】
Description	M3 information communication error, the drive cannot support this function command (Function code).		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-02h	Alarm title	【M3 Information exception: memory location error】
Description	M3 information communication error, access to invalid memory location.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-03h	Alarm title	【M3 Information exception: data length error】
Description	M3 informationcommunication error, the length of the accessed data does not match the size of the specified memory space.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		

Alarm ID	STATION-03h	Alarm title	【M3 Information exception: data length error】
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-04h	Alarm title	【M3 Information exception: data type error】
Description	M3 information communication error, the drive cannot support the specified data type.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-05h	Alarm title	【M3 Information exception: access authority error】
Description	M3 information communication error, no permission to access the specified memory space.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-06h	Alarm title	【M3 Information exception: data setting error】
Description	M3 information communication error, the set data size exceeds the allowable range.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		
Alarm ID	STATION-07h	Alarm title	【M3 Information exception: data range selection error】
Description	M3 information communication error, the selected data range is not a complete memory space, and data access is not possible.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		
Alarm ID	STATION-08h	Alarm title	【M3 Information exception: state environment error】
Description	M3 information communication error, the current state environment of the drive cannot execute this command.		
Possible Causes	1. The current servo status cannot execute this command. 2. The communication between the controller and the drive is unstable, resulting in a communication packet error. 3. The driver version does not support this information communication packet.		

Alarm ID	STATION-08h	Alarm title	【M3 Information exception: state environment error】
Solution	1. Try again after changing the servo state (Servo On/Off). 2. Replace the cable and make sure the communication between the controller and the driver is stable. 3. Update the driver firmware version. 4. Update the controller software version. 5. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-09h	Alarm title	【M3 Information exception: program processing conflicts】
Description	M3 information communication error, this command conflicts with the currently executing program.		
Possible Causes	1. The current servo status cannot execute this command. 2. The communication between the controller and the drive is unstable, resulting in a communication packet error. 3. The driver version does not support this information communication packet.		
Solution	1. Abort the command other axis are executing. 2. Replace the cable and make sure the communication between the controller and the driver is stable. 3. Update the driver firmware version. 4. Update the controller software version. 5. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-41h	Alarm title	【FRAM read/write failed】
Description	External FRAM read and write failed.		
Possible Causes	The external FRAM is corrupted, causing the program fail to write/read normally.		
Solution	Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-42h	Alarm title	【Preamp FPGA cannot be used】
Description	Pre-amp FPGA is not available		
Possible Causes	There is an error in the program in the preamplifier FPGA, which makes the FPGA unusable.		

Alarm ID	STATION-42h	Alarm title	【Preamp FPGA cannot be used】
Solution	Reburn preamp firmware, after burning completed reboot the driver.		

Alarm ID	STATION-43h	Alarm title	【Burning failure: the driver cannot perform burning in servo enabled state】
Description	Cannot enter burning mode while the drive is still in the servo-on state.		
Possible Causes	There is servo axis in the servo-on state, which makes it impossible to enter the burning mode.		
Solution	Re-perform burning after confirming that all servo axis in connection is in servo-off state.		

Alarm ID	STATION-44h	Alarm title	【DSP program check error】
Description	DSP program check error.		
Possible Causes	The firmware in the drive's preamp is incomplete.		
Solution	Reburn preamp firmware, upon burning completion reboot the driver.		

Alarm ID	STATION-45h	Alarm title	【DSP firmware version check error】
Description	DSP firmware version check error.		
Possible Causes	The firmware version in the drive's preamp is incompatible.		
Solution	Reburn preamp firmware, upon burning completion reboot the driver.		

Alarm ID	STATION-46h	Alarm title	【The driver needs to be recharged】
Description	The drive needs to be powered back on.		
Possible Causes	After the drive firmware has been burned, the drive needs to be rebooted.		
Solution	Reboot the driver.		

Alarm ID	STATION-47h	Alarm title	【Module ID Data Error】
Description	Reading module ID data error		
Possible Causes	1. Module ID data error causing by memory ageing or communication interfered. 2. Any of module number, extend card number, add-on card number is over range.		
Solution	1. Check wiring, especially if shielding is connected to ground correctly. Then reboot the drive. 2. Check the drive, IO extend card, and add-on card are official version. 3. Send back to Syntec.		

Alarm ID	STATION-51h	Alarm title	【Burning failure: packet count is not continuous】
Description	Packet count is not continuous.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. After reboot controller and driver, re-burn firmware. 3. Re-burn the driver with driver tuning software.		

Alarm ID	STATION-52h	Alarm title	【Burning failure: missing packet search failed】
Description	Try to re-search for missing packets, but an error occurred during the search.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed. After the communication packet is missing, try to search for the missing packet data again, but the search fails.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. After reboot the controller and driver, then re-burn the driver. 3. Re-burn the driver with driver tuning software.		

Alarm ID	STATION-61h	Alarm title	【Burning failure: missing packet recording failed】
Description	An error occurred while recording the relevant environmental information of the missing package.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed. After the communication packet is missing, try to record the environmental information of the missing packet, but an error occurs in the record.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Reboot the controller and the driver and re-burn. 3. Re-burn with the driver tuning software.		

Alarm ID	STATION-71h	Alarm title	【Burning failure: invalid command】
Description	The controller sent the wrong burn command.		
Possible Causes	Drive firmware version does not support this burn command.		
Solution	Re-burn after updating the driver firmware version.		

Alarm ID	STATION-72h	Alarm title	【Burning failure: failed to perform burn initialization】
Description	The burn initialization is not performed before the drive starts burning.		
Possible Causes	The communication between the controller and the drive is unstable, causing the initialization command to be missed.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Re-burn with the driver tuning software. 3. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-73h	Alarm title	【Burning failure: failed to notify file transmission is complete】
Description	After the burn file transfer completed, the drive has not been notified that the file transfer has been completed.		

Alarm ID	STATION-73h	Alarm title	【Burning failure: failed to notify file transmission is complete】
Possible Causes	The communication between the controller and the drive is unstable, and the notification command for the completion of the file transfer is missing.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Re-burn with the driver tuning software. 3. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-81h	Alarm title	【Burning failure: The element is damaged or non-existent】
Description	Component is damaged or does not exist.		
Possible Causes	The programming component is damaged, making it impossible to perform the burn operation.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-82h	Alarm title	【Burning failure: memory space cannot be erased】
Description	Unable to erase memory space (Flash).		
Possible Causes	Memory space cannot be erased when the drive performs initialization.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-83h	Alarm title	【Burning failure: data size does not match memory space】
Description	Data size does not match memory space.		
Possible Causes	The file size of the burning installation package does not match the drive memory space.		

Alarm ID	STATION-83h	Alarm title	【Burning failure: data size does not match memory space】
Solution	1. Confirm whether the installation package is damaged. Re-download the package and re-burn. 2. Verify that the content of the installation package is compatible with the hardware module in the driver. 3. Re-burn with the driver tuning software.		

Alarm ID	STATION-84h	Alarm title	【Burning failure: write address error】
Description	Write in wrong address .		
Possible Causes	The module address of the burning installation package does not match the module configuration in the drive.		
Solution	1. Confirm whether the installation package is damaged. Re-download the package and re-burn. 2. Verify that the content of the installation package is compatible with the hardware module in the driver. 3. Re-burn with the driver tuning software.		

Alarm ID	STATION-85h	Alarm title	【Burning failure: error in memory writing】
Description	An error occurred while writing to the memory.		
Possible Causes	Memory write failure occurred during burning.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-86h	Alarm title	【Burning failure: data packet error】
Description	At the end of the burning process, the drive checks that the file size does not match the expected.		
Possible Causes	The communication between the controller and the drive is unstable, resulting in a communication packet error.		

Alarm ID	**STATION-86h**	**Alarm title**	**【Burning failure: data packet error】**
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Confirm whether the installation package is damaged. Re-downloading the package and re-burn. 3. Re-burn with the driver tuning software.		
Alarm ID	**STATION-87h**	**Alarm title**	**【Burning failure: file check error】**
Description	An error occurred while the drive was checking the contents of burn installation package.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The file content of the **burning** installation package is damaged.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Confirm whether the installation package is damaged. Re-download the package and re-burn. 3. Re-burn with the driver tuning software.		

Alarm ID	**STATION-100h**	**Alarm title**	**【Burning failure: Drive communication timeout】**
Description	Drive communication timeout during the burning process.		
Possible Causes	1. The M3 drive does not support M3 message communication. 2. Communication between controller and driver is not stable, causing the M3 drive message or EtherCAT drive mailbox communication takes too much time.		
Solution	1. If it is M3 network, confirm that the drive can support M3 message communication. 2. Replace the cable and make sure the communication between the controller and the drive is stable. 3. Update the drive firmware version.		

Alarm ID	**STATION-101h**	**Alarm title**	**【Burning failure: Burning can only be conducted if the servo axis is in servo disabled state】**
Description	The controller must enter the burn mode when all servo axes are in the servo-off state.		

Alarm ID	STATION-101h	Alarm title	【Burning failure: Burning can only be conducted if the servo axis is in servo disabled state】
Possible Causes	The controller still has the servo axis in the servo-on state, so the drive burn mode cannot be connected.		
Solution	Re-burn after confirming that the all connected servo axis is in servo-off state.		

Alarm ID	STATION-102h	Alarm title	【Communication failure: the driver is in burn mode】
Description	The drive is in burn mode and cannot provide servo and I/O functions on the drive side.		
Possible Causes	1. After the drive firmware burned, the drive did not reboot. 2. The driver preamp firmware is incomplete and cannot enable servo and I/O functions.		
Solution	1. Reboot the driver. 2. Reburn preamp firmware before reboot the driver. 3. Re-burn preamp firmware with the driver tuning software. 4. Check if the driver hardware is malfunctioned or damaged.		

Alarm ID	STATION-103h	Alarm title	【Burning failure: abnormal interruption occurred during the previous burning】
Description	Burn failed. The abnormality occurred during the previous burn process, causing the process to be interrupted abnormally and the new burning operation could not be performed.		
Possible Causes	An abnormality occurred during the previous burn process and the burning action was interrupted abnormally.		
Solution	1. Re-burn after reboot the controller and driver. 2. Re-burn with the driver tuning software.		

Alarm ID	STATION-104h	Alarm title	【Burning failure: error in checking the burning environment】
Description	Burning failed, error occurred while checking the burning environment.		
Possible Causes	The internal file of the controller is damaged, resulting in the inability to perform the programming action.		

Alarm ID	STATION-104h	Alarm title	【Burning failure: error in checking the burning environment】
Solution	1. Re-burn with the driver tuning software. 2. Update the controller software version. 3. If the problem persists, please contact the distributor or the Vendor for repair.		

Alarm ID	STATION-105h	Alarm title	【Burning failure: the drive cannot enter burn mode】
Description	The burning failed, the drive cannot enter the burning mode.		
Possible Causes	The driver firmware version does not support burn files through the controller.		
Solution	Re-burn with the driver tuning software.		

Alarm ID	STATION-106h	Alarm title	【The controller cannot identify the device information of the driver】
Description	The controller does not recognize the drive device information of this version.		
Possible Causes	Controller software version is too old to recognize this version’s drive device information.		
Solution	Update the controller software version		

Alarm ID	STATION-107h	Alarm title	【The version of the driver boot loader does not support remote burning】
Description	The boot loader version of the drive is too old to support firmware burning via the controller.		
Possible Causes	The drive loader version of the drive is too old.		
Solution	1. Re-burn with the driver tuning software. 2. Contact the Vendor to update the driver boot loader.		

14.1 STATION-01h M3 Information abnormal: function command error

Alarm ID	STATION-01h	Alarm title	【M3 Information abnormal: function command error】
Description	M3 information communication error, the drive cannot support this function command (Function code).		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.2 STATION-02h M3 Information exception: memory location error

Alarm ID	STATION-02h	Alarm title	【M3 Information exception: memory location error】
Description	M3 information communication error, access to invalid memory location.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.3 STATION-03h M3 Information exception: data length error

Alarm ID	STATION-03h	Alarm title	【M3 Information exception: data length error】
Description	M3 informationcommunication error, the length of the accessed data does not match the size of the specified memory space.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.4 STATION-04h M3 Information exception: data type error

Alarm ID	STATION-04h	Alarm title	【M3 Information exception: data type error】
Description	M3 information communication error, the drive cannot support the specified data type.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.5 STATION-05h M3 Information exception: access authority error

Alarm ID	STATION-05h	Alarm title	【M3 Information exception: access authority error】
Description	M3 information communication error, no permission to access the specified memory space.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.6 STATION-06h M3 Information exception: data setting error

Alarm ID	STATION-06h	Alarm title	【M3 Information exception: data setting error】
Description	M3 information communication error, the set data size exceeds the allowable range.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.7 STATION-07h M3 Information exception: data range selection error

Alarm ID	STATION-07h	Alarm title	【M3 Information exception: data range selection error】
Description	M3 information communication error, the selected data range is not a complete memory space, and data access is not possible.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The driver firmware version does not support this information communication packet.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Update the driver firmware version. 3. Update the controller software version. 4. If the problem persists, please contact the distributor or the Vendor for repair.		

14.8 STATION-08h M3 Information exception: state environment error

Alarm ID	STATION-08h	Alarm title	【M3 Information exception: state environment error】
Description	M3 information communication error, the current state environment of the drive cannot execute this command.		
Possible Causes	1. The current servo status cannot execute this command. 2. The communication between the controller and the drive is unstable, resulting in a communication packet error. 3. The driver version does not support this information communication packet.		
Solution	1. Try again after changing the servo state (Servo On/Off). 2. Replace the cable and make sure the communication between the controller and the driver is stable. 3. Update the driver firmware version. 4. Update the controller software version. 5. If the problem persists, please contact the distributor or the Vendor for repair.		

14.9 STATION-09h M3 Information exception: program processing conflicts

Alarm ID	STATION-09h	Alarm title	【M3 Information exception: program processing conflicts】
Description	M3 information communication error, this command conflicts with the currently executing program.		
Possible Causes	1. The current servo status cannot execute this command. 2. The communication between the controller and the drive is unstable, resulting in a communication packet error. 3. The driver version does not support this information communication packet.		
Solution	1. Abort the command other axis are executing. 2. Replace the cable and make sure the communication between the controller and the driver is stable. 3. Update the driver firmware version. 4. Update the controller software version. 5. If the problem persists, please contact the distributor or the Vendor for repair.		

14.10 STATION-41h FRAM read/write failed

Alarm ID	STATION-41h	Alarm title	【FRAM read/write failed】
Description	External FRAM read and write failed.		
Possible Causes	The external FRAM is corrupted, causing the program fail to write/read normally.		
Solution	Check if the driver hardware is malfunctioned or damaged.		

14.11 STATION-42h Preamp FPGA cannot be used

Alarm ID	STATION-42h	Alarm title	【Preamp FPGA cannot be used】
Description	Pre-amp FPGA is not available		
Possible Causes	There is an error in the program in the preamplifier FPGA, which makes the FPGA unusable.		

Alarm ID	STATION-42h	Alarm title	【Preamp FPGA cannot be used】
Solution	Reburn preamp firmware, after burning completed reboot the driver.		

14.12 STATION-43h Burning failure: the driver cannot perform burning in servo enabled state

Alarm ID	STATION-43h	Alarm title	【Burning failure: the driver cannot perform burning in servo enabled state】
Description	Cannot enter burning mode while the drive is still in the servo-on state.		
Possible Causes	There is servo axis in the servo-on state, which makes it impossible to enter the burning mode.		
Solution	Re-perform burning after confirming that all servo axis in connection is in servo-off state.		

14.13 STATION-44h DSP program check error

Alarm ID	STATION-44h	Alarm title	【DSP program check error】
Description	DSP program check error.		
Possible Causes	The firmware in the drive's preamp is incomplete.		
Solution	Reburn preamp firmware, upon burning completion reboot the driver.		

14.14 STATION-45h DSP firmware version check error

Alarm ID	STATION-45h	Alarm title	【DSP firmware version check error】
Description	DSP firmware version check error.		
Possible Causes	The firmware version in the drive's preamp is incompatible.		
Solution	Reburn preamp firmware, upon burning completion reboot the driver.		

14.15 STATION-46h The driver needs to be recharged

Alarm ID	STATION-46h	Alarm title	【The driver needs to be recharged】
Description	The drive needs to be powered back on.		
Possible Causes	After the drive firmware has been burned, the drive needs to be rebooted.		
Solution	Reboot the driver.		

14.16 STATION-47h Module ID Data Error

Alarm ID	STATION-47h	Alarm title	【Module ID Data Error】
Description	Reading module ID data error		
Possible Causes	1. Module ID data error causing by memory ageing or communication interfered. 2. Any of module number, extend card number, add-on card number is over range.		
Solution	1. Check wiring, especially if shielding is connected to ground correctly. Then reboot the drive. 2. Check the drive, IO extend card, and add-on card are official version. 3. Send back to Syntec.		

14.17 STATION-51h Burning failure: packet count is not continuous

Alarm ID	STATION-51h	Alarm title	【Burning failure: packet count is not continuous】
Description	Packet count is not continuous.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. After reboot controller and driver, re-burn firmware. 3. Re-burn the driver with driver tuning software.		

14.18 STATION-52h Burning failure: missing packet search failed

Alarm ID	STATION-52h	Alarm title	【Burning failure: missing packet search failed】
Description	Try to re-search for missing packets, but an error occurred during the search.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed. After the communication packet is missing, try to search for the missing packet data again, but the search fails.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. After reboot the controller and driver, then re-burn the driver. 3. Re-burn the driver with driver tuning software.		

14.19 STATION-61h Burning failure: missing packet recording failed

Alarm ID	STATION-61h	Alarm title	【Burning failure: missing packet recording failed】
Description	An error occurred while recording the relevant environmental information of the missing package.		
Possible Causes	The communication between the controller and the drive is unstable, causing the communication packet to be missed. After the communication packet is missing, try to record the environmental information of the missing packet, but an error occurs in the record.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Reboot the controller and the driver and re-burn. 3. Re-burn with the driver tuning software.		

14.20 STATION-71h Burning failure: invalid command

Alarm ID	STATION-71h	Alarm title	【Burning failure: invalid command】
Description	The controller sent the wrong burn command.		

Alarm ID	STATION-71h	Alarm title	【Burning failure: invalid command】
Possible Causes	Drive firmware version does not support this burn command.		
Solution	Re-burn after updating the driver firmware version.		

14.21 STATION-72h Burning failure: failed to perform burn initialization

Alarm ID	STATION-72h	Alarm title	【Burning failure: failed to perform burn initialization】
Description	The burn initialization is not performed before the drive starts burning.		
Possible Causes	The communication between the controller and the drive is unstable, causing the initialization command to be missed.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Re-burn with the driver tuning software. 3. Check if the driver hardware is malfunctioned or damaged.		

14.22 STATION-73h Burning failure: failed to notify file transmission is complete

Alarm ID	STATION-73h	Alarm title	【Burning failure: failed to notify file transmission is complete】
Description	After the burn file transfer completed, the drive has not been notified that the file transfer has been completed.		
Possible Causes	The communication between the controller and the drive is unstable, and the notification command for the completion of the file transfer is missing.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Re-burn with the driver tuning software. 3. Check if the driver hardware is malfunctioned or damaged.		

14.23 STATION-81h Burning failure: The element is damaged or non-existent

Alarm ID	STATION-81h	Alarm title	【Burning failure: The element is damaged or non-existent】
Description	Component is damaged or does not exist.		
Possible Causes	The programming component is damaged, making it impossible to perform the burn operation.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

14.24 STATION-82h Burning failure: memory space cannot be erased

Alarm ID	STATION-82h	Alarm title	【Burning failure: memory space cannot be erased】
Description	Unable to erase memory space (Flash).		
Possible Causes	Memory space cannot be erased when the drive performs initialization.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

14.25 STATION-83h Burning failure: data size does not match memory space

Alarm ID	STATION-83h	Alarm title	【Burning failure: data size does not match memory space】
Description	Data size does not match memory space.		
Possible Causes	The file size of the burning installation package does not match the drive memory space.		
Solution	1. Confirm whether the installation package is damaged. Re-download the package and re-burn. 2. Verify that the content of the installation package is compatible with the hardware module in the driver. 3. Re-burn with the driver tuning software.		

14.26 STATION-84h Burning failure: write address error

Alarm ID	STATION-84h	Alarm title	【Burning failure: write address error】
Description	Write in wrong address .		
Possible Causes	The module address of the burning installation package does not match the module configuration in the drive.		
Solution	1. Confirm whether the installation package is damaged. Re-download the package and re-burn. 2. Verify that the content of the installation package is compatible with the hardware module in the driver. 3. Re-burn with the driver tuning software.		

14.27 STATION-85h Burning failure: error in memory writing

Alarm ID	STATION-85h	Alarm title	【Burning failure: error in memory writing】
Description	An error occurred while writing to the memory.		
Possible Causes	Memory write failure occurred during burning.		
Solution	1. Re-burn with the driver tuning software. 2. Check if the driver hardware is malfunctioned or damaged.		

14.28 STATION-86h Burning failure: data packet error

Alarm ID	STATION-86h	Alarm title	【Burning failure: data packet error】
Description	At the end of the burning process, the drive checks that the file size does not match the expected.		
Possible Causes	The communication between the controller and the drive is unstable, resulting in a communication packet error.		

Alarm ID	STATION-86h	Alarm title	【Burning failure: data packet error】
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Confirm whether the installation package is damaged. Re-downloading the package and re-burn. 3. Re-burn with the driver tuning software.		

14.29 STATION-87h Burning failure: file check error

Alarm ID	STATION-87h	Alarm title	【Burning failure: file check error】
Description	An error occurred while the drive was checking the contents of burn installation package.		
Possible Causes	1. The communication between the controller and the drive is unstable, resulting in a communication packet error. 2. The file content of the **burning** installation package is damaged.		
Solution	1. Replace the cable and make sure the communication between the controller and the driver is stable. 2. Confirm whether the installation package is damaged. Re-download the package and re-burn. 3. Re-burn with the driver tuning software.		

14.30 STATION-100h Burning failure: Drive communication timeout

Alarm ID	STATION-100h	Alarm title	【Burning failure: Drive communication timeout】
Description	Drive communication timeout during the burning process.		
Possible Causes	1. The M3 drive does not support M3 message communication. 2. Communication between controller and driver is not stable, causing the M3 drive message or EtherCAT drive mailbox communication takes too much time.		
Solution	1. If it is M3 network, confirm that the drive can support M3 message communication. 2. Replace the cable and make sure the communication between the controller and the drive is stable. 3. Update the drive firmware version.		

14.31 STATION-101h Burning failure: Burning can only be conducted if the servo axis is in servo disabled state

Alarm ID	STATION-101h	Alarm title	【Burning failure: Burning can only be conducted if the servo axis is in servo disabled state】
Description	The controller must enter the burn mode when all servo axes are in the servo-off state.		
Possible Causes	The controller still has the servo axis in the servo-on state, so the drive burn mode cannot be connected.		
Solution	Re-burn after confirming that the all connected servo axis is in servo-off state.		

14.32 STATION-102h Communication failure: the driver is in burn mode

Alarm ID	STATION-102h	Alarm title	【Communication failure: the driver is in burn mode】
Description	The drive is in burn mode and cannot provide servo and I/O functions on the drive side.		
Possible Causes	1. After the drive firmware burned, the drive did not reboot. 2. The driver preamp firmware is incomplete and cannot enable servo and I/O functions.		
Solution	1. Reboot the driver. 2. Reburn preamp firmware before reboot the driver. 3. Re-burn preamp firmware with the driver tuning software. 4. Check if the driver hardware is malfunctioned or damaged.		

14.33 STATION-103h Burning failure: abnormal interruption occurred during the previous burning

Alarm ID	STATION-103h	Alarm title	【Burning failure: abnormal interruption occurred during the previous burning】
Description	Burn failed. The abnormality occurred during the previous burn process, causing the process to be interrupted abnormally and the new burning operation could not be performed.		

Alarm ID	STATION-103h	Alarm title	【Burning failure: abnormal interruption occurred during the previous burning】
Possible Causes	An abnormality occurred during the previous burn process and the burning action was interrupted abnormally.		
Solution	1. Re-burn after reboot the controller and driver. 2. Re-burn with the driver tuning software.		

14.34 STATION-104h Burning failure: error in checking the burning environment

Alarm ID	STATION-104h	Alarm title	【Burning failure: error in checking the burning environment】
Description	Burning failed, error occurred while checking the burning environment.		
Possible Causes	The internal file of the controller is damaged, resulting in the inability to perform the programming action.		
Solution	1. Re-burn with the driver tuning software. 2. Update the controller software version. 3. If the problem persists, please contact the distributor or the Vendor for repair.		

14.35 STATION-105h Burning failure: the drive cannot enter burn mode

Alarm ID	STATION-105h	Alarm title	【Burning failure: the drive cannot enter burn mode】
Description	The burning failed, the drive cannot enter the burning mode.		
Possible Causes	The driver firmware version does not support burn files through the controller.		
Solution	Re-burn with the driver tuning software.		

14.36 STATION-106h The controller cannot identify the device information of the driver

Alarm ID	STATION-106h	Alarm title	【The controller cannot identify the device information of the driver】
Description	The controller does not recognize the drive device information of this version.		
Possible Causes	Controller software version is too old to recognize this version's drive device information.		
Solution	Update the controller software version		

14.37 STATION-107h The version of the driver boot loader does not support remote burning

Alarm ID	STATION-107h	Alarm title	【The version of the driver boot loader does not support remote burning】
Description	The boot loader version of the drive is too old to support firmware burning via the controller.		
Possible Causes	The drive loader version of the drive is too old.		
Solution	1. Re-burn with the driver tuning software. 2. Contact the Vendor to update the driver boot loader.		

SYNTEC

15 Mechanical Logic Control Alarm - MLC

Mechanical Logic Control Alarm-MLC	
001	R40.0: X axis exceeds the positive hardware stroke limit.
002	R40.1: X axis exceeds the negative hardware stroke limit.
003	R40.2: X axis has not yet returned to origin.
004	R40.3: X axis servo driver alarm.
009	R40.8: Y axis exceeds the positive hardware stroke limit.
010	R40.9: Y axis exceeds the negative hardware stroke limit.
011	R40.10: Y axis has not yet returned to the origin.
012	R40.11: Y axis servo driver alarm.
017	R41.0: Z axis exceeds the positive hardware stroke limit.
018	R41.1: Z axis exceeds the negative hardware stroke limit.
019	R41.2: Z axis has not yet returned to the origin.
020	R41.3: Z axis servo driver alarm.
025	R41.8: A axis exceeds the positive hardware stroke limit.
026	R41.9: A axis exceeds the negative hardware stroke limit.
027	R41.10: A axis has not yet returned to the origin.
028	R41.11: A axis servo driver alarm.
033	R42.0: B axis exceeds the positive hardware stroke limit.
034	R42.1: B axis exceeds the negative hardware stroke limit.

Mechanical Logic Control Alarm-MLC	
035	R42.2: B axis has not yet returned to the origin.
036	R42.3: B axis servo driver alarm.
041	R42.8: C axis exceeds the positive hardware stroke limit.
042	R42.9: C axis exceeds the negative hardware stroke limit.
043	R42.10: C axis has not yet returned to the origin.
044	R42.11: C axis servo driver alarm.
049	R43.0: Spindle servo driver alarm.
050	R43.1: cutting fluid motor overload.
051	R43.2: hydraulic oil pressure system alarm.
052	R43.3: insufficient air pressure.
053	R43.4: insufficient lubricant oil.
054	R43.5: insufficient cutting fluid level.
055	R43.6: the gripper not clamped.
056	R43.7: tool magazine system alarm.
057	CPU temperature is too high, more than 80 degrees. (10.116.24Y, 10.116.36F, 10.116.40, 10.117.40 and before) CPU temperature is too high, more than 100 degrees. (10.116.24Y, 10.116.36F,10.116.40, 10.117.40 and above) (only supports x86/i.MX motherboard.)
058	Insufficient remaining system memory.
299	R43.16: cutting fluid motor overload.

Mechanical Logic Control Alarm-MLC	
300	R43.17: hydraulic oil pressure system alarm.
301	R43.18: insufficient air pressure.
302	R43.19: insufficient lubricant oil.
303	R43.20: insufficient cutting fluid level.
304	R43.21: CPU temperature is over 80 degrees. (only supports x86/i.MX motherboard.)
313	R43.30: Protective Start activated, press SIMU FWD to move to Target Pt. (Remarks: This alarm is triggered by Robot Products only.)
314	R43.31: Protective Start complete, press START to run program. (Remarks: This alarm is triggered by Robot Products only.)

1. MLC-001~MLC-048 are axis alarms (8 alarms/axis), and they are also arranged by aixs ID
   e.g.
   MLC-001~MLC-008 are alarms of NO. 1 axis
   MLC-009~MLC-016 are alarms of NO. 2 axis
   ....
   MLC-041~MLC-048 are alarms of NO. 6 axis
2. If some axis is not opened, the corresponding alarms will not be issued.
   e.g.
   No.5 axis is not opened(Pr25 is set by 0)， then MLC-033~040 will not be issued.
3. If some axis name doesn't match the corresponding alarm string in the above list, then user should define their own alarm string.
   e.g.
   Pr321【No.1 axis name】is set by 200(axis-y), and Pr322【No.2 axis name】is set by 100(axis-x).
   The alarm string of MLC-001~MLC-008 should be modified as "Y axis"
   The alarm string of MLC-009~MLC-016 should be modified as "X axis"

SYNTEC

16 Laser Marking Alarm - LASERCTRL

Alarm ID	**LASERCTRL-001**	**Alarm title**	**【Exceeding the software process limit】**
Description	The target position exceeds the stroke limit setting range.		
Possible Cause	1. The galvanometer is intended to travel beyond the stroke limit. 2. The galvanometer parameter setting is incorrect.		
Solution	1. Modify target location. 2. Set correct galvanometer parameters and set correct resolution and scanning angles as per Laser Marking PC Software Manual.		
Alarm ID	**LASERCTRL-002**	**Alarm title**	**【Absolute coordinates are invalid】**
Description	The absolute coordinate setting is invalid.		
Possible Cause	After origin is set, reset the mechanism parameters.		
Solution	Please contact the staff of the machine manufacturer to reset the origin.		
Alarm ID	**LASERCTRL-003**	**Alarm title**	**【The laser axis/galvanometer axis control function cannot be started】**
Description	The galvanometer axis or laser axis does not activated properly.		
Possible Cause	1. In general, this action will not be triggered by any operation. 2. The possible cause is that the software does not match the firmware.		
Solution	Please consult the Vendor.		
Alarm ID	**LASERCTRL-005**	**Alarm title**	**【Laser file loading failed】**
Description	The laser machining file cannot be turned on correctly.		
Possible Cause	The file name, file placement folder or machining file header is set incorrectly.		

Alarm ID	LASERCTRL-005	Alarm title	【Laser file loading failed】
Solution	1. Verify that the machining file name is saved under the machining file root directory. 2. Make sure the file name is entered correctly. 3. Confirm that the machining files are saved under the machining root file directory. 4. Check if the controller supports the version of the laser processing file. 5. Check if the version number of the laser machining file supports the currently enabled marking mode. 6. If the above method does not work, please contact the machine manufacturer.		

Alarm ID	LASERCTRL-006	Alarm title	【Illegal repeated interpretations number】
Description	The number of repeated interpretation machining paths is set incorrectly.		
Possible Cause	R17018 is set to 1 (limited times of repeatedly interpreted machining paths), and R17019 (repeatedly interpreted machining paths) is set to zero or negative.		
Solution	Set the number of R17019 reinterpretation machining paths to a positive integer.		

Alarm ID	LASERCTRL-007	Alarm Title	【The memory configuration of the repeated machining path is abnormal】
Description	The memory required to repeatedly interpret the machining path cannot be configured correctly.		
Possible Cause	When R17018 is set to 1, 2 or 3 (repeated processing path mode, or record mode for small files), and any of the following occurs: 1. The laser machining file is too large. 2. Exceed the max. machining time under the record mode for small files. 3. Use the machining file generated by the old version of CADCAM software.		
Solution	1. Reduce the file size of the machining path file. 2. Please confirm that the machining file has been generated using the new version of CADCAM software. 3. Please contact the Vendor.		

Unable to render include or excerpt-include. Could not retrieve page.

Alarm ID	LASERCTRL-021	Alarm title	【Coordinate synchronization error】
Description	Coordinate synchronization failed.		
Possible Cause	When the coordinates are synchronized, there are still commands not issued.		
Solution	Re-trigger reset. The reset clears the command and synchronize coordinates.		

Alarm ID	LASERCTRL-022	Alarm title	【Distortion correction table out of range】
Description	The galvanometer position table checking failed.		
Possible Cause	The target point is out of range and the table check fails.		
Solution	1. Trigger Reset controller. 2. Change the target position to be within the range of the distortion correction table. 3. Check whether the displacement and zoom of the calibration table are set. If set, the scope of the calibration table will be changed, causing the target position to be out of the calibration range. 4. Refer to Laser Marking PC Software Manual to expand the scope of the distortion correction table.		

Alarm ID	LASERCTRL-024	Alarm title	【Correction table header error】
Description	The distortion correction header has an error.		
Possible Cause	The calibration table version does not match the controller version.		
Solution	1. Confirm that the controller version is consistent with the marking software version. 2. Refer to Laser Marking PC Software Manual for recalibration.		

Alarm ID	LASERCTRL-026	Alarm title	【Laser machining file content error】
Description	Incorrect laser machining file content.		
Possible Cause	The laser machining file has incorrect command.		

Alarm ID	LASERCTRL-026	Alarm title	【Laser machining file content error】
Solution	1. Reconvert the machining files 2. Please contact the staff of the machine manufacturer.		

Alarm ID	LASERCTRL-027	Alarm title	【XY axis command beyond the distortion correction table range】
Description	The X and Y axis commands are outside the range of the distortion correction table.		
Possible Cause	The X and Y axis commands are outside the range of the distortion correction table.		
Solution	1. Trigger Reset to reset the controller. 2. Change the XY axis target position of the pattern to be within the range of the distortion correction table. 3. Check whether the overall shift and rotate value(R1241~R1244) and galvo shift and rotate value(R17001~R17004) is zero, or lower the shift rotate value to let pattern inside the calibration range. 4. Check whether the displacement, zoom, and process limit of the calibration table are set. If set, the scope of the calibration table will be changed, causing the XY axis target position to be out of the calibration range. 5. Refer to Laser Marking PC Software Manual; check the galvanometer parameters settings to uncheck the stroke limit and use a full calibration table size; if this alarm is still activated when using the complete distortion correction table, it is necessary to recalibrate a range to meet the calibration table with machining process. 6. When synchronization marking is enabled, refer to Laser Synchronization Marking Operation Manual, and confirm the setting value of laser control parameters Pr61~Pr62.		

Alarm ID	LASERCTRL-028	Alarm title	【Z axis command beyond the distortion correction table range】
Description	The Z axis command is outside the range of the distortion correction table.		
Possible Cause	The Z axis command is outside the range of the distortion correction table.		
Solution	1. Trigger Reset to reset the controller. 2. Change the Z axis target position to be within the range of the distortion correction table. 3. Refer to the Laser Marking PC Software Manual to expand the scope of the distortion correction table. 4. When synchronization marking is enabled, refer to Laser Synchronization Marking Operation Manual, and confirm the setting value of laser control parameter Pr63.		

Alarm ID	LASERCTRL-029	Alarm title	【3D laser marking function is not supported】
Description	The option software Option35 "3D Laser Marking" is not activated, and 3D laser machining files is not supported.		
Possible Cause	The option software Option35 "3D Laser Marking" is not activated, and 3D laser machining files is not supported.		
Solution	1. Purchase Option35 for 3D laser marking. 2. For only 2D laser marking, the Z axis instruction of laser processing files must be zero.		

Alarm ID	LASERCTRL-030	Alarm title	【The distortion correction table does not exist】
Description	Distortion correction table does not exist.		
Possible Cause	Initial processing but the distortion correction table does not exist.		
Solution	1. Trigger Reset to reset the controller. 2. Re-import the distortion correction table.		

Alarm ID	LASERCTRL-041	Alarm Title	【Illegal usage in record mode or play mode】
Description	There is a problem of illegal operation in record mode or play mode.		
Possible Cause	1. Try to do the synchronous marking or the tracking marking, when R17018 is set to 3 or 4. 2. Try to switch R17018 under the state of laser running.		
Solution	1. R17018 must be set to 0 in the synchronous marking and the tracking marking. 2. Do not switch R17018 under the state of laser running.		

Alarm ID	LASERCTRL-042	Alarm Title	【The specific mode is not supported】
Description	The specific mode is not supported.		
Possible Cause	1. Try to use the record/ play feature for small files with 70SB.		

Solution	1. R17018 must be set to 0, 1 or 2, since this model does not support the record/ play mode for small files.

16.1 LASERCTRL-001 Exceeding the software process limit

Alarm ID	**LASERCTRL-001**	**Alarm title**	**【Exceeding the software process limit】**
Description	The target position exceeds the stroke limit setting range.		
Possible Cause	1. The galvanometer is intended to travel beyond the stroke limit. 2. The galvanometer parameter setting is incorrect.		
Solution	1. Modify target location. 2. Set correct galvanometer parameters and set correct resolution and scanning angles as per Laser Marking PC Software Manual.		

16.2 LASERCTRL-002 Absolute coordinates are invalid

Alarm ID	**LASERCTRL-002**	**Alarm title**	**【Absolute coordinates are invalid】**
Description	The absolute coordinate setting is invalid.		
Possible Cause	After origin is set, reset the mechanism parameters.		
Solution	Please contact the staff of the machine manufacturer to reset the origin.		

16.3 LASERCTRL-003 The laser axis/galvanometer axis control function cannot be started

Alarm ID	**LASERCTRL-003**	**Alarm title**	**【The laser axis/galvanometer axis control function cannot be started】**
Description	The galvanometer axis or laser axis does not activated properly.		
Possible Cause	1. In general, this action will not be triggered by any operation. 2. The possible cause is that the software does not match the firmware.		

Alarm ID	LASERCTRL-003	Alarm title	【The laser axis/galvanometer axis control function cannot be started】
Solution	Please consult the Vendor.		

16.4 LASERCTRL-005 Laser file loading failed

Alarm ID	LASERCTRL-005	Alarm title	【Laser file loading failed】
Description	The laser machining file cannot be turned on correctly.		
Possible Cause	The file name, file placement folder or machining file header is set incorrectly.		
Solution	1. Verify that the machining file name is saved under the machining file root directory. 2. Make sure the file name is entered correctly. 3. Confirm that the machining files are saved under the machining root file directory. 4. Check if the controller supports the version of the laser processing file. 5. Check if the version number of the laser machining file supports the currently enabled marking mode. 6. If the above method does not work, please contact the machine manufacturer.		

16.5 LASERCTRL-006 Illegal repeated interpretations number

Alarm ID	LASERCTRL-006	Alarm title	【Illegal repeated interpretations number】
Description	The number of repeated interpretation machining paths is set incorrectly.		
Possible Cause	R17018 is set to 1 (limited times of repeatedly interpreted machining paths), and R17019 (repeatedly interpreted machining paths) is set to zero or negative.		
Solution	Set the number of R17019 reinterpretation machining paths to a positive integer.		

16.6 LASERCTRL-007 The memory configuration of the repeated machining path is abnormal

Alarm ID	LASERCTRL-007	Alarm Title	【The memory configuration of the repeated machining path is abnormal】
Description	The memory required to repeatedly interpret the machining path cannot be configured correctly.		
Possible Cause	When R17018 is set to 1, 2 or 3 (repeated processing path mode, or record mode for small files), and any of the following occurs: 1. The laser machining file is too large. 2. Exceed the max. machining time under the record mode for small files. 3. Use the machining file generated by the old version of CADCAM software.		
Solution	1. Reduce the file size of the machining path file. 2. Please confirm that the machining file has been generated using the new version of CADCAM software. 3. Please contact the Vendor.		

16.7 LASERCTRL-008 2D simultaneous marking is not supported

Alarm ID	LASERCTRL-008	Alarm title	【2D simultaneous marking is not supported】
Description	The software optional function Option39 "2D simultaneous marking" is not activated, and the 2D simultaneous marking machining file is not supported.		
Possible Cause	Use the simultaneous marking function without Option 39. During machining, it was detected that two or more axis had a displacement in the mechanical command within first three axis of CNC. Use overall shift rotate R value(R1241~R1244) when only has 1D simultaneous marking, leads to two axis movement in flying marking.		

Solution	1. To use the 2D simultaneous marking function, please purchase Option39. 2. If use overall shift rotate R value(R1241~R1244), please purchase Option39. If the shift rotate value is small or only has same direction shift without rotate, consider using laser parameter Pr61~63 to close galvo axis to syncronize with CNC axis. 3. If you do not use the 2D simultaneous marking function, please use the correct marking machining file.

16.8 LASERCTRL-021 Coordinate synchronization error

Alarm ID	**LASERCTRL-021**	**Alarm title**	**【Coordinate synchronization error】**
Description	Coordinate synchronization failed.		
Possible Cause	When the coordinates are synchronized, there are still commands not issued.		
Solution	Re-trigger reset. The reset clears the command and synchronize coordinates.		

16.9 LASERCTRL-022 Distortion correction table out of range

Alarm ID	**LASERCTRL-022**	**Alarm title**	**【Distortion correction table out of range】**
Description	The galvanometer position table checking failed.		
Possible Cause	The target point is out of range and the table check fails.		
Solution	1. Trigger Reset controller. 2. Change the target position to be within the range of the distortion correction table. 3. Check whether the displacement and zoom of the calibration table are set. If set, the scope of the calibration table will be changed, causing the target position to be out of the calibration range. 4. Refer to Laser Marking PC Software Manual to expand the scope of the distortion correction table.		

16.10 LASERCTRL-024 Correction table header error

Alarm ID	LASERCTRL-024	Alarm title	【Correction table header error】
Description	The distortion correction header has an error.		
Possible Cause	The calibration table version does not match the controller version.		
Solution	1. Confirm that the controller version is consistent with the marking software version. 2. Refer to Laser Marking PC Software Manual for recalibration.		

16.11 LASERCTRL-026 Laser machining file content error

Alarm ID	LASERCTRL-026	Alarm title	【Laser machining file content error】
Description	Incorrect laser machining file content.		
Possible Cause	The laser machining file has incorrect command.		
Solution	1. Reconvert the machining files 2. Please contact the staff of the machine manufacturer.		

16.12 LASERCTRL-027 XY axis command beyond the distortion correction table range

Alarm ID	LASERCTRL-027	Alarm title	【XY axis command beyond the distortion correction table range】
Description	The X and Y axis commands are outside the range of the distortion correction table.		
Possible Cause	The X and Y axis commands are outside the range of the distortion correction table.		

Alarm ID	LASERCTRL-027	Alarm title	【XY axis command beyond the distortion correction table range】
Solution	1. Trigger Reset to reset the controller. 2. Change the XY axis target position of the pattern to be within the range of the distortion correction table. 3. Check whether the overall shift and rotate value(R1241~R1244) and galvo shift and rotate value(R17001~R17004) is zero, or lower the shift rotate value to let pattern inside the calibration range. 4. Check whether the displacement, zoom, and process limit of the calibration table are set. If set, the scope of the calibration table will be changed, causing the XY axis target position to be out of the calibration range. 5. Refer to Laser Marking PC Software Manual; check the galvanometer parameters settings to uncheck the stroke limit and use a full calibration table size; if this alarm is still activated when using the complete distortion correction table, it is necessary to recalibrate a range to meet the calibration table with machining process. 6. When synchronization marking is enabled, refer to Laser Synchronization Marking Operation Manual, and confirm the setting value of laser control parameters Pr61~Pr62.		

16.13 LASERCTRL-028 Z axis command beyond the distortion correction table range

Alarm ID	LASERCTRL-028	Alarm title	【Z axis command beyond the distortion correction table range】
Description	The Z axis command is outside the range of the distortion correction table.		
Possible Cause	The Z axis command is outside the range of the distortion correction table.		
Solution	1. Trigger Reset to reset the controller. 2. Change the Z axis target position to be within the range of the distortion correction table. 3. Refer to the Laser Marking PC Software Manual to expand the scope of the distortion correction table. 4. When synchronization marking is enabled, refer to Laser Synchronization Marking Operation Manual, and confirm the setting value of laser control parameter Pr63.		

16.14 LASERCTRL-029 3D laser marking function is not supported

Alarm ID	LASERCTRL-029	Alarm title	【3D laser marking function is not supported】
Description	The option software Option35 "3D Laser Marking" is not activated, and 3D laser machining files is not supported.		
Possible Cause	The option software Option35 "3D Laser Marking" is not activated, and 3D laser machining files is not supported.		
Solution	1. Purchase Option35 for 3D laser marking. 2. For only 2D laser marking, the Z axis instruction of laser processing files must be zero.		

16.15 LASERCTRL-030 The distortion correction table does not exist

Alarm ID	LASERCTRL-030	Alarm title	【The distortion correction table does not exist】
Description	Distortion correction table does not exist.		
Possible Cause	Initial processing but the distortion correction table does not exist.		
Solution	1. Trigger Reset to reset the controller. 2. Re-import the distortion correction table.		

16.16 LASERCTRL-041 Illegal usage in record mode or play mode

Alarm ID	LASERCTRL-041	Alarm Title	【Illegal usage in record mode or play mode】
Description	There is a problem of illegal operation in record mode or play mode.		
Possible Cause	1. Try to do the synchronous marking or the tracking marking, when R17018 is set to 3 or 4. 2. Try to switch R17018 under the state of laser running.		
Solution	1. R17018 must be set to 0 in the synchronous marking and the tracking marking. 2. Do not switch R17018 under the state of laser running.		

16.17 LASERCTRL-042 The specific mode is not supported

Alarm ID	LASERCTRL-042	Alarm Title	【The specific mode is not supported】
Description	The specific mode is not supported.		
Possible Cause	1. Try to use the record/ play feature for small files with 70SB.		
Solution	1. R17018 must be set to 0, 1 or 2, since this model does not support the record/ play mode for small files.		

SYNTEC

17 CIP Alarm - CIP

Alarm ID	**CIP-001**	**Alarm Title**	**【Connection failure】**
Description	A connection related service failed along the connection path.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-002**	**Alarm Title**	**【Resource unavailable】**
Description	Resources needed for the object to perform the requested service were unavailable.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-003**	**Alarm Title**	**【Invalid parameter value】**
Description	See Status Code 0x20, which is the preferred value to use for this condition.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-004**	**Alarm Title**	**【Path segment error】**
Description	The path segment identifier or the segment syntax was not understood by the processing node. Path processing shall stop when a path segment error is encountered.		
Possible Cause	The processed node does not understand the path paragraph identifier or paragraph syntax. When the path is wrong, the processing node will stop the path processing.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-005**	**Alarm Title**	**【Path destination unknown】**

Description	The path is referencing an object class, instance or structure element that is not known or is not contained in the processing node. Path processing shall stop when a path destination unknown error is encountered.
Possible Cause	This path uses an object Class, Instance or Structure item that is unknown or not included in the processing node. When the path target is unknown, the processing node stops the path processing.
Solution	Please refer to the CIP manual specifications.

Alarm ID	**CIP-006**	**Alarm Title**	**【Partial transfer】**
Description	Only part of the expected data was transferred.		
Possible Cause	There is not enough information required for the service.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	**CIP-007**	**Alarm Title**	**【Connection lost】**
Description	The messaging connection was lost.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	**CIP-008**	**Alarm Title**	**【Service not supported】**
Description	The requested service was not implemented or was not defined for this Object Class/ Instance.		
Possible Cause	The Service code of the instruction is not implemented or undefined in the corresponding object Class/Instance.		
Solution	Please use the object Class/Instance to have a implemented or defined Service code.		

Alarm ID	**CIP-009**	**Alarm Title**	**【Invalid Attribute value】**
Description	Invalid Attribute data detected.		
Possible Cause	The specified Attribute data is non-compliance.		

Solution	According to the description of the Attribute, give the correct value.		
Alarm ID	**CIP-010**	**Alarm Title**	**【Attribute list error】**
Description	An attribute in the Get_Attribute_List or Set_Attribute_List response has a non-zero status.		
Possible Cause	When using the Get_Attribute_List (0x03) or Set_Attribute_List (0x04) service code, if an access to an Attribute data is unsuccessful, this alarm will be issued.		
Solution	According to the access specification of the object related to the Attribute list, confirm the following specifications: 1. Whether the specified Attribute can be obtained or set. 2. Whether the set Attribute value is correct.		
Alarm ID	**CIP-011**	**Alarm Title**	**【Already in requested mode/state】**
Description	The object is already in the mode/state being requested by the service.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-012**	**Alarm Title**	**【Object state conflict】**
Description	The object cannot perform the requested service in its current mode/state.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-013**	**Alarm Title**	**【Object already exists】**
Description	The requested instance of object to be created already exists.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-014**	**Alarm Title**	**【Attribute not settable】**

Description	A request to modify a non-modifiable attribute was received.		
Possible Cause	Try to set an Attribute that cannot be modified.		
Solution	Do not set Attributes that cannot be modified.		
Alarm ID	**CIP-015**	**Alarm Title**	**【Privilege violation】**
Description	A permission/privilege check failed.		
Possible Cause	The permission/rights check failed.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-016**	**Alarm Title**	**【Device state conflict】**
Description	The device’s current mode/state prohibits the execution of the requested service.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-017**	**Alarm Title**	**【Reply data too large】**
Description	The data to be transmitted in the response buffer is larger than the allocated response buffer.		
Possible Cause	The amount of data responded is greater than the size of the response buffer configuration.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-018**	**Alarm Title**	**【Fragmentation of a primitive value 】**
Description	The service specified an operation that is going to fragment a primitive data value, i.e. half a REAL data type.		
Possible Cause	This service operation splits the basic data values, for example: half REAL data type.		
Solution	The service cannot split the data values of the basic type.		

Alarm ID	CIP-019	Alarm Title	【Not enough data】
Description	The service did not supply enough data to perform the specified operation.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-020	Alarm Title	【Attribute not supported】
Description	The attribute specified in the request is not supported.		
Possible Cause	The Attribute code of the command is unimplemented or undefined in the corresponding object Class/Instance.		
Solution	Please use the object Class/Instance to have an Attribute code with implementation or definition.		

Alarm ID	CIP-021	Alarm Title	【Too much data】
Description	The service supplied more data than was expected.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-022	Alarm Title	【object does not exist】
Description	The object specified does not exist in the device.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-023	Alarm Title	【Service fragmentation sequence not in progress】
Description	The fragmentation sequence for this service is not currently active for this data.		
Possible Cause	The target node failed to receive the information tag correctly.		

Alarm ID	CIP-023	Alarm Title	【Service fragmentation sequence not in progress】
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-024	Alarm Title	【No stored attribute data】
Description	The attribute data of this object was not saved prior to the requested service.		
Possible Cause	The Attribute data for this object was not stored until the service was requested.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-025	Alarm Title	【Store operation failure】
Description	The attribute data of this object was not saved due to a failure during the attempt.		
Possible Cause	An error occurred in the process of storing the Attribute data to target node.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-026	Alarm Title	【Routing failure, request packet too large】
Description	The service request packet was too large for transmission on a network in the path to the destination. The routing device was forced to abort the service.		
Possible Cause	The command packet of the service is too large to be transmitted on the network path to the destination, and the routing device is forced to terminate the service.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-027	Alarm Title	【Routing failure, response packet too large】
Description	The service response packet was too large for transmission on a network in the path from the destination. The routing device was forced to abort the service.		
Possible Cause	The response packet of the service is too large to be transmitted from the target network path, and the routing device is forced to terminate the service.		

Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-028**	**Alarm Title**	**【Missing attribute list entry data】**
Description	The service did not supply an attribute in a list of attributes that was needed by the service to perform the requested behavior.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-029**	**Alarm Title**	**【Invalid Attribute Value List】**
Description	The service is returning the list of attributes supplied with status information for those attributes that were invalid.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-030**	**Alarm Title**	**【Embedded service error】**
Description	An embedded service resulted in an error.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-031**	**Alarm Title**	**【Vendor specific error】**
Description	A vendor specific error has been encountered. The Additional Code Field of the Error Response defines the particular error encountered. Use of this General Error Code should only be performed when none of the Error Codes presented in this table or within an Object Class definition accurately reflect the error.		
Possible Cause	Use this error code only when the "CIP General Status Code" or "Object Class Definition" does not accurately reflect the error.		
Solution	This special error is described in the response packet with the code field of "Extra Status". Currently, you need to contact the Vendor to get the "extra status" code. Then follow the Vendor's manual for troubleshooting!		

Alarm ID	CIP-032	Alarm Title	【Invalid parameter】
Description	A parameter associated with the request was invalid. This code is used when a parameter does not meet the requirements of this specification and/or the requirements defined in an Application Object Specification.		
Possible Cause	When the parameter does not meet the "necessary requirements of the specification" and/or "the requirements defined in the object specification", this alarm is issued.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-033	Alarm Title	【Write-once value or medium already written】
Description	An attempt was made to write to a write-once medium (e.g. WORM drive, PROM) that has already been written, or to modify a value that cannot be changed once established.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-034	Alarm Title	【Invalid Reply Received】
Description	An invalid reply is received (e.g. reply service code does not match the request service code, or reply message is shorter than the minimum expected reply size). This status code can serve for other causes of invalid replies		
Possible Cause	1. The responsed Service code does not match the service code of the command. 2. The length of the response message is shorter than the minimum expected length. Note: This CIP alarm is also reserved for other reasons causing invalid response.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-035	Alarm Title	【buffer overflow】
Description	The message received is larger than the receiving buffer can handle. The entire message was discarded.		
Possible Cause	Please refer to the CIP manual specifications.		

Solution	Please refer to the CIP manual specifications.		
Alarm ID	**CIP-036**	**Alarm Title**	**【Message Format Error】**
Description	The format of the received message is not supported by the server.		
Possible Cause	Please refer to the CIP manual specifications		
Solution	Please refer to the CIP manual specifications		
Alarm ID	**CIP-037**	**Alarm Title**	**【Key Failure in path】**
Description	The Key Segment that was included as the first segment in the path does not match the destination module. The object specific status shall indicate which part of the key check failed.		
Possible Cause	The first segment in the path that contains the key does not match the target module.		
Solution	Referring to the particular state of the object, it will indicate which part of the key check failed.		
Alarm ID	**CIP-038**	**Alarm Title**	**【Path Size Invalid】**
Description	The size of the path which was sent with the Service Request is either not large enough to allow the Request to be routed to an object or too much routing data was included.		
Possible Cause	1. The path sent is not large, not enough to allow the request command to be sent to the specified object. 2. The path sent is too long, and contains too much mediation information on the path.		
Solution	Please refer to the CIP manual specifications		
Alarm ID	**CIP-039**	**Alarm Title**	**【Unexpected attribute in list】**
Description	An attempt was made to set an attribute that is not able to be set at this time.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-040	Alarm Title	【Invalid member ID】
Description	The Member ID specified in the request does not exist in the specified Class/ Instance/Attribute.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-041	Alarm Title	【Member not settable】
Description	A request to modify a non-modifiable member was received.		
Possible Cause	Try to set a member that cannot be modified.		
Solution	Do not set up members that cannot be modified.		

Alarm ID	CIP-042	Alarm Title	【Group 2 only server general failure】
Description	This error code may only be reported by DeviceNet Group 2 Only servers with 4K or less code space and only in place of Service not supported, Attribute not supported and Attribute not settable.		
Possible Cause	This wrong alarm is only issued by a server with DeviceNet Group 2 that has 4K or less of code space. And only replace the unsupported Service, the unsupported Attribute and Attribute are not set.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-043	Alarm Title	【Unknown Modbus Error】
Description	A CIP to Modbus translator received an unknown Modbus Exception Code.		
Possible Cause	The CIP to Modbus converter received an unknown Modbus error code.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	CIP-044	Alarm Title	【Attribute not gettable】
Description	A request to read a non-readable attribute was received.		

Possible Cause	Try to get a value from an Attribute that cannot be read.		
Solution	Do not get value on Attributes that cannot be read.		
Alarm ID	**CIP-045**	**Alarm Title**	**【Instance Not Deletable】**
Description	The requested object instance cannot be deleted.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

Alarm ID	**CIP-046**	**Alarm Title**	**【Service Not Supported for Specified Path 】**
Description	The object supports the service, but not for the designated application path (e.g. attribute). NOTE: Not to be used for any set service (use General Status Code 0x0E or 0x29 instead)		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.1 CIP-001 Connection failure

Alarm ID	**CIP-001**	**Alarm Title**	**【Connection failure】**
Description	A connection related service failed along the connection path.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.2 CIP-002 Resource unavailable

Alarm ID	**CIP-002**	**Alarm Title**	**【Resource unavailable】**
Description	Resources needed for the object to perform the requested service were unavailable.		

Possible Cause	Please refer to the CIP manual specifications.
Solution	Please refer to the CIP manual specifications.

17.3 CIP-003 Invalid parameter value

Alarm ID	**CIP-003**	**Alarm Title**	**【Invalid parameter value】**
Description	See Status Code 0x20, which is the preferred value to use for this condition.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.4 CIP-004 Path segment error

Alarm ID	**CIP-004**	**Alarm Title**	**【Path segment error】**
Description	The path segment identifier or the segment syntax was not understood by the processing node. Path processing shall stop when a path segment error is encountered.		
Possible Cause	The processed node does not understand the path paragraph identifier or paragraph syntax. When the path is wrong, the processing node will stop the path processing.		
Solution	Please refer to the CIP manual specifications.		

17.5 CIP-005 Path destination unknown

Alarm ID	**CIP-005**	**Alarm Title**	**【Path destination unknown】**
Description	The path is referencing an object class, instance or structure element that is not known or is not contained in the processing node. Path processing shall stop when a path destination unknown error is encountered.		
Possible Cause	This path uses an object Class, Instance or Structure item that is unknown or not included in the processing node. When the path target is unknown, the processing node stops the path processing.		

Solution	Please refer to the CIP manual specifications.

17.6 CIP-006 Partial transfer

Alarm ID	**CIP-006**	**Alarm Title**	**【Partial transfer】**
Description	Only part of the expected data was transferred.		
Possible Cause	There is not enough information required for the service.		
Solution	Please refer to the CIP manual specifications.		

17.7 CIP-007 Connection lost

Alarm ID	**CIP-007**	**Alarm Title**	**【Connection lost】**
Description	The messaging connection was lost.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.8 CIP-008 Service not supported

Alarm ID	**CIP-008**	**Alarm Title**	**【Service not supported】**
Description	The requested service was not implemented or was not defined for this Object Class/ Instance.		
Possible Cause	The Service code of the instruction is not implemented or undefined in the corresponding object Class/Instance.		
Solution	Please use the object Class/Instance to have a implemented or defined Service code.		

17.9 CIP-009 Invalid Attribute value

Alarm ID	**CIP-009**	**Alarm Title**	**【Invalid Attribute value】**

Description	Invalid Attribute data detected.
Possible Cause	The specified Attribute data is non-compliance.
Solution	According to the description of the Attribute, give the correct value.

17.10 CIP-010 Attribute list error

Alarm ID	**CIP-010**	**Alarm Title**	**【Attribute list error】**
Description	An attribute in the Get_Attribute_List or Set_Attribute_List response has a non-zero status.		
Possible Cause	When using the Get_Attribute_List (0x03) or Set_Attribute_List (0x04) service code, if an access to an Attribute data is unsuccessful, this alarm will be issued.		
Solution	According to the access specification of the object related to the Attribute list, confirm the following specifications: 1. Whether the specified Attribute can be obtained or set. 2. Whether the set Attribute value is correct.		

17.11 CIP-011 Already in requested mode/state

Alarm ID	**CIP-011**	**Alarm Title**	**【Already in requested mode/state】**
Description	The object is already in the mode/state being requested by the service.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.12 CIP-012 Object state conflict

Alarm ID	**CIP-012**	**Alarm Title**	**【Object state conflict】**
Description	The object cannot perform the requested service in its current mode/state.		
Possible Cause	Please refer to the CIP manual specifications.		

Solution	Please refer to the CIP manual specifications.

17.13 CIP-013 Object already exists

Alarm ID	**CIP-013**	**Alarm Title**	**【Object already exists】**
Description	The requested instance of object to be created already exists.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.14 CIP-014 Attribute not settable

Alarm ID	**CIP-014**	**Alarm Title**	**【Attribute not settable】**
Description	A request to modify a non-modifiable attribute was received.		
Possible Cause	Try to set an Attribute that cannot be modified.		
Solution	Do not set Attributes that cannot be modified.		

17.15 CIP-015 Privilege violation

Alarm ID	**CIP-015**	**Alarm Title**	**【Privilege violation】**
Description	A permission/privilege check failed.		
Possible Cause	The permission/rights check failed.		
Solution	Please refer to the CIP manual specifications.		

17.16 CIP-016 Device state conflict

Alarm ID	**CIP-016**	**Alarm Title**	**【Device state conflict】**
Description	The device’s current mode/state prohibits the execution of the requested service.		

Possible Cause	Please refer to the CIP manual specifications.
Solution	Please refer to the CIP manual specifications.

17.17 CIP-017 Reply data too large

Alarm ID	**CIP-017**	**Alarm Title**	**【Reply data too large】**
Description	The data to be transmitted in the response buffer is larger than the allocated response buffer.		
Possible Cause	The amount of data responded is greater than the size of the response buffer configuration.		
Solution	Please refer to the CIP manual specifications.		

17.18 CIP-018 Fragmentation of a primitive value

Alarm ID	**CIP-018**	**Alarm Title**	**【Fragmentation of a primitive value 】**
Description	The service specified an operation that is going to fragment a primitive data value, i.e. half a REAL data type.		
Possible Cause	This service operation splits the basic data values, for example: half REAL data type.		
Solution	The service cannot split the data values of the basic type.		

17.19 CIP-019 Not enough data

Alarm ID	**CIP-019**	**Alarm Title**	**【Not enough data】**
Description	The service did not supply enough data to perform the specified operation.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.20 CIP-020 Attribute not supported

Alarm ID	CIP-020	Alarm Title	【Attribute not supported】
Description	The attribute specified in the request is not supported.		
Possible Cause	The Attribute code of the command is unimplemented or undefined in the corresponding object Class/Instance.		
Solution	Please use the object Class/Instance to have an Attribute code with implementation or definition.		

17.21 CIP-021 Too much data

Alarm ID	CIP-021	Alarm Title	【Too much data】
Description	The service supplied more data than was expected.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.22 CIP-022 object does not exist

Alarm ID	CIP-022	Alarm Title	【object does not exist】
Description	The object specified does not exist in the device.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.23 CIP-023 Service fragmentation sequence not in progress

Alarm ID	CIP-023	Alarm Title	【Service fragmentation sequence not in progress】
Description	The fragmentation sequence for this service is not currently active for this data.		

Alarm ID	CIP-023	Alarm Title	【Service fragmentation sequence not in progress】
Possible Cause	The target node failed to receive the information tag correctly.		
Solution	Please refer to the CIP manual specifications.		

17.24 CIP-024 No stored attribute data

Alarm ID	CIP-024	Alarm Title	【No stored attribute data】
Description	The attribute data of this object was not saved prior to the requested service.		
Possible Cause	The Attribute data for this object was not stored until the service was requested.		
Solution	Please refer to the CIP manual specifications.		

17.25 CIP-025 Store operation failure

Alarm ID	CIP-025	Alarm Title	【Store operation failure】
Description	The attribute data of this object was not saved due to a failure during the attempt.		
Possible Cause	An error occurred in the process of storing the Attribute data to target node.		
Solution	Please refer to the CIP manual specifications.		

17.26 CIP-026 Routing failure, request packet too large

Alarm ID	CIP-026	Alarm Title	【Routing failure, request packet too large】
Description	The service request packet was too large for transmission on a network in the path to the destination. The routing device was forced to abort the service.		
Possible Cause	The command packet of the service is too large to be transmitted on the network path to the destination, and the routing device is forced to terminate the service.		
Solution	Please refer to the CIP manual specifications.		

17.27 CIP-027 Routing failure, response packet too large

Alarm ID	CIP-027	Alarm Title	【Routing failure, response packet too large】
Description	The service response packet was too large for transmission on a network in the path from the destination. The routing device was forced to abort the service.		
Possible Cause	The response packet of the service is too large to be transmitted from the target network path, and the routing device is forced to terminate the service.		
Solution	Please refer to the CIP manual specifications.		

17.28 CIP-028 Missing attribute list entry data

Alarm ID	CIP-028	Alarm Title	【Missing attribute list entry data】
Description	The service did not supply an attribute in a list of attributes that was needed by the service to perform the requested behavior.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.29 CIP-029 Invalid Attribute Value List

Alarm ID	CIP-029	Alarm Title	【Invalid Attribute Value List】
Description	The service is returning the list of attributes supplied with status information for those attributes that were invalid.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.30 CIP-030 Embedded service error

Alarm ID	CIP-030	Alarm Title	【Embedded service error】

Description	An embedded service resulted in an error.
Possible Cause	Please refer to the CIP manual specifications.
Solution	Please refer to the CIP manual specifications.

17.31 CIP-031 Vendor specific error

Alarm ID	**CIP-031**	**Alarm Title**	**【Vendor specific error】**
Description	A vendor specific error has been encountered. The Additional Code Field of the Error Response defines the particular error encountered. Use of this General Error Code should only be performed when none of the Error Codes presented in this table or within an Object Class definition accurately reflect the error.		
Possible Cause	Use this error code only when the "CIP General Status Code" or "Object Class Definition" does not accurately reflect the error.		
Solution	This special error is described in the response packet with the code field of "Extra Status". Currently, you need to contact the Vendor to get the "extra status" code. Then follow the Vendor's manual for troubleshooting!		

17.32 CIP-032 Invalid parameter

Alarm ID	**CIP-032**	**Alarm Title**	**【Invalid parameter】**
Description	A parameter associated with the request was invalid. This code is used when a parameter does not meet the requirements of this specification and/or the requirements defined in an Application Object Specification.		
Possible Cause	When the parameter does not meet the "necessary requirements of the specification" and/or "the requirements defined in the object specification", this alarm is issued.		
Solution	Please refer to the CIP manual specifications.		

17.33 CIP-033 Write-once value or medium already written

Alarm ID	**CIP-033**	**Alarm Title**	**【Write-once value or medium already written】**

Description	An attempt was made to write to a write-once medium (e.g. WORM drive, PROM) that has already been written, or to modify a value that cannot be changed once established.
Possible Cause	Please refer to the CIP manual specifications.
Solution	Please refer to the CIP manual specifications.

17.34 CIP-034 Invalid Reply Received

Alarm ID	**CIP-034**	**Alarm Title**	**【Invalid Reply Received】**
Description	An invalid reply is received (e.g. reply service code does not match the request service code, or reply message is shorter than the minimum expected reply size). This status code can serve for other causes of invalid replies		
Possible Cause	1. The responsed Service code does not match the service code of the command. 2. The length of the response message is shorter than the minimum expected length. Note: This CIP alarm is also reserved for other reasons causing invalid response.		
Solution	Please refer to the CIP manual specifications.		

17.35 CIP-035 buffer overflow

Alarm ID	**CIP-035**	**Alarm Title**	**【buffer overflow】**
Description	The message received is larger than the receiving buffer can handle. The entire message was discarded.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.36 CIP-036 Message Format Error

Alarm ID	**CIP-036**	**Alarm Title**	**【Message Format Error】**
Description	The format of the received message is not supported by the server.		

Possible Cause	Please refer to the CIP manual specifications
Solution	Please refer to the CIP manual specifications

17.37 CIP-037 Key Failure in path

Alarm ID	**CIP-037**	**Alarm Title**	**【Key Failure in path】**
Description	The Key Segment that was included as the first segment in the path does not match the destination module. The object specific status shall indicate which part of the key check failed.		
Possible Cause	The first segment in the path that contains the key does not match the target module.		
Solution	Referring to the particular state of the object, it will indicate which part of the key check failed.		

17.38 CIP-038 Path Size Invalid

Alarm ID	**CIP-038**	**Alarm Title**	**【Path Size Invalid】**
Description	The size of the path which was sent with the Service Request is either not large enough to allow the Request to be routed to an object or too much routing data was included.		
Possible Cause	1. The path sent is not large, not enough to allow the request command to be sent to the specified object. 2. The path sent is too long, and contains too much mediation information on the path.		
Solution	Please refer to the CIP manual specifications		

17.39 CIP-039 Unexpected attribute in list

Alarm ID	**CIP-039**	**Alarm Title**	**【Unexpected attribute in list】**
Description	An attempt was made to set an attribute that is not able to be set at this time.		
Possible Cause	Please refer to the CIP manual specifications.		

Solution	Please refer to the CIP manual specifications.

17.40 CIP-040 Invalid member ID

Alarm ID	**CIP-040**	**Alarm Title**	**【Invalid member ID】**
Description	The Member ID specified in the request does not exist in the specified Class/ Instance/Attribute.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.41 CIP-041 Member not settable

Alarm ID	**CIP-041**	**Alarm Title**	**【Member not settable】**
Description	A request to modify a non-modifiable member was received.		
Possible Cause	Try to set a member that cannot be modified.		
Solution	Do not set up members that cannot be modified.		

17.42 CIP-042 Group 2 only server general failure

Alarm ID	**CIP-042**	**Alarm Title**	**【Group 2 only server general failure】**
Description	This error code may only be reported by DeviceNet Group 2 Only servers with 4K or less code space and only in place of Service not supported, Attribute not supported and Attribute not settable.		
Possible Cause	This wrong alarm is only issued by a server with DeviceNet Group 2 that has 4K or less of code space. And only replace the unsupported Service, the unsupported Attribute and Attribute are not set.		
Solution	Please refer to the CIP manual specifications.		

17.43 CIP-043 Unknown Modbus Error

Alarm ID	CIP-043	Alarm Title	【Unknown Modbus Error】
Description	A CIP to Modbus translator received an unknown Modbus Exception Code.		
Possible Cause	The CIP to Modbus converter received an unknown Modbus error code.		
Solution	Please refer to the CIP manual specifications.		

17.44 CIP-044 Attribute not gettable

Alarm ID	CIP-044	Alarm Title	【Attribute not gettable】
Description	A request to read a non-readable attribute was received.		
Possible Cause	Try to get a value from an Attribute that cannot be read.		
Solution	Do not get value on Attributes that cannot be read.		

17.45 CIP-045 Instance Not Deletable

Alarm ID	CIP-045	Alarm Title	【Instance Not Deletable】
Description	The requested object instance cannot be deleted.		
Possible Cause	Please refer to the CIP manual specifications.		
Solution	Please refer to the CIP manual specifications.		

17.46 CIP-046 Service Not Supported for Specified Path

Alarm ID	CIP-046	Alarm Title	【Service Not Supported for Specified Path 】
Description	The object supports the service, but not for the designated application path (e.g. attribute). NOTE: Not to be used for any set service (use General Status Code 0x0E or 0x29 instead)		

Possible Cause	Please refer to the CIP manual specifications.
Solution	Please refer to the CIP manual specifications.

SYNTEC

18 ENIP Alarm - ENIP

Alarm ID	**ENIP-001**	**Alarm Title**	**【Failed to establish connection】**
Description	Failed to establish connection to the target EnIP device.		
Possible Cause	1. The hardware device mulfunction or the network cable is loosed. 2. Exceeds the maximum number of connections supported by the controller. 3. The controller network card IP, device IP or connected device changes. 4. The connection initialization failed.		
Solution	1. Check hardware devices and network routing materials. 2. Currently controller supports up to 32 sets of different device IPs' connection. Please confirm the total connection number. 3. Rescan and update the connected device information. 4. Please contact Vendor.		

Alarm ID	**ENIP-002**	**Alarm Title**	**【Command failed to send】**
Description	Failed to send an instruction to the target EnIP device.		
Possible Cause	The "instruction data length" of this command is greater than the default 256-bytes.		
Solution	Check if the “instruction data length” is correct. If it is necessary to set value greater than the default 256-bytes, please contact Vendor.		

Alarm ID	**ENIP-003**	**Alarm Title**	**【Instruction reply timeout】**
Description	The target EnIP device reply command timeout.		
Possible Cause	The device response time exceeds the controller timeout setting (default 40ms).		
Solution	Please contact Vendor. (the appropriate timeout settings will be opened for user to modify in the future)		

Alarm ID	**ENIP-004**	**Alarm Title**	**【Package information error】**
Description	Encountered an unexpected error when analyzing the EnIP package.		
Possible Cause	The device returns unexpected package information.		
Solution	Please contact the Vendor.		

Alarm ID	ENIP-005	Alarm Title	【Network connection interruption】
Description	The connection to the target EnIP device network is interrupted.		
Possible Cause	1. Hardware device malfunction or the network cable is loose. 2. The controller is abnormal and cannot send or read network packets.		
Solution	1. Check hardware devices and network cables. 2. Please contact the Vendor.		

Alarm ID	ENIP-006	Alarm Title	【Establish Implicit Connection Failure】
Description	Failed to establish an implicit connection to the target EnIP device.		
Possible Cause	1. Hardware equipment failure or loose network cable. 2. Exceeding the maximum number of controller connections of 32 groups. 3. The controller network card IP, device IP, or connected device changes. 4. Connection initialization failed. 5. The implicit connection parameter settings of the hardware device is wrong.		
Solution	1. Check hardware devices and network cables. 2. Currently, the controller supports a maximum of 32 groups of different device IPs for connection. 3. Broadcast again to update connected device information. 4. Please reconfirm whether the T2O/O2T instance ID and configuration instance ID parameters are set correctly. 5. Please reconfirm whether the T2O/O2T data length parameter is set correctly. 6. After trying the above solutions, please restart the controller and hardware devices. 7. Please contact OEM Syntec.		

Alarm ID	ENIP-007	Alarm Title	【Implicit Connection Exception】
Description	The implicit connection to the target EnIP device was interrupted.		
Possible Cause	Hardware equipment failure or loose network cable.		
Solution	1. Check hardware devices and network cables. 2. Broadcast again to update connected device information. 3. After trying the above solutions, please restart the controller and hardware devices. 4. Please contact OEM Syntec.		

Alarm ID	ENIP-008	Alarm Title	【Unsupported Request Packet Interval】
Description	The target EnIP device does not support the RPI value set by the user.		
Possible Cause	The set RPI value is not in the range of [10000, 3200000].		
Solution	Adjust the RPI value, and restart the controller and hardware devices.		

Alarm ID	ENIP-005	Alarm Title	【Invalid Start Registry Settings】
Description	The T2O/O2T start registry is not set properly.		
Possible Cause	1. The exchange packet register location intervals overlap. 2. The register location is set to the range occupied by the controller system. 3. The register location is already occupied by another device.		
Solution	Adjust the T2O/O2T data length and the T2O/O2T start registry parameters, and restart the controller and hardware devices.		

18.1 ENIP-001 Failed to establish connection

Alarm ID	ENIP-001	Alarm Title	【Failed to establish connection】
Description	Failed to establish connection to the target EnIP device.		
Possible Cause	1. The hardware device mulfunction or the network cable is loosed. 2. Exceeds the maximum number of connections supported by the controller. 3. The controller network card IP, device IP or connected device changes. 4. The connection initialization failed.		
Solution	1. Check hardware devices and network routing materials. 2. Currently controller supports up to 32 sets of different device IPs' connection. Please confirm the total connection number. 3. Rescan and update the connected device information. 4. Please contact Vendor.		

18.2 ENIP-002 Command failed to send

Alarm ID	ENIP-002	Alarm Title	【Command failed to send】

Description	Failed to send an instruction to the target EnIP device.
Possible Cause	The "instruction data length" of this command is greater than the default 256-bytes.
Solution	Check if the “instruction data length” is correct. If it is necessary to set value greater than the default 256-bytes, please contact Vendor.

18.3 ENIP-003 Instruction reply timeout

Alarm ID	**ENIP-003**	**Alarm Title**	**【Instruction reply timeout】**
Description	The target EnIP device reply command timeout.		
Possible Cause	The device response time exceeds the controller timeout setting (default 40ms).		
Solution	Please contact Vendor. (the appropriate timeout settings will be opened for user to modify in the future)		

18.4 ENIP-004 Package information error

Alarm ID	**ENIP-004**	**Alarm Title**	**【Package information error】**
Description	Encountered an unexpected error when analyzing the EnIP package.		
Possible Cause	The device returns unexpected package information.		
Solution	Please contact the Vendor.		

18.5 ENIP-005 Network connection interruption

Alarm ID	**ENIP-005**	**Alarm Title**	**【Network connection interruption】**
Description	The connection to the target EnIP device network is interrupted.		
Possible Cause	1. Hardware device malfunction or the network cable is loose. 2. The controller is abnormal and cannot send or read network packets.		
Solution	1. Check hardware devices and network cables. 2. Please contact the Vendor.		

18.6 ENIP-006 Establish Implicit Connection Failure

Alarm ID	ENIP-006	Alarm Title	【Establish Implicit Connection Failure】
Description	Failed to establish an implicit connection to the target EnIP device.		
Possible Cause	1. Hardware equipment failure or loose network cable. 2. Exceeding the maximum number of controller connections of 32 groups. 3. The controller network card IP, device IP, or connected device changes. 4. Connection initialization failed. 5. The implicit connection parameter settings of the hardware device is wrong.		
Solution	1. Check hardware devices and network cables. 2. Currently, the controller supports a maximum of 32 groups of different device IPs for connection. 3. Broadcast again to update connected device information. 4. Please reconfirm whether the T2O/O2T instance ID and configuration instance ID parameters are set correctly. 5. Please reconfirm whether the T2O/O2T data length parameter is set correctly. 6. After trying the above solutions, please restart the controller and hardware devices. 7. Please contact OEM Syntec.		

18.7 ENIP-007 Implicit Connection Exception

Alarm ID	ENIP-007	Alarm Title	【Implicit Connection Exception】
Description	The implicit connection to the target EnIP device was interrupted.		
Possible Cause	Hardware equipment failure or loose network cable.		
Solution	1. Check hardware devices and network cables. 2. Broadcast again to update connected device information. 3. After trying the above solutions, please restart the controller and hardware devices. 4. Please contact OEM Syntec.		

18.8 ENIP-008 Unsupported Request Packet Interval

Alarm ID	ENIP-008	Alarm Title	【Unsupported Request Packet Interval】
Description	The target EnIP device does not support the RPI value set by the user.		

Possible Cause	The set RPI value is not in the range of [10000, 3200000].
Solution	Adjust the RPI value, and restart the controller and hardware devices.

18.9 ENIP-009 Invalid Start Registry Settings

Alarm ID	**ENIP-005**	**Alarm Title**	**【Invalid Start Registry Settings】**
Description	The T2O/O2T start registry is not set properly.		
Possible Cause	1. The exchange packet register location intervals overlap. 2. The register location is set to the range occupied by the controller system. 3. The register location is already occupied by another device.		
Solution	Adjust the T2O/O2T data length and the T2O/O2T start registry parameters, and restart the controller and hardware devices.		

SYNTEC

19 Extended Software Stroke Limit Alarm - EXTSTROKELIMIT

Alarm ID	EXTSTROKELIMIT-001	Alarm Title	【Group N Extended Software Stroke Limit Prohibited Entry】
Description	The moving position of the machine enters the extended software limit protection range.		
Possible Cause	1. Set the wrong protection range, causing the current position of the machine to enter the protection range. 2. The moving direction of the machine is about to enter the extended software limit protection area.		
Solution	1. Check the corresponding extended software limit protection area parameters, and reset the alarm after resetting the protection range correctly. 2. In the auto mode, check whether the machining program will eater the protection zone. After modifying the machining program/protection zone setting value, Reset to clear the alarm. 3. In the manual mode, check whether the protection range setting is correct. If yes, then move the axis in reverse direction to release alarm; if not, modify the protection range parameter or turn off the expansion software stroke limit, Reset to clear the alarm.		
Alarm ID	**EXTSTROKELIMIT-002**	**Alarm Title**	**【Group N Extended Software Stroke Limit Protection Range Error】**
Description	Positive and negative limit parameter setting error.		
Possible Cause	Positive limit value is less than negative limit value.		
Solution	Set the correct extended software limit protection parameters.		

19.1 EXTSTROKELIMIT-001 Group N Extended Software Stroke Limit Prohibited Entry

Alarm ID	EXTSTROKELIMIT-001	Alarm Title	【Group N Extended Software Stroke Limit Prohibited Entry】
Description	The moving position of the machine enters the extended software limit protection range.		
Possible Cause	1. Set the wrong protection range, causing the current position of the machine to enter the protection range. 2. The moving direction of the machine is about to enter the extended software limit protection area.		

Solution	1. Check the corresponding extended software limit protection area parameters, and reset the alarm after resetting the protection range correctly. 2. In the auto mode, check whether the machining program will eater the protection zone. After modifying the machining program/protection zone setting value, Reset to clear the alarm. 3. In the manual mode, check whether the protection range setting is correct. If yes, then move the axis in reverse direction to release alarm; if not, modify the protection range parameter or turn off the expansion software stroke limit, Reset to clear the alarm.

19.2 EXTSTROKELIMIT-002 Group N Extended Software Stroke Limit Protection Range Error

Alarm ID	**EXTSTROKELIMIT-002**	**Alarm Title**	**【Group N Extended Software Stroke Limit Protection Range Error】**
Description	Positive and negative limit parameter setting error.		
Possible Cause	Positive limit value is less than negative limit value.		
Solution	Set the correct extended software limit protection parameters.		

SYNTEC

20 Setpoint Axis Alarm - SETPOINTAXIS

Alarm ID	**SETPOINTAXIS-001**	**Alarm Title**	Setpoint axis station ID is repeated with other devices
Description	Setpoint axis station number is repeated with other station number, and the system enters Not Ready state.		
Possible Cause	Setpoint axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	After setting the station number correctly, reboot the controller and the drive.		

Alarm ID	**SETPOINTAXIS-002**	**Alarm Title**	Start R Register setting is out of range, please reassign R register
Description	The start register is set incorrectly, which is in the system protected area or bit access is not supported.		
Possible Cause	1. The command status base address R(m) is in the CNC system interface area. 2. The command status base address R(m) does not support bit access.		
Solution	1. Set R(m) between [50,80] or [256, 511]. 2. After setting correctly, reboot the controller and the drive.		

Alarm ID	**SETPOINTAXIS-003**	**Alarm Title**	Acceleration time or speed setting error
Description	Acceleration time or speed setting value is out of range.		

SYNTEC

Possible Cause	1. The acceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the drive. (The acceleration is based on the pulse wave per square sec. of gear teeth number on screw side, gear teeth number on motor side, pitch amount, encoder resolution. When using the "Yakawa" drive, it must fall within the range of 1~655,350,000.) 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration time setting exceeds the range of drive parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" drive, the parameter setting range is 0~60,000)
Solution	1. Reset to clear the alarm. 2. Properly enlarge acceleration time to a reasonable value to avoid issuing this alarm again. 3. Properly reduce the target speed and maximum moving speed to a reasonable value to avoid issuing this alarm again. 4. When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot the controller and the drive. 5. When using the Syntec 2nd generation M2 driver, set the acceleration time between 0 and 60,000.

Alarm ID	**SETPOINTAXIS-004**	**Alarm Title**	Check window setting error
Description	Check window setting error.		
Possible Cause	The position check window setting is out of range. (The number of pulse converted by the position check window according to gear teeth number on screw side, gear teeth number on motor side, pitch amount, encoder resolution, must fall within the range of 0~1,073,741,824)		
Solution	Set the position check window within a reasonable range.		

Alarm ID	**SETPOINTAXIS-005**	**Alarm Title**	Incremental encoder is not supported
Description	Setpoint axis does not support incremental encoder, only supports absolute encoder.		
Possible Cause	1. Incremental encoder is used. 2. Absolute encoder is used, but drive parameter is set as incremental type.		
Solution	1. Use absolute encoder. 2. Set the drive parameter as absolute type. 3. After solving the above problems, reboot the controller and the drive.		

Alarm ID	SETPOINTAXIS-006	Alarm Title	Home position is not set, please set home position first
Description	Home position is not set, please set home position first.		
Possible Cause	Home position is not set.		
Solution	Move axis to home position, then trigger R(m).2 to set home position.		

Alarm ID	SETPOINTAXIS-007	Alarm Title	Absolute encoder related parameter changed,
Description	The user has modified the absolute encoder related parameter. please reboot reboot the controller and t		
Possible Cause	1. Modified any of the following parameters: gear teeth number on screw side, gear teeth number on mo 2. Modified Pr17: Control precision.		
Solution	Reboot the controller and the drive.		

Alarm ID	SETPOINTAXIS-008	Alarm Title	Drive station ID do
Description	Drive corresponding to the station ID does not exist on the communication network.		
Possible Cause	1. There is no drive corresponding to the station ID on the communication network. 2. Setpoint axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: a. System axis b. Serial PLC axis c. ROT axis d. ATC axis e. Setpoint axis		
Solution	After setting the correct station ID, reboot the controller and the drive.		

Alarm ID	SETPOINTAXIS-009	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		

Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.

Alarm ID	**SETPOINTAXIS-010**	**Alarm Title**	Drive parameter writing failed
Description	Setpoint axis axis failed to write drive parameter.		
Possible Cause	The communication between the controller and the drive is unstable, causing the drive parameter writing timeout.		
Solution	After solving the cause of communication error, reboot the controller and drive.		

Alarm ID	**SETPOINTAXIS-011**	**Alarm Title**	Drive power is abnormal
Description	The controller detected the drive power status error.		
Possible Cause	1. The drive power-amp back power supply is unstable. 2. Drive failure.		
Solution	1. Check if the drive power supply status is normal. 2. Check if the drive power cable is loose or damaged. 3. Replace the drive. 4. After solving the above problems, reboot the controller and the drive.		

Alarm ID	**SETPOINTAXIS-012**	**Alarm Title**	Setpoint axis command exceeds limit
Description	Setpoint axis command exceeds limit.		

Possible Cause	The command calculated by the current position and the target position is greater than the limit pulse: 2,147,483,647. (This alarm is issued to avoid when setting target position too far, the user misjudges the clockwise/counterclockwise failure and the inability to move to the target position.) [Method of calculating the limit of the command] **Limit of the command(um) = limit pulse x (number of gear on motor side x screw width) / (number of gear on screw side x position sensor resolution x 4)** For example: Number of gear on the motor side = 1, number of gear on the screw side = 1, position sensor resolution = 262,144 (pulse/rev), screw width = 5,000 (um/rev) The limit of command is 2147483647 x (1 x 5000) / (1 x 262144 x 4) = 10239999 (um) =10239.999 (mm). In other words, if the calculated command between the current position and the target position exceeds 10239.999 mm, this alarm will be issued; If it does not exceed 10239.999 mm, it is a reasonable displacement .
Solution	1. Reset to clear the alarm. 2. Change the target position to avoid exceeding the the limit of displacement. 3. Set drive electronic gear numerator, and reduce the encoder resolution setting of controller. i.e. According to the above setting, set the drive electronic gear numerator as 8 and the encoder resolution as 32,768, the limit of posing displacement will become 81919.999mm. 4. Adjust the gear number on motor side / screw side. 5. Adjust the screw width. 6. Once the above adjustments are properly made, this alarm should not be issued again.

Alarm ID	**SETPOINTAXIS-013**	**Alarm Title**	The firmware update of Syntec encoder is finished, please reboot the drive
Description	The firmware update of Syntec encoder is finished, please reboot the drive		
Possible Cause	Performed encoder firmware upgrade.		
Solution	Reboot the controller and drive.		

Alarm ID	**SETPOINTAXIS-014**	**Alarm Title**	Syntec encoder information reading timeout
Description	Syntec encoder information reading timeout.		
Possible Cause	1. The communication between the controller and the drive is abnormal. 2. The drive status is abnormal.		

Solution	1. Check if the communication cable is loose. 2. Reboot the controller and drive.

Alarm ID	**SETPOINTAXIS-015**	**Alarm Title**	Axial feedback is abnormal
Description	If the difference between the axial feedback value exceeds 4 times of the maximum speed set by the controller, the number of abnormal times will be accumulated by 1 each time. This alarm will be issued when two consecutive abnormalities occur. The maximum speed is the value of parameter: Maximum moving speed(mm/min)		
Possible Cause	1. Communication between controller and drive is abnormal. 2. Parameter: Maximum moving speed settings is inappropriate.		
Solution	1. Check if the communication cable is loose. 2. Adjust parameter setting: Maximum moving speed.		

Alarm ID	**SETPOINTAXIS-016**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

Alarm ID	**SETPOINTAXIS-017**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		
Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.		

Alarm ID	SETPOINTAXIS-018	Alarm title	The communication packet is disturbed by noise
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

Alarm ID	SETPOINTAXIS-019	Alarm title	Driver software misses communication packet
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

Alarm ID	SETPOINTAXIS-020	Alarm Title	Exceeding the positive software stroke limit
Description	The axial mechanical coordinate exceeds the positive software stroke limit.		
Possible Cause	The axial movement exceeds the setting value.		
Solution	1. Reset to clear the alarm. 2. Move the axial in negative direction to be out of the software stroke protection range, and can remove alarm.		

Alarm ID	SETPOINTAXIS-021	Alarm Title	Exceeding the negative software stroke limit
Description	The axial mechanical coordinate exceeds the negative software stroke limit.		
Possible Cause	The axial movement exceeds the setting value.		

Solution	1. Reset to clear the alarm. 2. Move the axial in positive direction to be out of the software stroke protection range, and can remove alarm.

20.1 SETPOINTAXIS-001 Setpoint axis station ID is repeated with other devices

Alarm ID	**SETPOINTAXIS-001**	**Alarm Title**	Setpoint axis station ID is repeated with other devices
Description	Setpoint axis station number is repeated with other station number, and the system enters Not Ready state.		
Possible Cause	Setpoint axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: 1. System axis 2. Serial PLC axis 3. ROT axis 4. ATC axis 5. Setpoint axis		
Solution	After setting the station number correctly, reboot the controller and the drive.		

20.2 SETPOINTAXIS-002 Start R Register setting is out of range, please reassign R register

Alarm ID	**SETPOINTAXIS-002**	**Alarm Title**	Start R Register setting is out of range, please reassign R register
Description	The start register is set incorrectly, which is in the system protected area or bit access is not supported.		
Possible Cause	1. The command status base address R(m) is in the CNC system interface area. 2. The command status base address R(m) does not support bit access.		
Solution	1. Set R(m) between [50,80] or [256, 511]. 2. After setting correctly, reboot the controller and the drive.		

20.3 SETPOINTAXIS-003 Acceleration time or speed setting error

Alarm ID	**SETPOINTAXIS-003**	**Alarm Title**	Acceleration time or speed setting error
Description	Acceleration time or speed setting value is out of range.		
Possible Cause	1. The acceleration time setting is too small, causing the corresponding acceleration exceed the allowed set value range of the drive. (The acceleration is based on the pulse wave per square sec. of gear teeth number on screw side, gear teeth number on motor side, pitch amount, encoder resolution. When using the "Yakawa" drive, it must fall within the range of 1~655,350,000.) 2. The position sensor has a high resolution, causing the corresponding acceleration exceed the allowed set value range of the driver. 3. The target speed and maximum moving speed are set too large, causing the corresponding acceleration exceed the allowed set value range of the driver. 4. Acceleration time setting exceeds the range of drive parameter values. (If the maximum acceleration is set for the Pn306 of the "SyntecII M2" drive, the parameter setting range is 0~60,000)		
Solution	1. Reset to clear the alarm. 2. Properly enlarge acceleration time to a reasonable value to avoid issuing this alarm again. 3. Properly reduce the target speed and maximum moving speed to a reasonable value to avoid issuing this alarm again. 4. When using Yaskawa Sigma M2 series, the problems can be eliminated by initializing the parameters or setting Pn833 to 1 and reboot the controller and the drive. 5. When using the Syntec 2nd generation M2 driver, set the acceleration time between 0 and 60,000.		

20.4 SETPOINTAXIS-004 Check window setting error

Alarm ID	**SETPOINTAXIS-004**	**Alarm Title**	Check window setting error
Description	Check window setting error.		
Possible Cause	The position check window setting is out of range. (The number of pulse converted by the position check window according to gear teeth number on screw side, gear teeth number on motor side, pitch amount, encoder resolution, must fall within the range of 0~1,073,741,824)		
Solution	Set the position check window within a reasonable range.		

20.5 SETPOINTAXIS-005 Incremental encoder is not supported

Alarm ID	**SETPOINTAXIS-005**	**Alarm Title**	Incremental encoder is not supported
Description	Setpoint axis does not support incremental encoder, only supports absolute encoder.		
Possible Cause	1. Incremental encoder is used. 2. Absolute encoder is used, but drive parameter is set as incremental type.		
Solution	1. Use absolute encoder. 2. Set the drive parameter as absolute type. 3. After solving the above problems, reboot the controller and the drive.		

20.6 SETPOINTAXIS-006 Home position is not set, please set home position first

Alarm ID	**SETPOINTAXIS-006**	**Alarm Title**	Home position is not set, please set home position first
Description	Home position is not set, please set home position first.		
Possible Cause	Home position is not set.		
Solution	Move axis to home position, then trigger R(m).2 to set home position.		

20.7 SETPOINTAXIS-007 Absolute encoder related parameter changed, please reboot the CNC controller and reset home position

Alarm ID	**SETPOINTAXIS-007**	**Alarm Title**	Absolute encoder related parameter changed,
Description	The user has modified the absolute encoder related parameter. please reboot reboot the controller and t		
Possible Cause	1. Modified any of the following parameters: gear teeth number on screw side, gear teeth number on mo 2. Modified Pr17: Control precision.		
Solution	Reboot the controller and the drive.		

20.8 SETPOINTAXIS-008 Drive station ID does not exist

Alarm ID	SETPOINTAXIS-008	Alarm Title	Drive station ID do
Description	Drive corresponding to the station ID does not exist on the communication network.		
Possible Cause	1. There is no drive corresponding to the station ID on the communication network. 2. Setpoint axis station number setting is repeated with other axis station number. Please check if the station number setting in the following axis is repeated: a. System axis b. Serial PLC axis c. ROT axis d. ATC axis e. Setpoint axis		
Solution	After setting the correct station ID, reboot the controller and the drive.		

20.9 SETPOINTAXIS-009 Drive communication Error

Alarm ID	SETPOINTAXIS-009	Alarm title	Drive communication error
Description	At least 2 errors listed below occur for over 10ms totally in communication. 1. Drive response packet not received. 2. Drive response packet CRC error. 3. Drive response packet watchdog error. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose. 2. The quality of the communication cable is poor, or there is noise interference. 3. Drive response packet watchdog error.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Check whether the machine is properly grounded, and add a terminal resistor to the communication port of the end drive. 3. Contact Syntec OEM.		

20.10 SETPOINTAXIS-010 Drive parameter writing failed

Alarm ID	SETPOINTAXIS-010	Alarm Title	Drive parameter writing failed
Description	Setpoint axis axis failed to write drive parameter.		

Possible Cause	The communication between the controller and the drive is unstable, causing the drive parameter writing timeout.
Solution	After solving the cause of communication error, reboot the controller and drive.

20.11 SETPOINTAXIS-011 Drive power is abnormal

Alarm ID	**SETPOINTAXIS-011**	**Alarm Title**	Drive power is abnormal
Description	The controller detected the drive power status error.		
Possible Cause	1. The drive power-amp back power supply is unstable. 2. Drive failure.		
Solution	1. Check if the drive power supply status is normal. 2. Check if the drive power cable is loose or damaged. 3. Replace the drive. 4. After solving the above problems, reboot the controller and the drive.		

20.12 SETPOINTAXIS-012 Setpoint axis command exceeds limit

Alarm ID	**SETPOINTAXIS-012**	**Alarm Title**	Setpoint axis command exceeds limit
Description	Setpoint axis command exceeds limit.		
Possible Cause	The command calculated by the current position and the target position is greater than the limit pulse: 2,147,483,647. (This alarm is issued to avoid when setting target position too far, the user misjudges the clockwise/counterclockwise failure and the inability to move to the target position.) [Method of calculating the limit of the command] **Limit of the command(um) = limit pulse x (number of gear on motor side x screw width) / (number of gear on screw side x position sensor resolution x 4)** For example: Number of gear on the motor side = 1, number of gear on the screw side = 1, position sensor resolution = 262,144 (pulse/rev), screw width = 5,000 (um/rev) The limit of command is 2147483647 x (1 x 5000) / (1 x 262144 x 4) = 10239999 (um) =10239.999 (mm). In other words, if the calculated command between the current position and the target position exceeds 10239.999 mm, this alarm will be issued; If it does not exceed 10239.999 mm, it is a reasonable displacement .		

Solution	1. Reset to clear the alarm. 2. Change the target position to avoid exceeding the the limit of displacement. 3. Set drive electronic gear numerator, and reduce the encoder resolution setting of controller. i.e. According to the above setting, set the drive electronic gear numerator as 8 and the encoder resolution as 32,768, the limit of posing displacement will become 81919.999mm. 4. Adjust the gear number on motor side / screw side. 5. Adjust the screw width. 6. Once the above adjustments are properly made, this alarm should not be issued again.

20.13 SETPOINTAXIS-013 The firmware update of Syntec encoder is finished, please reboot the drive

Alarm ID	**SETPOINTAXIS-013**	**Alarm Title**	The firmware update of Syntec encoder is finished, please reboot the drive
Description	The firmware update of Syntec encoder is finished, please reboot the drive		
Possible Cause	Performed encoder firmware upgrade.		
Solution	Reboot the controller and drive.		

20.14 SETPOINTAXIS-014 Syntec encoder information reading timeout

Alarm ID	**SETPOINTAXIS-014**	**Alarm Title**	Syntec encoder information reading timeout
Description	Syntec encoder information reading timeout.		
Possible Cause	1. The communication between the controller and the drive is abnormal. 2. The drive status is abnormal.		
Solution	1. Check if the communication cable is loose. 2. Reboot the controller and drive.		

20.15 SETPOINTAXIS-015 Axial feedback is abnormal

Alarm ID	**SETPOINTAXIS-015**	**Alarm Title**	Axial feedback is abnormal

Description	If the difference between the axial feedback value exceeds 4 times of the maximum speed set by the controller, the number of abnormal times will be accumulated by 1 each time. This alarm will be issued when two consecutive abnormalities occur. The maximum speed is the value of parameter: Maximum moving speed(mm/min)
Possible Cause	1. Communication between controller and drive is abnormal. 2. Parameter: Maximum moving speed settings is inappropriate.
Solution	1. Check if the communication cable is loose. 2. Adjust parameter setting: Maximum moving speed.

20.16 SETPOINTAXIS-016 Poor contact of communication wire

Alarm ID	**SETPOINTAXIS-016**	**Alarm title**	**Poor contact of communication wire**
Description	Drive response packet is not received for over 10ms in drive communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The communication cable between the controller and the driver is loose.		
Solution	1. Check the wiring of the communication cable between the controller and the drive. 2. Contact Syntec OEM.		

20.17 SETPOINTAXIS-017 The hardware doesn't receive communication packet

Alarm ID	**SETPOINTAXIS-017**	**Alarm title**	**The hardware doesn't receive communication packet**
Description	Drive response packet is not received for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The drive station number does not correspond to the controller parameters correctly. 2. The communication cable between the controller and the drive is loose.		

Solution	1. Check whether the drive parameter DIP switch setting corresponds to the controller parameter (Pr21 ~) correctly. 2. Check the wiring of the communication cable between the controller and the drive. 3. Contact Syntec OEM.

20.18 SETPOINTAXIS-018 The communication packet is disturbed by noise

Alarm ID	**SETPOINTAXIS-018**	**Alarm title**	**The communication packet is disturbed by noise**
Description	Drive response packet is disturbed by noise to generate CRC error for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	1. The quality of the communication cable is poor, or there is noise interference.		
Solution	1. Check whether the machine is properly grounded. 2. Contact Syntec OEM.		

20.19 SETPOINTAXIS-019 Driver software misses communication packet

Alarm ID	**SETPOINTAXIS-019**	**Alarm title**	**Driver software misses communication packet**
Description	Drive response packet watchdog is wrong for over 10ms in communication. Example: According to the communication time setting(Pr3203), 6 consecutive errors for 2ms; 4 consecutive errors for 3ms.		
Possible Reason	Drive response packet watchdog is wrong		
Solution	Contact Syntec OEM.		

20.20 SETPOINTAXIS-020 Exceeding the positive software stroke limit

Alarm ID	**SETPOINTAXIS-020**	**Alarm Title**	**Exceeding the positive software stroke limit**

Description	The axial mechanical coordinate exceeds the positive software stroke limit.
Possible Cause	The axial movement exceeds the setting value.
Solution	1. Reset to clear the alarm. 2. Move the axial in negative direction to be out of the software stroke protection range, and can remove alarm.

20.21 SETPOINTAXIS-021 Exceeding the negative software stroke limit

Alarm ID	**SETPOINTAXIS-021**	**Alarm Title**	**Exceeding the negative software stroke limit**
Description	The axial mechanical coordinate exceeds the negative software stroke limit.		
Possible Cause	The axial movement exceeds the setting value.		
Solution	1. Reset to clear the alarm. 2. Move the axial in positive direction to be out of the software stroke protection range, and can remove alarm.		

SYNTEC

21 Routine Script-RS

Alarm ID	RS–001	Alarm title	【File too big】
Description	Script file size is too big. The file size limit id 64KB.		
Reason	Script file size is too big.		
Solution	Modify script file, make sure the file size is smaller than 64KB.		

Alarm ID	RS-002	Alarm title	【Too many script file】
Description	The count of script is too many. System will stop execute part of script. The maximum file count is 8.		
Reason	The number of script is too many.		
Solution	Please remove unused script, or merge the script has similar function.		

Alarm ID	RS–003	Alarm title	Script loading failure
Description	The script has syntax error, cause system can not load script.		
Reason	Programming error.		
Solution	Please check the script.		

Alarm ID	RS–004	Alarm title	Unsupported syntax
Description	Use the unsupported syntax in script.		
Reason	1. Programming error. 2. Use unsupported function.		
Solution	Please check the script, make sure the syntax and function is supported.		

Alarm ID	RS-005	Alarm title	Illegal variable access
Description	Attempted to access a variable that does not exist.		

Reason	1. Programming error. 2. Illegal access to # or @ variable.
Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable.

Alarm ID	**RS-006**	**Alarm title**	**Invalid array index type**
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

Alarm ID	**RS-007**	**Alarm title**	**Invalid argument type**
Description	Th argument type of function or operator is invalid. i.e. @1=#1 MOD #2, if type of #1 or #2 is float, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the script, make sure that argument type is right.		

Alarm ID	**RS-008**	**Alarm title**	**Operation domain error**
Description	The argument of function has wrong domain. i.e: SQRT(#5), if #5 < 0, system will issue this alarm.		
Reason	Programming error		
Solution	Please check the script, make sure the domain of function argument is right.		

Alarm ID	**RS-009**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of script is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		

Reason	Programming error.
Solution	Check the script to ensure that the denominator is not equal to 0.

21.1 RS-001 File too big

Alarm ID	**RS–001**	**Alarm title**	**【File too big】**
Description	Script file size is too big. The file size limit id 64KB.		
Reason	Script file size is too big.		
Solution	Modify script file, make sure the file size is smaller than 64KB.		

21.2 RS-002 Too many script file

Alarm ID	**RS-002**	**Alarm title**	**【Too many script file】**
Description	The count of script is too many. System will stop execute part of script. The maximum file count is 8.		
Reason	The number of script is too many.		
Solution	Please remove unused script, or merge the script has similar function.		

21.3 RS-003 Script loading failure

Alarm ID	**RS–003**	**Alarm title**	**Script loading failure**
Description	The script has syntax error, cause system can not load script.		
Reason	Programming error.		
Solution	Please check the script.		

21.4 RS-004 Unsupported syntax

Alarm ID	RS–004	Alarm title	Unsupported syntax
Description	Use the unsupported syntax in script.		
Reason	1. Programming error. 2. Use unsupported function.		
Solution	Please check the script, make sure the syntax and function is supported.		

21.5 RS-005 Illegal variable access

Alarm ID	RS-005	Alarm title	Illegal variable access
Description	Attempted to access a variable that does not exist.		
Reason	1. Programming error. 2. Illegal access to # or @ variable.		
Solution	1. Make sure the accessing variable exists. 2. Make sure the variable to read is readable. 3. Make sure the variable to write is writable.		

21.6 RS-006 Invalid array index type

Alarm ID	RS-006	Alarm title	Invalid array index type
Description	Indirect assigned variable number is empty. i.e. @[#1], if #1 is empty, this alarm will be issued.		
Reason	Programming error.		
Solution	Please check the NC program, make sure that indirect assigned variable's number is not empty.		

21.7 RS-007 Invalid argument type

Alarm ID	RS-007	Alarm title	Invalid argument type

Description	Th argument type of function or operator is invalid. i.e. @1=#1 MOD #2, if type of #1 or #2 is float, this alarm will be issued.
Reason	Programming error.
Solution	Please check the script, make sure that argument type is right.

21.8 RS-008 Domain error

Alarm ID	**RS-008**	**Alarm title**	**Operation domain error**
Description	The argument of function has wrong domain. i.e: SQRT(#5), if #5 < 0, system will issue this alarm.		
Reason	Programming error		
Solution	Please check the script, make sure the domain of function argument is right.		

21.9 RS-009 Devide by zero error

Alarm ID	**RS-009**	**Alarm title**	**Devide by zero error**
Description	If denominator in division of script is equal to 0 i.e. #1:=(#2/ #3); if #3 equals to zero, system will issue this alarm.		
Reason	Programming error.		
Solution	Check the script to ensure that the denominator is not equal to 0.		

SYNTEC

22 Alarm Class Table

Mandarin Version 中文版：警報類別對照表

ID	Categories	ID	Categories	ID	Categories	ID	Categories	ID	Categories
1	Mechanical Logic Control Alarm - MLC	15	ROT Turret Alarm - ROT	101	DELTA Servo Alarm	202	DELTA ECAT Servo Alarm	212	INVENTOR ECAT Servo Alarm
2	Program Execute Error Alarm - COR	16	Laser Marking Alarm - LASERCTRL	102	YASKAWA M2 Servo Alarm	203	DELTA ECAT Servo Warning	213	INVENTOR ECAT Servo Warning
3	Axial Motion Alarm - MOT	17	Device Alarm - STATION	103	SYNTEC M2 Servo Alarm	204	PANASONIC ECAT Servo Alarm	214	INOVANCE ECAT Servo Alarm
4	Spindle Alarm - SPD	18	Extended Software Stroke Limit Alarm - EXTSTROKELIMIT	104	YASKAWA M3 Servo Alarm	205	PANASONIC ECAT Servo Warning	215	INOVANCE ECAT Servo Warning
5	Syntax Compiler Alarm - COM	19	ATC Alarm - ATC	105	SYNTEC M3 Servo Alarm	206	PANASONIC RTEX Servo Alarm	216	SYNTEC STEPPER Servo Alarm
6	Operation Alarm - OP	20	Setpoint Axis Alarm - SETPOINTAXIS	106	YASKAWA M2 Servo Warning	207	PANASONIC RTEX Servo Warning	217	SYNTEC STEPPER Servo Warning
7	Mechanical Logic Control Hint - MLC	21	Routine Script - RS	107	YASKAWA M3 Servo Warning	208	Servotronix ECAT Servo Alarm	218	SYNTEC ECAT Servo Alarm
9	MACRO Alarm - MAR	31	CIP Alarm - CIP	108	SYNTEC M2 Servo Warning	209	Servotronix ECAT Servo Warning	219	SYNTEC ECAT Servo Warning
10	Message Alarm - MSG	32	ENIP Alarm - ENIP	109	SYNTEC M3 Servo Warning	210	Mitsubishi ECAT Servo Alarm		

ID	Categories	ID	Categories	ID	Categories	ID	Categories	ID	Categories
11	Servo Alarm - SERVO	51	Robot Alarm - RBT	110	SYNTEC M2_V2 Servo Alarm	211	Mitsubishi ECAT Servo Warning		
12	Serial PLC Axis Alarm – SERIALPLCAXIS	52	Robot Warning - RBTWRN	111	SYNTEC M2_V2 Servo Warning				
13	SRI Alarm - SRI	53	Robot Critical - RBTCRI						
14	Background Execute Alarm - BGND								

Note: The alarm correlated information will be shown in R514 ~ R517 in PLC interface. Please refer to PLC Interface.

SYNTEC