Multicam™ Series

Toolpath Software
and
Machine Operators Guide

Multicam Systems
CNC Machinery Specialists

Manual Revised 21 January, 2019

Copyright

Trademarks

Warranty

With respect to the software and software media enclosed herein, Multicam Systems warrants the same to be free from defects in materials or workmanship for a period of 30 days from the date of purchase. In the event of notification within the warranty period of defects in materials or workmanship, Multicam Systems will replace the defective media. Warranty is limited to replacement and shall not encompass any other damages, including but not limited to loss of profit, special, incidental, consequential, or other similar claims.

Multicam Systems specifically disclaims all other warranties, expressed or implied.

For provisions of warranty relating to the Machine and accessories please refer to the warranty page at the end of this manual.

Head Office
Multicam Systems Pty Ltd
60 Enterprise Drive,
Beresfield South NSW 2322
Tel: 02 4964 1900
Fax: 02 4964 1911

Victorian Office
Multicam Systems Pty Ltd
11/40 Ricketts Rd
Mt Waverley VIC 3149
Tel: 03 9558 8022
Fax: 03 9558 8055

Queensland Office
Multicam Systems Pty Ltd
8/42 Trade St
Lytton QLD 4178
Tel: 07 3348 7866
Fax: 07 3348 4366

Table Of Contents

HEALTH AND SAFETY

General Safety And Machine Specifications

This manual and the safety guidelines contained within it are designed to enable the user to obtain the best performance from the machine over a long lifetime and to ensure the safety of the operator. Each person who will operate the equipment, perform a service or maintenance, or supervise its use must read the safety instructions in this manual. Please read it carefully before using or working on the machine.

The machine is a 3-axis router designed to cut and shape a variety of materials using either vacuum or manual clamping. The correct method of using these mechanisms is covered elsewhere in the users manual.

Ensure that you have read and understood the operational and health and safety sections of the manual before beginning to use the machine.

The machine has been designed and constructed to prevent the build-up of electrostatic charges during operation.

The machine is not intended for use in an explosive environment.

ALWAYS CONSULT SEPARATE MANUALS FOR ANCILLARY DEVICES SUCH AS MIST SPRAYER ATTACHMENTS, OSCILLATING TANGENTIAL KNIFE HEADS AND GANG DRILL HEADS FOR SAFE OPERATION.

Placement of the Machine and machine attachments

The machine must be placed with consideration given to the following;

- The actual floor space required by the machine and associated accessories and attachments.
- Adequate free air flow around vacuum pumps to reduce the risk of overheating.
- How you intend to move material to and from the cutting surface.
- OH&S requirements in relation to safe work practices. Refer to AS4024 & AS1473 for specific details in regards to placement of machinery. We recommend a minimum of 1meter of space around the machine. The amount of space left behind a machine, especially when placed in close proximity to a wall, must be sufficient so that workers, and our technicians can have safe and unobstructed access to allow for service and maintenance.

General Requirements

Noise Emission

The noise level produced by the machine will be dependant on the material being cut and the individual workplace environment. The appropriate health and safety regulations must be consulted when considering the location of the machine in the workplace, any sound-proofing or acoustic enclosure to be erected, or the provision of personal protective equipment (PPE) in the form of hearing protection.

A noise survey and specific risk assessments should be undertaken by a competent person in each workplace and for each work situation and material being cut, but in general the noise levels produced by the machine may be found to be in the region of the following readings:

Equivalent continuous A-weighted sound pressure level, L_{AEQ} above 90dB (A) when machining aluminium.

Equivalent continuous A-weighted sound pressure level, L_{AEQ} above 90dB (A) when machining medium density fibreboard (MDF).

There is not normally any impulsive noise from the machine.

It is recommended that precautions are taken against injury to machine operators and others from long-term exposure to noise, either by:

- enclosing the machine in an acoustic enclosure or booth fitted with closing doors
- requiring the operator, and other persons working or having cause to be nearby, to wear a suitable form of hearing protection.

Suitable mandatory hearing protection signs should be displayed on or adjacent to the acoustic enclosure or in the zone where the machine is located. The wearing of adequate protective equipment should be enforced

Operating Temperatures

The machines are designed to be operated between temperatures of -5ºC and +40ºC.
During operation machine components such as spindles, vacuum pumps and cutting tools will become hot. Care must me exercised when coming in to contact with these or any other items likely to become affected by heat.

Vacuum Pumps:
Vacuum pumps expel hot air at high velocity. Adequate provision must be made to allow for the safe and unrestricted expulsion of the exhaust air. DO NOT

- block the exhaust manifold
- place objects in close proximity to the exhaust air
- place vacuum pumps in confined spaces
- touch the surface of the vacuum pump as it will become hot during operation resulting in burns and scalding

MATERIALS SUITABLE FOR CUTTING

The machine is designed to cut, engrave and form a wide range of materials, as detailed below:

1. Plastics - all plastics can be cut, including the 'hard' plastics such as polycarbonate.

2. Woods -the majority of wood-based materials can be readily cut, including MDF, plywood and other composite boards.

   Note. When cutting some types of hard wood, e.g. mahogany, jarrah etc an appropriate form of jigging will need to be considered.

3. Metals - aluminium, brass and other soft metals can be readily cut. The hard metals such as stainless steel can be cut, but due care must be taken regarding the depth of cut, feed rate and cutter used. If in doubt, consult an appropriate person or the manufacturers agent or machine supplier.

   Other hard materials, such as carbides, should not be cut on the machine due to the risk of damaging the machine or endangering the operator.

   Note. When cutting metals a suitable lubricant should be used. Flammable fluids such as Methylated spirit must not be used.

4. Foams - all foam-based materials, such as sign foam, can be readily cut.

5. Composites and Laminates - some composites and laminates, such as fibreglass, can be cut. Please contact the manufacturers agent or machine supplier for more information.

Dust Extraction

Dust may be produced during machining operations and this may introduce a risk of fire or exposure of the operator to a hazardous substance. A suitable form of local exhaust ventilation (LEV) must be fitted to the machine to remove machining dust. Outlets for extraction hoses are fitted to the dust pickup head supplied with the machine. The hose must be firmly secured to the fitting.

Note. Always ensure that the extraction system is clear of obstructions and is in correct working order before using the machine. On dust extraction systems with manual cleaning devices follow the manufacturers recommendation for regular cleaning. Filters should be cleaned daily.

Mist Dispensing Systems

Mist dispensing systems, such as Trico and UNIST cooling/lubrication systems, are designed to assist in the cutting of ferrous and non-ferrous materials by delivering a fine mist of fluid under controlled pressure. Only use recommended fluids in these devices. Flammable fluids such as Methylated spirit *must not be used* and will cause damage to the devices. The use of non-recommended fluids in mist sprayers will void manufacturers warranty.

Corrosion

Some of the components of the machine may be subject to corrosion in unfavourable conditions. The machine should be used in a dry, moisture free atmosphere and should not be exposed to water-based fluids or lubricants. The machine should not be exposed to acidic or alkaline substances and it should be maintained in accordance with the servicing schedule.

GENERAL SAFETY RULES

It is essential that these instructions are made available to the machine operators. It is also important to hand them on if the machine is sold or given to another user.

The machine should only be used by a suitably trained operator. Other personnel within the organisation should be aware of the methods to stop the machine in an emergency.

The machine is intended for use solely as a 3-axis router and cutter of materials. It should never be hammered on, nor used as a general workbench, or for any other purpose than that for which it was designed. This is likely to damage the machine and may compromise operator safety. It may also void the manufacturers warranty.

The machine must not be operated after the consumption of alcohol or the taking of any medication or drugs which may cause drowsiness or loss of attention.

Machine must only be operated with all safety guarding in place. The machine cutting heads including spindles and oscillating knife heads are supplied with safety guarding. Do not operate the machine with safety guards removed. Do not activate clamping systems unless it is safe to do so. Ensure all clamping devices are clear of obstruction and that machine operators or other personnel cannot be injured prior to activating the device.

All emergency stop devices must be in full working order before operating machine.

We STRONGLY RECOMMEND that safety mats, laser safety curtains or other safety barrier devices be fitted to machines with a process area exceeding 3600mm in length or are fitted with aggregate head attachments or any other attachment that can pose a safety risk. Safety interlock outputs are available on the controller for the connection of these devices.

Machine must not be operated with danger or non-operation tags in place.

The machine must not be left unattended whilst in operation.

Personal Protective Equipment

It is recommended that the operator, and other persons working in the vicinity of the machine, use the following personal protective equipment in conjunction with the machine:

1. Hearing protection
2. Eye protection
3. Gloves when handling materials to be machined
4. Cut proof gloves when changing router bits, knife blades or other cutting tools.

Note: The supply and wearing of personal protective equipment is regulated by Workcover.

SAFE OPERATION OF MACHINES

The machine should be operated only in a well-lit environment in order that the operator can monitor the working of the machine and carry out safe loading and unloading. If the workplace is poorly lit, then additional lighting will need to be provided.

The machine's safety devices should be checked regularly for correct operation, e g the emergency stop mechanism and guarding.

When the machine is running, keep clear of any rotating or moving parts, e.g. the cutting bit, spindle, moving gantry and head parts.

Do not stand or lean on the machine whilst it is in operation. No persons other than the operator, or other authorised persons, should enter the working area whilst the machine is in operation.

Parts of the machine, particularly the cutting bit, may become hot during use. Take care when handling these parts and wear protective gloves if necessary.

Prior to operating the machine the following safety checks must be performed:

- Ensure that there are no tools or materials left on or near the rack and bearing rails of the gantry and the axis of movement.
- Ensure all guarding is in place including dust extraction devices.
- Ensure that all tools have been removed from the cutting area.
- Ensure that the guards are in place and correctly positioned.
- Ensure that the material being cut and the sacrificial board is no wider or thicker at any point than the width and height under the gantry and does not impede the movement of the gantry.
- Ensure that only recommended cutting fluids are used in mist dispensing systems.
- Ensure that all cutting tools to be used are sharp and in correct working order.
- Ensure that the filter for the vacuum pump is clean and that the correct vacuum valves are On/Off and are clear of any obstructions.
- Ensure that there is at least 500mm of clear open space around the machine.
- Ensure that all electrical and air supply components are in correct working order and have not been damaged in any way.
- Ensure that the vacuum pump inlet/outlet ports and pressure release valve are not covered.
- Ensure that the area around the machine is clean and free from obstructions which could lead to slips, trips or falls.
- Ensure that tools are correctly fitted in to collets and that collets are free from wear or damage.
- Ensure that ISO tool holders are in good condition and are free from wear or damage.
- If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to “wobble” the tool holder it is not correctly engaged.

It is recommended that checklists are used and logged at the end of each shift.

SAFETY DURING MAINTENANCE

Changes or additions to the main electrical system in connection with the installation and operation of the machine should always be carried out by a suitably competent and licenced electrical engineer.

Repairs carried out by untrained or unauthorised personnel may result in personal injury or serious malfunction and damage to the machine.

Do not alter or modify the specification of the machine as this may introduce an uncontrolled hazard or risk.

Servicing and maintenance of the machine should be carried out on a regular, planned basis and by a trained and authorised engineer.

Before working on any component or part of the machine always isolate the machine from the mains electrical and air supplies. These should be locked off.

SAFETY DURING CLEANING

To maintain optimum performance and obtain the longest service life from the machine, it must be cleaned regularly, either with a vacuum cleaner or a suitable cleaning solution. When cleaning, the machine must not be in operation. Compressed air must not be used to clean down the machine due the risk of particles being blown into the eyes or working parts of the machine.

Do not use pressurised jet washers or free flowing liquids to clean the machine. These will damage the machine.

WARNING LABELS

The following chart details the warning signs and labels and their locations on the machine. They are designed to assist and remind the operator to exercise caution when approaching the machine and the working area.

Operators should read carefully and take due note of the warnings and instructions for safe operation before operating the machine.

In addition to the warning labels on the machine the workplace should also be adequately signed for the wearing of personal protective equipment, e g eye and hearing protection, and the zones where these precautions are necessary clearly marked.

HAZARD NOTICE TABLE		
Hazard	**Requirement**	**Location**
Hearing Protection	Mandatory	On the Machine Head
Eye Protection	Mandatory	On the Machine Head
Electrical Power	Warning	On the Inverter Box
Electrical Power	Warning	On the Controller Box
Electrical Power	Warning	On the Power Box
Cutting Hazard	Warning	On the Machine Spindle
Emergency Stops	Mandatory	On both ends of the Gantry
Air Pressure	Warning	On the Machine Frame
Striking	Warning	On the X axis Bearing Cover
Striking	Warning	On the Tool Changer Cover
Striking	Warning	On the Machine Head
Cutting Hazard	Mandatory	On the Spindle Pickup Housing
Some labels may not be present on M-I machines		

Note. The warning labels must not be removed from the machine.

SAFETY RULES

MANUALS. Always read the appropriate manual or instructions before attempting to operate or maintain equipment. Make sure that you understand all instructions.

CORRECT SUPERVISION. When in doubt as to the correct way to operate the equipment always consult your supervisor.

OPERATION OF EQUIPMENT. Do not operate equipment unless maintenance has been performed on a regular basis, and the equipment is known to work properly. The machine must not be left unattended whilst in operation.

EMERGENCY SWITCHES. Ensure that you know where all emergency switches and stop buttons are, and you know how to isolate the machine from the main power supply.

WARNING NOTICES. Do not remove any of the warning notices or mandatory signs from the machine. Ensure that you comply with all such information.

SAFETY DEVICES AND GUARDS. Inspect all safety devices and guards to be certain that they are in good condition and functioning properly before any operation.

HAZARDOUS CONDITIONS. Do not operate the machine if unusual or excessive heat, noise, smoke or vibration occurs. Switch off the machine and report any unusual occurrence.

EARTHING. Ensure that the equipment is properly earthed. The equipment contains digital signal systems, which are effected by electrical spikes and noise.

ELECTRICAL ISOLATION. Before any repair or maintenance work is undertaken on the machine ensure that the main electrical power is fully isolated.

MAINTENANCE WORK. Do not reach into any control or power box unless the electrical power is turned off, do not touch any electrical equipment if hands are wet or when standing on a wet surface.

AUTHORISED PERSONNEL. Do not allow unauthorized personnel to have access to electrical enclosures containing electrical equipment. Do not allow untrained personnel to operate or repair the equipment.

REPLACEMENT EQUIPMENT. Always replace blown fuses with fuses of the same rating, size and type as the original.

PERSONAL PROTECTIVE EQUIPMENT. Wear the correct safety glasses, ear protection and foot protection at all times.

LOOSE EQUIPMENT. Do not operate any machine while wearing rings, watches, jewellery, loose clothing, neckties or untied long hair.

SAFE WORKING AREA. Keep the area around the equipment well lit, dry and free from external obstructions and waste material.

FLAMMABLE EQUIPMENT. Keep chemical and flammable material away from electrical or operating equipment.

FIRE EXTINGUISHER. Always have the correct type of fire extinguisher ready when machining combustible material and keep chips clear of the work area.

MACHINE OPERATION. Do not operate any equipment while anyone is close to any potentially hazardous area.

PINCH POINTS. Always beware of pinch points created by moving components of equipment.

LOOSE TOOLS AND EQUIPMENT. Remove any loose parts, work pieces, or tools left in the operating area of the equipment, which may come into contact with moving machinery.

WASTE MATERIAL. Do not remove splinters, chips, swarf or waste material with bare hands. Beware of sharp and burred edges on newly machined work pieces.

ADJUSTMENT OF MACHINERY. Do not adjust tooling or misters or extractor hoses while the machine is running.

CUT MATERIAL. Do not attempt to remove any cut material while the machine is running.

BRAKING OF MACHINERY. Do not attempt to brake or slow down the machine with your hands or any foreign device.

CUTTING TOOLS. Be cautious when working around cutting tools, their cutting edges are very sharp and can cause injury.

MOUNTING TOOL SURFACES. Make certain that all tool-mounting surfaces are clean before mounting tools.

ALWAYS THINK SAFETY FIRST

Vacuum Pump Operating Information

To protect the vacuum pump from premature failure it is important that the following procedures be adhered to;

Maximum Pressure

The maximum allowable pressure is not exceed -40kPa. Exceeding this value will cause the pump to overheat causing severe damage and voiding warranty.

Operating Temperature

The vacuum pump surface will become extremely hot during operation. Care must me exercised when coming in to contact with the surface of the pump or any other items likely to become affected by heat.

Vacuum pump performance is affected by the ambient and inlet temperatures. Vacuum pumps must not be operated where the inlet temperature exceeds 50oC. As the inlet temperature approaches 40oC the vacuum pump must operate at a reduced duty cycle allowing time for adequate cooling of the pump between cycles. Duty cycle is determined by ambient temperature. The higher the ambient temperature the shorter the duty cycle.

Vacuum pumps expel hot air at high velocity. Adequate provision must be made to allow for the safe and unrestricted expulsion of the exhaust air.

DO

- ensure that there is adequate space around the vacuum pumps to allow a free flow of air on all sides. We recommend a least 1m of free, unobstructed space on all sides.
- keep vacuum pumps clear of debris.
- check correct function of relief valve regularly.
- consult with specialists if you intend to duct exhaust from the pump. Incorrect ducting will affect vacuum pump performance and can lead to failure of the pump and overheating leading to injury or fire.

DO NOT

- exceed the maximum operating pressure.
- exceed the maximum inlet operating temperature.
- block the exhaust manifold.
- place objects in close proximity to the exhaust air.
- place vacuum pumps in confined spaces.
- enclose the vacuum pumps in enclosed spaces.
- place vacuum pumps under machines unless absolutely necessary. This will result in limited air flow around the pump causing premature failure and a drop in performance and cause surrounding machinery to become affected by heat.
- touch the surface of the vacuum pump as it will become hot during operation resulting in burns.
- move the vacuum pump from its originally installed location.

Failure to follow these instructions will void your warranty

Multicam Machine Electrical Start-up and Shut-down Procedures

In order to comply with safety issues it is important that the correct start-up and shut-down sequence is followed. These procedures are designed to ensure operator safety and machine longevity.

Start-Up Procedure.

- Turn on the air supply to the machine.
- Turn on the 3 phase power to the machine.

Shut-Down Procedure.

- Remove any tools from the spindle
- Turn off the 3 phase power to the machine.
- Turn off the air supply to the machine.

SERVICE AND MAINTENANCE

Overview

Multicam CNC Routing machines are driven using electronic drives on the X, Y and Z axes. The X axis runs on bearings along a chrome plated rail, driven by rack and pinion. The Y axis runs on linear bearings along 2 guide rails, and is also driven by rack and pinion. The Z axis runs up and down on linear bearings along 2 guide rails driven by a ball screw.

Multicam Models

Multicam S

1300 x 2500 process area and larger. Machines without automatic Tool Change units fitted.

Multicam SR

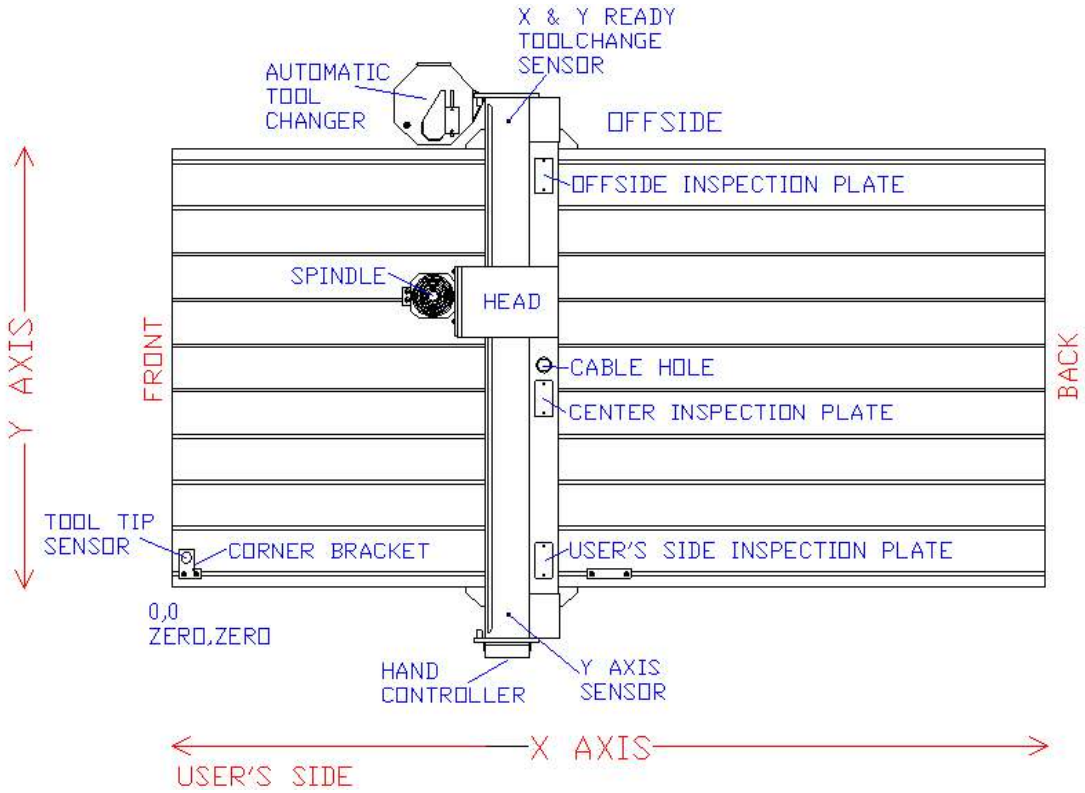
1300 x 2500 process area and larger. Machines with automatic Rotary Tool Change unit fitted.

Machine Beds

Aluminium T-Slot
Aluminium T Slot Vacuum Hold Down

Multicam CNC Routers are precision manufactured and thoroughly tested before leaving the factory. Due to the demands which are placed on these machines in your business it is important to observe certain operating procedures to avoid damage and premature failure.

Regular maintenance and servicing will ensure many years of reliable operation.



Preventative Maintenance

Cleanliness

Ensure that the machine bed is clear at all times of debris, tools and material as these can easily become caught under the X axis runners and cause damage to the machine. This could also lead to the current job being machined out of shape.

Using Lubricants

General Notes - If you are using lubricants for metal cutting we recommend that the lubricant be delivered through a microdrop mist sprayer. These units allow you to present the fluid directly to the point of cutting thus aiding in cooling and lubrication of the cutter. The Multicam bed should be wiped down after using a lubricant. Using corflute as a waste board will allow lubricants to be drained through the flutes and also minimise the amount of fluid entering the vacuum system.

Aluminium T-Slot - Keep slots clear and do not allow swarf to collect in the slots, especially if using lubricant.

Aluminium Vacuum Hold down - We do not recommend the use of lubricants on these machines as the fluid can collect in the vacuum pump. Use the supplied T slots if additional clamping is required or if the use of lubricant is absolutely necessary.

HSD Motors - If your machine includes a HSD motor it is important that you empty the air reservoir of any accumulated water. This unit is designed to trap any moisture that may be present in your air lines and prevent it from being passed through cooling system of the motor. If you find that an excessive amount of water builds up you may need to attach a separate water trap to your air line.

Dust

Excessive dust such as craftwood dust coating the machine can lead to failure of electronic components and computers. If the Multicam work area is excessively dusty it is important to seal the control box to prevent dust from entering and to locate your computer workstation away from dusty areas if possible. Generally removing dust from bearing slides etc is also recommended.

If a dust collector was supplied with the machine the filter MUST BE cleaned daily as instructed and the bag level checked. DO NOT allow the bag to become overfull. This could lead to excessive heat build up and fire.

Things To Avoid

Manual Jogging

The Multicam machine has sensitive drive cards and when the machine is pushed by manual methods (i.e. not moved by the hand controller), electronic surges are sent through the drive cards and can lead to failure. Please ensure the machine is only moved by the hand controller. Failure to do this will certainly lead to failure of the micro stepper drive motors.

Use of Air Guns

Using high pressure air guns to clear the table free of debris can lead to clogging of crevices and seizing of crucial moving parts.

T Slot Tables

Make sure that when screwing bolts into the T nuts that they do not project through the bottom of the nut. Failure to do this may result in damage to the Table Top extrusion.

Special Notes for High Frequency Motors

If your machine was supplied with either an HSD or Perske High Frequency motor you should pay careful attention to the following points

- Always use genuine collets, ISO 30 collet holders, cover nuts and spanners on your motors. Third party items of this nature may not be made to the same standards as original parts and can cause serious, and expensive, damage to the motors. ***Use of non genuine parts will void the manufacturers warranty of these motors***.
- When fitting collets to the motors ensure that the collets are free of dust or other contaminants. Dust and grime on the collets will prevent the taper of the collet from seating correctly in the spindle, thus preventing correct tool holding.
- Ensure that the threads of the cover nuts *and* motor are free of dirt. If the cover nut is difficult to tighten then you probably have a build up of dirt on the thread. *Never* force a cover nut on to the motor thread.
- In the case of HSD ATC (Quick Change) motors *always* wipe the taper of the holder with a clean rag to remove dust etc before fitting it to the motor. Failure to do so will result in damage to the internal taper of the motor once again causing incorrect seating of the ISO 30 holder.
- ISO 30 Holders as used on the HSD ATC motors are precision components, you must ensure that these items are stored in such a manner as to prevent damage to the taper and surface rusting.
- Do not keep ISO 30 Tool Holders mounted in the HSD motors for long periods if not in use, such as weekends. The close tolerances of these components means that the holder may become “stuck” in the HSD motor.
- If your machine will not be used for long periods it is advisable to place a clean rag into the HSD ATC motor. The rag may be lightly impregnated with oil. This will prevent surface rusting on the inside of the motor.
- To remove surface rust from collets and collet holders use a very fine grade of wet and dry paper to rub off the rust. You should also remove surface rust from the shanks of tooling prior to inserting the tool into a collet.
- If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to “wobble” the tool holder it is not correctly engaged.

Daily Maintenance

- Multicam tables should be cleared of any debris, tools or other items which may obstruct the machines operation.
- If any lubricants have been used the table should be wiped down after use.
- Clear any reservoirs and air filters associated with Vacuum tables or HSD motors. Pay particular attention to compressed air line filters that can accumulate moisture very quickly.
- On machines fitted with self-draining micro air filters it is important that the compressed air supply not be disconnected from the filter. The self draining mechanism is activated when the air pressure supply drops *BELOW* a certain level. Leave the air supply fitted to the filter and turn the air compressor off. As the pressure drops the self draining mechanism will automatically drain any accumulated moisture from the machine.
- If a dust collector was supplied with the machine the filter MUST BE cleaned daily as instructed and the bag level checked. DO NOT allow the bag to become overfull. This could lead to excessive heat build up and fire.
- ALWAYS warm up spindles prior to commencement of work. When starting the spindle for the first time each day, or after a long period of non-use allow the spindle to undergo a short warm up period. This will allow the bearings to slowly reach a uniform operating temperature. the following cycle speeds are recommended;

  50% of maximum speed for 2 minutes
  75% of maximum speed for 2 minutes
  100% of maximum speed for 1 minute

Scheduled Servicing and Warranty

A scheduled service must be conducted by a Multicam technician at least every 3 months or after every 250 hours of running time. Failure to have the machine serviced in accordance with this schedule will constitute a breach of the Terms of Warranty and render warranty void. Regular servicing and inspection will ensure that your machine maintains peak performance. Please contact your nearest Multicam office to arrange scheduled services.

Important Note

Failure to follow the above procedures will result in damage to spindles and vacuum pumps and will void terms of the warranty provisions of the machine and accessories.

Calibrating Your Multicam

After a period of time it may be necessary, due to normal wear and tear, to re-calibrate the machine.

Re-calibrating the machine, through the ToolPath Configuration software, lets the software tell the motors how far to travel to give you your required size.

NOTE: Standard tape measures are inaccurate for this purpose, it is recommended that you use a 1metre engineers rule or calibrated tape measure.

Steps to follow

NOTE: Your machine must be turned on and communicating with your computer in order for you to re-calibrate your machine. If you need to change the calibration of your machine then you will must delete any jobs at the machine and re-send them otherwise the new calibrations settings will have no effect on those jobs.

- Place reference markers (such as adhesive tape) on the X and Y axes and perform a 1 metre measured move in both the X and Y axes using **FUNCTION 6**.
- Measure the actual distance that the machine has moved in both the X and Y axes and record the values in the X direction and the Y direction.
- Go back to your computer and start ToolPath if it is not already started.
- From the Setup Menu choose the Calibration tab. The settings currently displayed will be the settings that we are going to recalculate and correct if necessary.
- Divide 1000 by the number you got as your reading for X. (example 999.5).
- Take the new value and multiply it by the current value of calibration in your configuration tab. (example 63.2629)

  So the formula is

  1000 divided by (x reading for 1 metre) multiplied by (x calibration)

  In this case

  (1000÷999.5) x 63.2629 = 63.2945

  The new value of X calibration is **63.2945.**

- Repeat this process with the Y axis.
- Enter the new values if required the close the Setup dialogue box. Click on OK to update the machine controller.
- Repeat the process starting at (1.) to check your settings.

Remember to delete any jobs at the machine and re-send them otherwise the new calibrations settings will have no effect on those jobs.

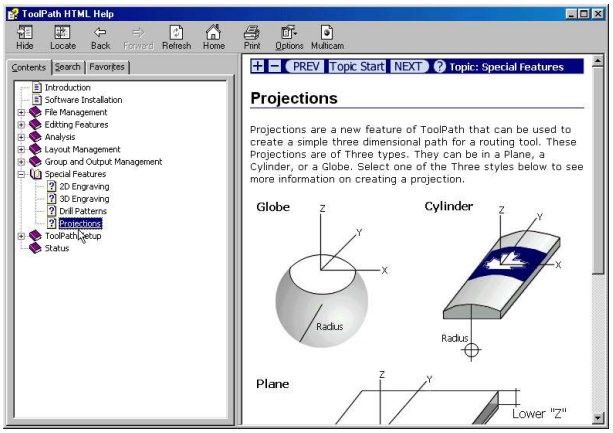
TOOLPATH INSTALLATION AND OPERATION

Online Help System

ToolPath incorporates an advanced Web Style Online Help System that may be accessed at any time from the Help Menu

Try using the Online Help System before contacting us. You will find most of your questions will be answered by consulting the Help Topics.

Help Topics will allow you to search for help by using either the **Contents**, or **Search** headings (also known as tabs).



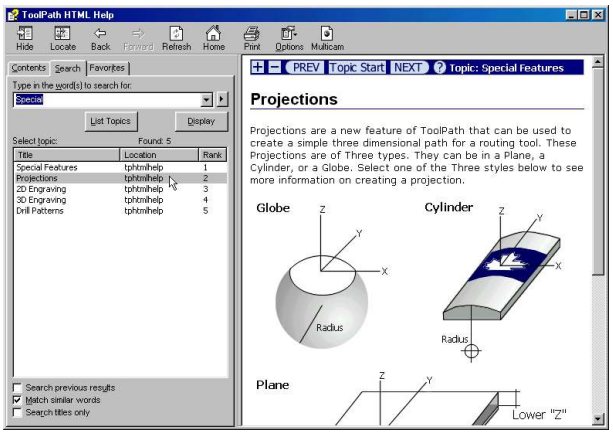
For example to find help on Projections in the Special Menu follow these steps;

- double click on **ToolPath for Windows Software**
- go to the Help Menu and click on **help Topics**
- click on to + symbol next to **Special Features** to expand the help topics for Special Features.
- click on **Projections**
- information on the Projections feature will be displayed in the window to the right of the help index.

Many of the help topics have “hot spots” that, when clicked will bring up additional information.

Hot spots are usually associated with Screen shots of the ToolPath menus, but may also be associated with links to other related help topics. You can tell when you have found a “hot spot” when your mouse pointer changes to a pointing finger Click on the “hot spot” and additional help will be displayed.

Another way of obtaining help is to click on the **Search** tab. This will allow you to type in words you are looking for in the help file. Type in a word or feature that you wish to obtain help on and then click on the **List Topics** button. A list of topics that match your search criteria will be displayed below the search entry field.



Click on the matching result that you wish to view and then click on the **Display** button. Information on this topic will be displayed in the window to the right.

The web style help system is very powerful and flexible and will allow you to find information on a given topic quickly and easily. You may print out any information that you require by selecting the Print button at the top of the Help menu window.

INTRODUCTION

Welcome to the Multicam **Australian Machine Controller (AMC)** and **ToolPath** instruction manual. These two tools work in conjunction to create a very powerful motion control system that allows you to interface most popular CAD / CAM software packages, with your **AMC** controlled Multicam CNC machinery.

There are three main components to an automated machine:

Design Station. The design station is a Windows based personal computer such as a Pentium, running a CAD or design software package such as AutoCAD, CASmate, CorelDraw, Microstation, MasterCAM etc that outputs DXF, HPGL or NC code.

AMC Interface. This is combination of software and electronics which actually converts the computerised design created by the design station to motor movements which drive the tools on the machine. The interface consists of the **ToolPath** software and the Multicam controller. These two devices work in concert to produce your job.

Machine. This is the equipment on which the work is actually done. The machine may be any of the current model Multicam machines.

This section concerns itself with the **Multicam AMC Interface**. There are several parts to the interface:

AMC ToolPath. This is a Windows ™ software program that facilitates the actual data conversion from a CAD design format such as **DXF , HPGL, or NC** to a **"ToolPath"** that can be used by a machine. Although there are many excellent CAD packages available for reasonable cost, they generally provide a design consisting only of line and point data. No consideration is given to tool type and size, cutting speeds, material layout and usage etc. The ToolPath software facilitates these machine operation requirements.

AMC Multicam Operating System (AMC O/S). The **AMC ToolPath** is run on the PC design station by the design operator. **AMC O/S** operates the control circuit board of the machine. It actually turns the motors and provides the keyboard interface to the machine's operator.

AMC Electronics. The electronics are the physical hardware that make up the interface. The main component is the **controller**. It is a microprocessor based circuit board and could be considered to be the "brains" of the machine. It runs the **AMC O/S** software which actually operates the motors and the operator's keypad. The operator's keypad is a separate component connected to the controller and is generally referred to as the **"Sub Console"**. Communications between the machine and the design station is established via a communications module that plugs in to design station and a long range communications (LRC) cable.

This manual describes the implementation and operation of the **AMC** system focussing primarily on the operation of the ToolPath software. The first section concentrates on the interface software that allows your CAD design to be turned into tool paths for your machine to follow. The second section provides a quick example of how to manipulate your CAD drawing in ToolPath, preparing it to be down loaded to your machine. The third section is an overview of the Multicam machine features. This section describes what you do with your CAD design once you have converted it to tool paths and transmitted it to your Multicam machine.

INSTALLATION

Hardware Installation

In order to transmit jobs and information to the Multicam machine there must be a physical connection between your computer and your machine. The connection is only required to send files to the machine. Once the information has been sent to the machine the computer may be disconnected. The machine does not require the computer for the processing of jobs.

The method of establishing the link between the computer and the machine is through the RS232-RS485 connector.

Install the RS232/USB to RS485 Converter

The USB/RS485 Converter is the interface between your computer and your Multicam machine. It is used to convert the low power USB signal from your computer to a high power industrial signal more suitable to long range communications. The converter contains two devices. A USB interface device and a COM Port device. Drivers for both devices must be installed for the converter to function. There are two models of this Converter.

Model 485-1A is compatible with 32bit versions of Windows XP, Vista and Windows 7.

Model 485-1B is compatible with 64bit versions of Windows XP, Vista and Windows 7 and Windows 8.

In order for this device to function you must install the drivers on to your computer. The drivers supplied are compatible with 32bit and 64bit versions of Windows XP, Vista and Windows 7.

Installation of the drivers will vary slightly from platform to platform, but the basic procedure is the same.

Installing under Windows XP
When the device is plugged in to a USB port on the computer, the New Hardware Wizard will launch. Follow these instructions to install the driver;

- The Toolpath CD must be in the computer CD ROM drive before you commence.
- On the first window choose "No, not this time" then click Next
- Choose "Install from a list or specific location (Advanced)" then click Next
- Untick "Search removable media"
- Tick "Include this location in the search"
- Click the browse button and browse to the USB driver folder on the Toolpath CD. Click on the folder name then click on OK
- Click Next
- The driver will install, then click Finish
- You will need to repeat the process to install the second set of drivers.
- Reboot the computer

Installing under Windows Vista
When the device is plugged in to a USB port on the computer, the New Hardware Wizard will launch. Follow these instructions to install the driver;

- The Toolpath CD must be in the computer CD ROM drive before you commence.
- Click "Locate and install driver software (recommended)" If you are advised that Windows needs permission to continue click on Continue
- Click "I don't have the disc. Show me other options"
- Click "Browse my computer for driver software (Advanced)"
- Click the browse button and browse to the USB driver folder on the Toolpath CD. Click on the folder name then click on OK
- Click Next. If the security window appears click on "Install this driver software anyway"
- The driver will install, then click Close
- You may need to repeat the process to install the second set of drivers.
- Reboot the computer

Installing Under Windows 7
When the device is plugged in to a USB port on the computer, the New Hardware Wizard will launch and attempt to load the drivers, which in most cases is not successful

- On the Device Software Installation window click on Close
- Click the Start button then right click on Computer
- Left click on Manage
- In the left panel click on Device Manager
- In the Right panel you will see under Other devices USB <-> Serial.
- Right click on USB <-> Serial then left click on Update driver software
- Click "Browse my computer for driver software (Advanced)"
- Click the browse button and browse to the USB driver folder on the Toolpath CD. Click on the folder name then click on OK
- Click Next. If the security window appears click on "Install this driver software anyway"
- The driver will install and a new device called USB Serial Converter will appear under the Univesial Serial Bus controllers list. Click Close
- Under Other devices right click on USB Serial Port then left click on Update driver software
- Click "Browse my computer for driver software (Advanced)"
- Click the browse button and browse to the USB driver folder on the Toolpath CD. Click on the folder name then click on OK
- Click Next. If the security window appears click on "Install this driver software anyway"
- The driver will install and a new device called USB Serial Port will appear under the Ports (Comm & LPT) list then click Close
- Reboot the computer

Installing Under Windows 8/10
When the device is plugged in to a USB port on the computer, the New Hardware Wizard will launch and attempt to load the drivers. Some Windows 8 systems will already have the necessary device drivers present and will automatically load them. In other cases you can use Windows to search online for the correct drivers. This may tak a few minutes to accomplish. If this is still not successful point the wizard to the Toolpath CD and go to the Windows 8 Driver folder to load the driver.

Software Installation

All the required files for the installation of **ToolPath** are found on the CD that was included with this package. Our first step is to load the **ToolPath** software into the design station.

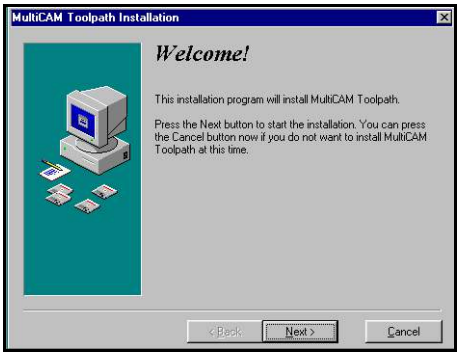
Installing the software

Insert the ToolPath CD into the CD ROM drive of the computer into which you wish to load **ToolPath**. If the installation routine does not start automatically then from your Windows desk top, select the **Start** button, and click on **Run**. The **Run** box should appear. Type in the drive letter for your CD (usually Drive D:) followed by :\setup.exe or use the **Browse** button to locate the setup file on your CD.

Click on **OK**, to continue and ToolPath installation will begin.

ToolPath Installation Procedure

Once you have started the Setup procedure, the first screen you see should look like this:



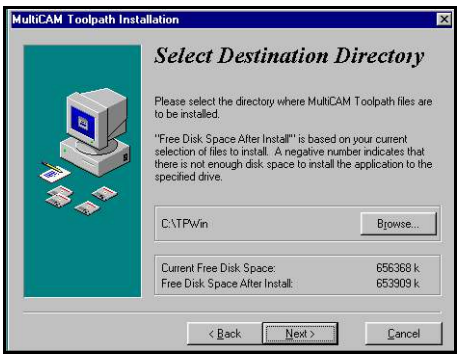
You can simply follow the instructions as they appear on the screen. If at any time you wish to stop the installation, click on **CANCEL**. Otherwise, click on **NEXT**.

The next step is to set up a working directory and program group for **ToolPath**. This defaults to:

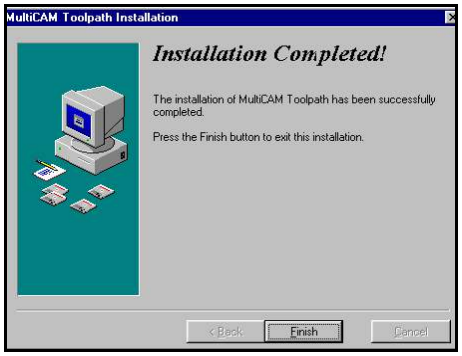
C:\TPWin

Click on **NEXT**.

You will next see the progress bar for the file transfer. The file transfer will require that you insert extra ToolPath disks when prompted.



Once file transfer has finished, you will see the following notice that the setup procedure was complete.



Additional files will now be copied from your CD. These are setup files specific to your machine. *If you were supplied with a Vendor Diskette you should insert it into your 3.5" floppy disk drive now.*

Click on **Yes** when prompted to do so.

Next you will be asked to update the HTML Help. Click on **Yes**. Then click on OK.

Following installation you will need to reboot your computer for the settings to take effect.

You will now have a program group named Multicam Toolpath, and your ToolPath icon will appear on the Windows Desktop. To start **ToolPath**, double click on the ToolPath icon.

IMPORTANT NOTE :
If you are using an RS485 converter that connects to your computer through a USB port you must install the USB device driver before the unit will function. When you first plug the converter in to your computer the *New Hardware Wizard* will start and guide you through the software installation process for the converter. The USB driver files are located on your Toolpath CD.

Understanding The Controller Software

There are two important files that make up the controller software package:

TPWin.EXE is the **ToolPath** software. It is the main program and must be loaded onto an IBM compatible design station and run on it.

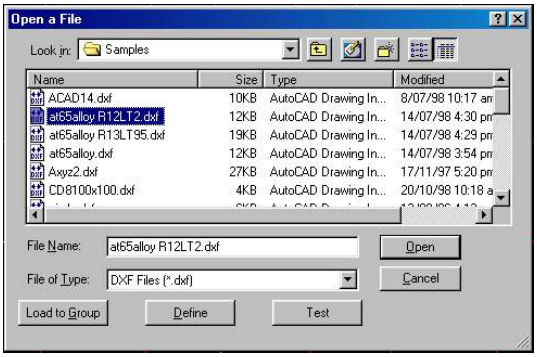
AMC Operating System. There are several variations depending on the machine and drive system you are using. They are called "**.IMG**" files. The one you will require is on the shipping disk. The default file for Multicam Machines is "**AXYZ_OS.IMG**". The operating system must be loaded into the same directory that **TPWin.EXE** is installed. It does not run on the PC, rather it is down loaded to the Multicam machine via the communication interface by **TPWin.EXE.**

ToolPath requires a mouse to operate.

MULTICAM TOOLPATH OPERATION

Starting The Program

To start **ToolPath** double click on the **ToolPath** icon on your Windows desktop or go to the **Start** button on your task bar then select **Programs**|**ToolPath for Windows**. The ToolPath splash screen will appear shortly followed by the **Open a File** dialogue box.

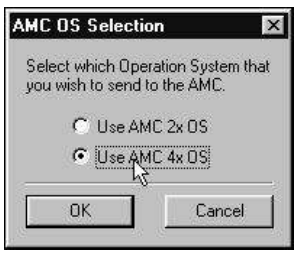


Select a file to load or if you don't have any files available browse to the Samples folder and open one of the sample files.

ToolPath will support the following file types. **DXF (AutoCAD)**, **PLT (HPGL), NC (G-Code)**, **PCD (P-Code)** or **RDY (ToolPath Ready)**. You may find that **NC** files may not have the suffix **.NC**. If this is the case, they should be renamed to include the suffix **.NC**.

Normally Toolpath will then load the selected file and display it on the Toolpath Main Screen. However the **Available Machines** dialogue box will appear if you have more than one machine connected to your computer or no connection can be made with a machine. If you have more than one machine connected to your computer you will need to choose from the list of available machines, the correct machine for the current job, then click on **OK**. If ToolPath is not able to establish a connection with your machine you should click on the **OK** button. Clicking on **Default** will load a set of configuration settings that may not be the same as the configurations for your machine.

If you are presented with the dialogue box shown below this means that whilst ToolPath is able to communicate with your machine, there is no operating system on the machine and it must be downloaded to it

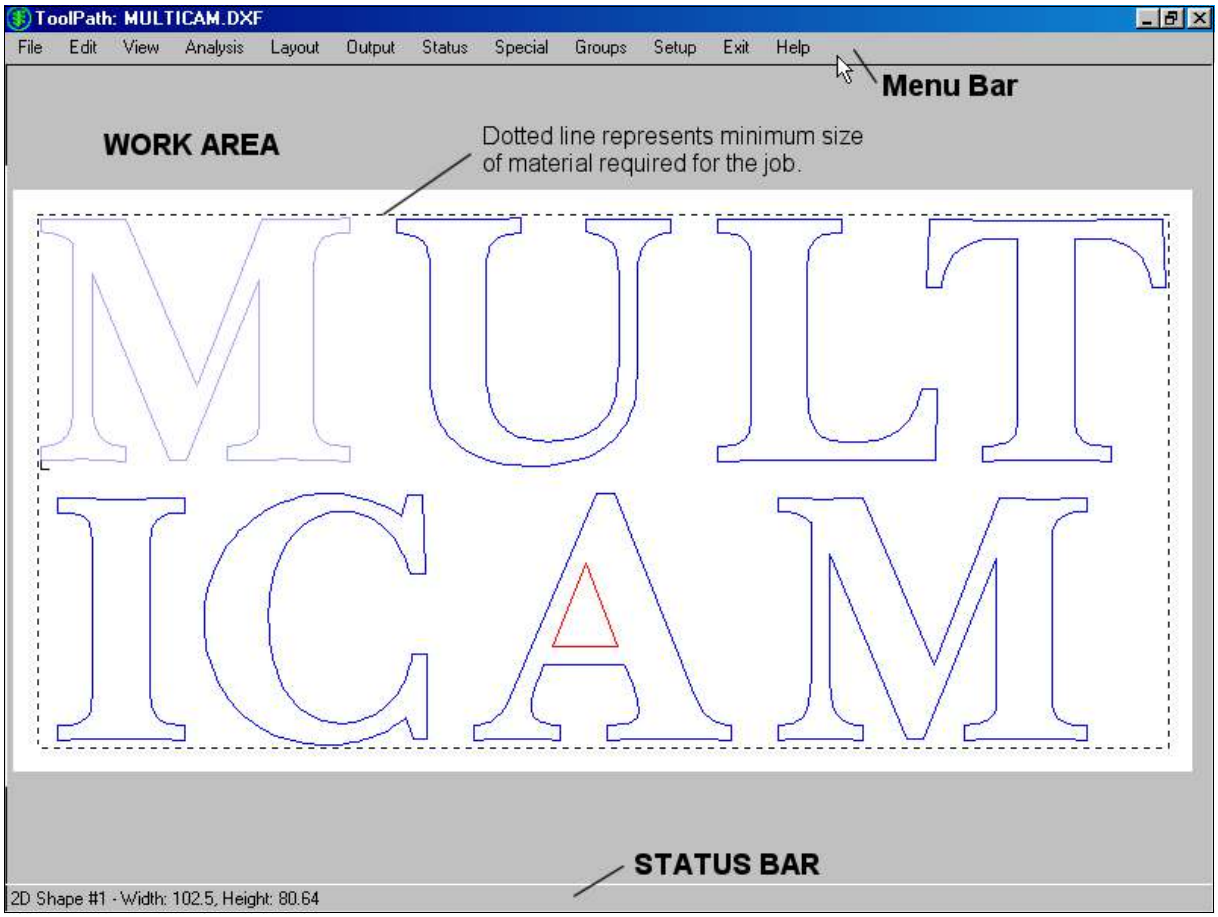


To transmit the operating system to the machine select **Use AMC 4x OS** and click on **OK.**

You will notice a progress bar at the bottom right of the ToolPath screen that will show the progress of the transmission. Once the operating system has been sent to the machine you can progress as normal.

The ToolPath screen consists of three main areas. These are the **Menu Bar, Work Area and Status Bar.** The Menu Bar is across the top of the screen. It lists all the available menus. To select a command click on it with your left mouse button. Depending on your choice the menu will change to present other commands or choices.

The work area makes up the bulk of the screen and displays your job file. This is where you can view your job and edit it to a certain extent. Individual shapes in the work area be selected (highlighted) by moving the cursor close to a shape boundary and clicking with your left mouse button. With some menus commands (such as Edit|Erase) you can also select shapes by dragging a bounding box around the shape or shapes you want with your left mouse button.



The Status Bar is at the bottom of the screen and displays information about your job (such as shape size, overall size, number of shapes etc) it will also display a progress bar when transmitting jobs to your machine or importing files.

ToolPath uses a colour coding system for easier identification of shapes.

Red shapes are inside shapes that are cut in a clockwise direction. These shapes are compensated to the right of the cutting direction

Blue shapes are outside shapes that are cut counter-clockwise. These shapes are compensated to the left of the cutting direction

Green shapes are open shapes. Open shapes can not be compensated.

Yellow is the colour used to define the material size, it appears as a dashed box around your design. Yellow is also used as the preliminary colour in engraving and is also the colour of the directional/compensation pointer that each shape has.

Crosses (drill points) appear as **Green,** crosses. They will not be compensated, regardless of the colour.

The following sections describe the functions of each of the Main Menu items. For most simple cut out jobs the only two items used are **Analysis** and **Output**. If you are using the system for the first time you may wish to skip to these two sections.

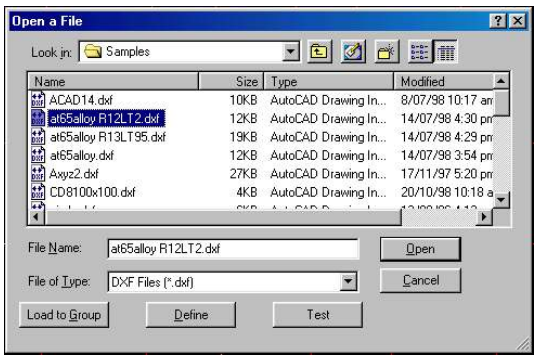
FILE

Clicking **File** on the Menu Bar will bring up the dialogue box you see to the right. This feature allows you to manoeuver through the computer and/or network directories to find and import or export job files. Click on **Import** to bring a file into ToolPath, **Export** to export a file from ToolPath or **Cancel** to cancel the operation.

If you have chosen either **Import** or **Export** a file the ToolPath File manager window will appear. This is basically the same as your standard Windows file manager with the addition of a few extra buttons, these being **Load to Group**, **Define**, and **Test**.

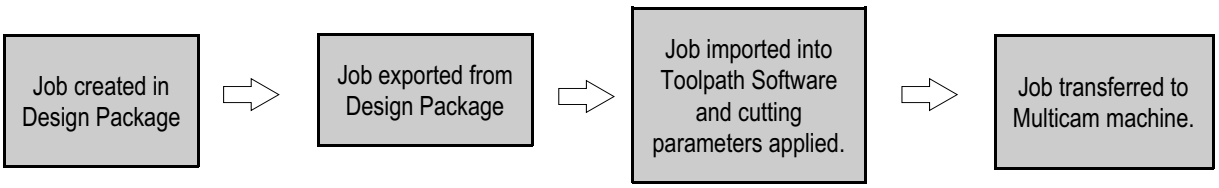
You can save the currently loaded file in Toolpath in any of the supported file types by clicking on the **Save** button. Choose a destination and file type and click on the **Save** button to export the file.

ToolPath will support the following file types. **DXF (AutoCAD)**, **PLT (HPGL), NC (G-Code)**, **PCD (P-Code)** or **RDY (ToolPath Ready)**. You may find that **NC** files may not have the suffix **.NC**. If this is the case, they should be renamed to include the suffix **.NC**.



Program Flow

The diagram below shows typical steps involved in transferring DXF and PLT files to the Multicam machine.



General Notes on Software Compatibility

DXF and PLT files allow the greatest flexibility in machine Toolpath creation. A DXF or PLT file imported in to the Multicam Toolpath software is able to make full use of the advanced machining features of the Toolpath software. Features such as Tool Compensation, Tool Assignment, Ramp Plunging, Tool Lead In, Multipass, 2D & 3D Engraving, Automatic Plunge Hole creation allow the machine operator to have full control over the finished product. DXF and PLT files also allow the machine operator to change the tool compensation to suit the cutting tools available without having to re-draw the job in the design software. This is beneficial if tooling is re-ground or the operator decides to use a different tool than originally intended by the designer.

General Compatibility Notes

- DXF files are most compatible
- If available export DXF files as polylines
- If DXF is not available use HPGL or PLT files
- If you intend to use NC (also known as G-Code files) your software must have a POST processor that is compatible with the machine controller.

AutoCAD Notes

- Export/Save DXF files compatible with AutoCAD R10, R11, R12, R13, LT2, LT95, LT97, LT98
- Do not export non machinable data (dimensions, text, title blocks, reference blocks, bitmap images, OLE data)
- On some later versions of AutoCAD spline curves and ellipses may need to be exploded.
- Shapes containing spline curves may need to be converted to polylines.
- Text will need to be exploded if you wish to export the text. You must have the Express Menu option installed in AutoCAD to explode the AutoCAD text.

CorelDraw! Notes

- Export DXF files compatible with AutoCAD R10, R11, R12, R13, LT2, LT95, LT97, LT98
- Export in mm. On some versions of CorelDraw! Export will be in inches despite drawing being made in mm. In these cases defining the Import filter for DXF files in Toolpath to inches will resolve this.
- Do not export non machinable data (dimensions, text, title blocks, reference blocks, bitmap images, OLE data, gradients, shading etc)
- Shapes need to be outlined with a hairline to export correctly.
- Export text as curves

Importing Layer Information from Design Programs

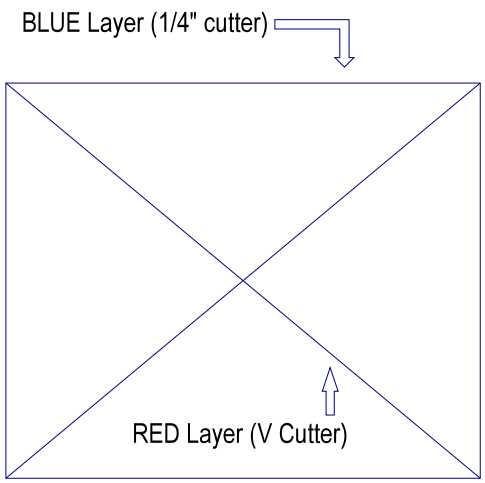
If the CAD package you are using supports the exporting of layers in DXF files it is possible to have ToolPath automatically assign these layers to a corresponding group in ToolPath. Programs such as AutoCAD, MultiCAD and DesignCAD support this feature. This can be useful in that it allows you to set up templates in ToolPath that can then use the layer information to create Tool Change jobs automatically. This is known as TOOL MAPPING and when utilised has the ability to streamline your entire manufacturing procedure and simplify the production process.

In order for mapping to work, the layers in your CAD program must be named to match group names that you want the information to appear in in Toolpath. That is WHITE, BLUE, GREEN, YELLOW, RED, MAGENTA, CYAN, CYAN and GRAY.

Note: If you are assigning anything to the GRAY layer it must be spelt GYRAY to differentiate the gray layer from the green layer. Toolpath only recognises the first two letters of the layer name so you can use layer names such as RED V Cutter for ease of operation in your CAD program.

When creating your design in your CAD package you can you can decide what part of the job is to be cut with which tool and create a layer that will represent that tool.

For example in this part drawn in AutoCAD the outside box is drawn in a layer named BLUE. The designer has decided that the outside box is to be cut with a 1/4" slot drill. The diagonal lines are drawn in a layer named RED which is associated with a V cutter.



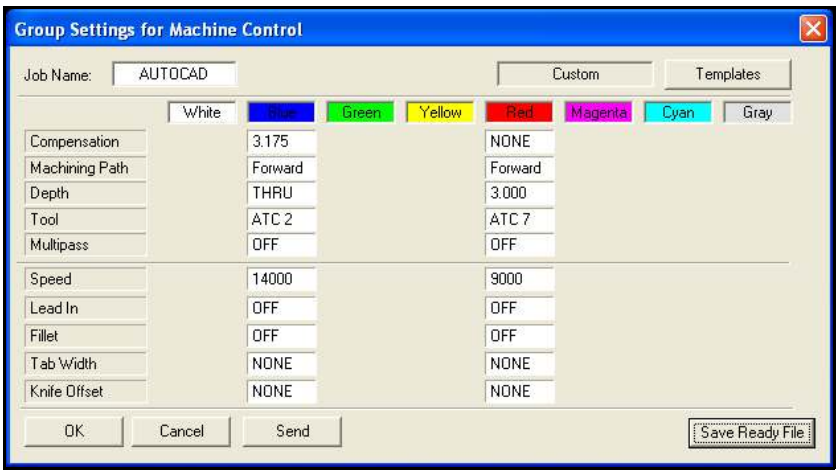
It is important that this information is matched in Toolpath so that the BLUE group is actually assigned to a 1/4" slot drill and that the RED group is assigned to a V cutter. If you have an Automatic Tool Changer on your machine then you need to ensure that the BLUE group is actually associated with a 1/4" slot drill in the Automatic Tool Change unit and that the RED group is associated with a V cutter in the Automatic Tool Change unit.

The use of a Tool Mapping Table similar to that shown below will assist your designers and machine operators and ensure the accurate flow of information.

CAD Program Layer Name	Toolpath Group	Tool Changer Tool Number- can be in any order	Tool Description
White	White	1	1/2" Compression Cutter
Blue	Blue	2	1/4" Slot Drill
Green	Green	3	1/8" Downcut
Yellow	Yellow	4	5mm Plunge Drill
Red	Red	7	V Cutter
Magenta	Magenta	6	Profile Cutter
Cyan	Cyan	5	1/2" Roughing Cutter
Gyray	Gray	8	

When this job is imported in to Toolpath you will notice that in the OUTPUT screen the BLUE and RED groups are active.

In the example below, which is being used on a machine with an Automatic Tool Change Unit, you will also notice that ATC2 is mapped to the BLUE group and the ATC7 is mapped to the RED group. This simply means that the machine will pick up tool 2 from the ATC unit when it needs to cut the outer perimeter of the job and tool 7 when it needs to cut the diagonal V grooves. Of course it is up to the machine operator to make sure that the correct tools are actually loaded in to the correct tool positions on the machine.



Refer to information commencing on page 95 about configuring tooling in Toolpath and setting up Toolpath when using Automatic Tool Changers.

In the absence of any layer information, or if the layers are incorrectly named, all information will come into Toolpath in the default WHITE group.

Toolpath Command Line Options and Machine Automation

If you are developing applications yourself that you wish to have DXF, PLT or NC files automatically transferred in to Toolpath, then you can launch Toolpath using a command line expression. The command line expression can be executed from a line of code within your application, a simple batch file, an application macro or even from the Windows *Run* command. The command line expression would look like this;

C:\Tpwin\Tpwin.exe *filename* (optional commands [s] [r] [d])

Where the available command line options are;

s	Autosend (Run Toolpath, Load File, Send To Machine, Close Toolpath)
r	Autosend and Autorun (Run Toolpath, Load File, Send To Machine, Close Toolpath, Run File On Machine, Delete File Off Machine)
d	Autosend and Autodelete (Run Toolpath, Load File, Send To Machine, Close Toolpath, Run File On Machine, Delete File Off Machine)

In order for the Automation process controlled by the command line options to work, you must make the following changes in the TPWin initialisation file, TPWIN.AXY

You can use Windows Notepad to edit the TPWin.axy file which is located in the C:\TPWIN folder.

These entries must appear in the C:\TPWIN\TPWIN.AXY file. If the section [Automation] is not present you will need to create it and type in the entries below it. Place the [Automation] section at the end of the TPWin.axy file.

[Automation]
Enable=1
Minimize=0
Table=##### MACHINE_ID of Active Table (enter the machine Flash ID number in place of #####)

General Notes for NC (G-Code) Files

NC (also known as G-Code) files are commonly created by more advanced CAD/CAM software packages and allow the creation of full 3D modelling toolpaths. The NC file contains ALL machining data including cutter compensation, direction of cut and tool selection. Unlike DXF files, this type of file does not allow any alteration to be made by the machine operator, nor can the advanced machining features of the Multicam Toolpath software be applied to NC files. The design package will require a compatible POST processor to create the NC files.

The NC file format was developed as a "language" for controlling the motion and operation of large machines. With the huge variant of machines in the Industrial Automation industry, the NC file format does not have a "totally" fixed standard. Nevertheless, there are many features of the NC file format that are common between the many versions.

NC files have two key structures; G-codes and M-codes. Within ToolPath, G-codes are interpreted as shapes, while M-codes are interpreted as one of a few motions.

With all the variable possibilities of G-code and M-code commands, ToolPath importer limits what commands it translates to those commands that deal with 2D and 3D motion and shape descriptions.

The NC importer can handle the following G-codes:
G00 – Rapid Positioning
G01 – Linear Motion
G02 – CW Circular Motion
G03 – CCW Circular Motion
G90 – Absolute Positioning Mode
G91 – Relative or Incremental Positioning Mode

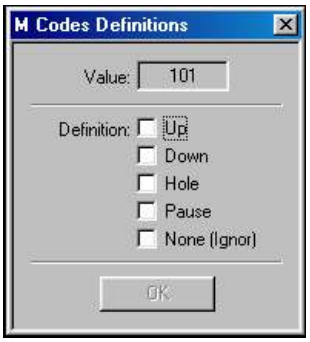
The corresponding position commands of X, Y, Z, I, J and R are also supported.

G-codes can also be written in two styles; "modal" and "non-modal". Modal means that a G-code command remains active until the next G-code command. Non-modal means that each line in an NC file must have an appropriate G-code command. Although ToolPath can handle either case, it is recommended to use non-modal for easier debugging of NC files.

ToolPath allocates M-codes to specific actions; Move Up, Move Down, Pause, Hole, and Ignore.

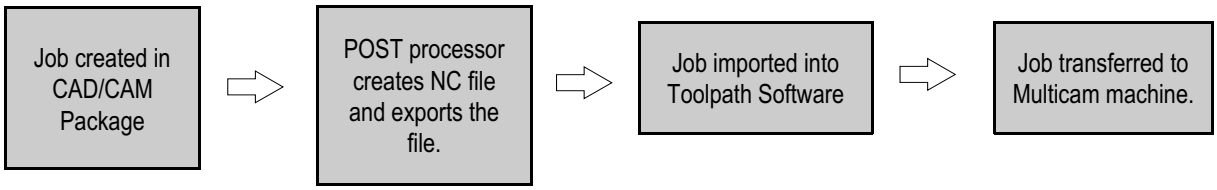
Many M-codes can be simply ignored. However, it is important to understand your design software and how it creates NC files. Sometimes M-codes are used instead of G00 to end a shape. Ignoring these M-codes could cause files to be misread into ToolPath resulting in poor quality cut jobs at the table.

When an M-code is discovered in an NC file that ToolPath has not seen before, ToolPath displays the M Codes Definitions dialogue shown to the right in order for you to choose between the 5 different options. If you believe that you've made a mistake in assigning an M-code to an inappropriate option, you can delete the assignment in the TPWIN.AXY file.



Program Flow

The diagram below shows typical steps involved in transferring NC files to the Multicam machine.



NC (G-Code) Specifications:

The controller on the MultiCam machines will accept G-Codes (or NC Codes) delivered through the ToolPath software to the machine. The controller is based on generic Fanuc G-Codes. Detailed specifications and sample code is available upon request.

1. The ToolPath G-Code input filter will perform linear, circular and helical interpolation
2. Multicam ToolPath handles G-Codes in the same fashion as it does DXF codes, i.e as a foreign Cad-System drawing. Files are converted to a propriety file format prior to transmitting to the machine.
3. Commands that activate and deactivate (G or M codes) appliances are ignored.
4. POSTs developed for the Multicam Controller should be non-modal. Movement commands must always be represented by the correct X, Y or Z position for each line of code.
5. Following is a list of acceptable codes:

G00	Rapid motion (e.g. G00 X100.0 Y100.0)
G01	Motion at feed rate (e.g. G01 X100.0)
G02	Circular interpolation clockwise (e.g. G03 X100.0 Y100.0 I0.0 J0.0)
G03	Circular interpolation anti-clockwise (e.g. G03 X100.0 Y100.0 I0.0 J0.0)
I	Absolute X value for arc centre. The value for I must be in absolute form if the X and Y co-ordinates are in absolute form
J	Absolute Y value for arc centre. The value for J must be in absolute form if the X and Y co-ordinates are in absolute form
Nx	Block number where x is a numerical value. Not required but may be used.
M12	Start of shape
M0	End of shape
R	Radius parameter in lieu of arc centre co-ordinates (I and J codes)
M62	Machine Pause
T0#	Tool command where # may be a value from 1 to 8.

NC Files and Tool Mapping

Any programs that can create NC files through a POST processor allow you to automate much of the setup time in Toolpath associated with cutting the job. Programs such as MasterCAM, ARTcam, Cabinet Builder and KitchenCAD Pro can be used to allocate a physical tool on the machine to a cut path in your job file. This gives the program designer greatest control of the cutting of the job by choosing which tool is to be used on what part of the job.

The process of making sure that the tools in your CAD/CAM program library match the tools in the Toolpath software library and the tools in the Automatic Tool Changer is known as **Mapping**.

The mapping of tools is usually performed during installation of your ToolPath software by the Multicam technician. It is assumed that the Tool Library has already been set up in your CAD/CAM program.

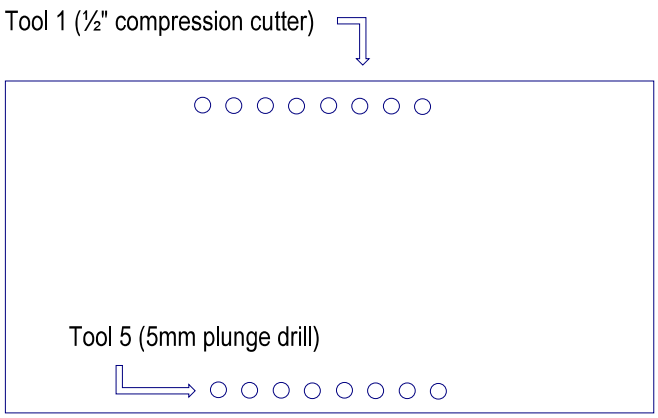
The use of a Tool Mapping Table similar to that shown below will assist your programmers and machine operators and ensure the accurate flow of information.

CAD Program Tool Number	Toolpath Group	Tool Changer Tool Number- can be in any order	Tool Description
1	White	1	1/2" Compression Cutter
2	Blue	2	1/4" Slot Drill
3	Green	3	1/8" Downcut
4	Yellow	4	5mm Plunge Drill
5	Red	5	V Cutter
6	Magenta	6	Profile Cutter
7	Cyan	7	1/2" Roughing Cutter
8	Gray	8	1/4" Ball Nose Cutter

Generally speaking your CAD\CAM software will decide which tool to use for each cutting path based upon preferences set in that program. When correctly mapped the NC file will pass from your CAD\CAM program to the machine with minimal input by the machine operator.

It is not necessary to have your physical tool order at the machine in the same order as your Toolpath Groups.

When this job is imported in to Toolpath you will notice that in the OUTPUT screen the WHITE and YELLOW groups are active as per the mapping table above. Provided that you have a compression cutter loaded in to Tool 1 position and a 5mm drill in the Tool 5 position the machine will select these tools as required to cut the job.

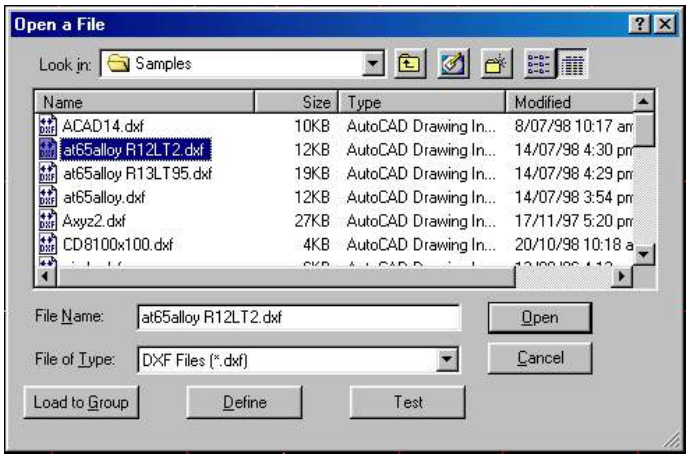


Refer to information commencing on page 95 about configuring tooling in Toolpath and setting up Toolpath when using Automatic Tool Changers.

If no tool numbers are called in your NC file then all cut paths will be imported in to Toolpath in the WHITE group.

Importing a file into ToolPath

To import a file select the *file type* you wish to import by clicking on the down arrow next to **File of Type** and choose either DXF, PLT, NC, PCD or RDY. A list of matching files will now be displayed. Click on the file you wish to import and then click on **Open**. You could also double click on the required file to open it.



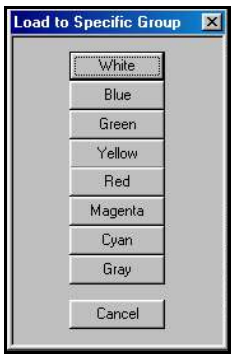
The file will be loaded and then displayed in the main Toolpath window.

If you start ToolPath and this dialogue box appears it means that you have more than one machine connected to your computer or Toolpath cannot establish a connection with your machine. Select your machine if it is on line (turned on and communicating with ToolPath). Click on the machine name and then click on **OK**. If you click on **Default** ToolPath will run with its default settings, which may not necessarily be the same as those of your machine.

Your design will now appear in ToolPath and you can begin working with it before sending it to the machine.

Load to Group

Clicking on **Load to Group** will load the selected job into ToolPath into any one of the eight Groups available. By default jobs are loaded into the White group. This means that you can only load a single job at a time. Load to Group allows you to select a group colour. The file will then be imported into that group. Simply pick one of the colours from the dialogue and the file will be imported into that group.



Note: All layer information in the current file will be lost when you specify to load the file to a single group. To preserve initial layer structure, open the file without Load to Group.

There are a couple of reasons why you would choose to use Load to Group:

1. If your job is to be run using a particular tool that has been setup in one group, you can direct the entire file to that group rather than moving shapes after the file has loaded.
2. The operator can use Load to Group to load more than one job at a time into ToolPath. For example, you may design a job with a border and text, but you may wish to do the Architectural border in a program such as AutoCAD and the text in CorelDraw. This way, by using a common reference point, you can import both jobs into ToolPath from separate applications, and run them together as a single job.

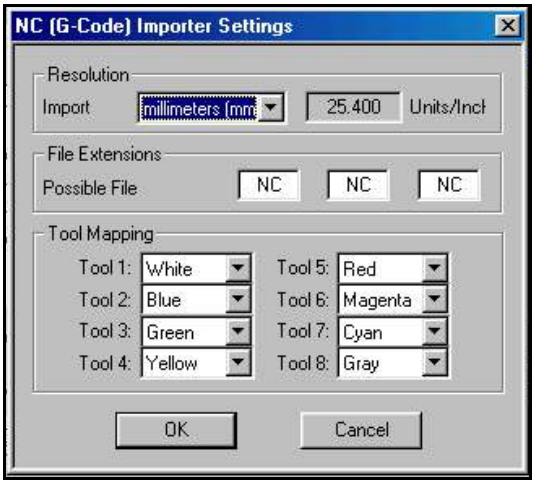
Refer to page 85 for more information on Groups.

Define

Defining File Types allows you to customize the import settings for the currently selected File Type. The ability to edit the import settings is important if you are moving between Metric and Imperial measurement systems, opening files from different customers that use different measurement systems, being able to reconfigure your HPGL configuration according to the software used to generate the plot, or to assign tools to match the tool assignments from G-Code files. It simply gives you greater flexibility to the variety of file requirements that have been developed in recent times.

For each of the Importer Filters (DXF, PLT, NC, PCD, and RDY) you can define Resolution (the units in which the files are imported), File Extensions (valid extensions for this file type) and Tool Mapping (which tool in the file is assigned to which group).

Some of these parameters may not be able to be defined for some file types.



If you use a design package that allows you to design in millimetres but will only export files in inches, you can set the ToolPath import filter Resolution to inches. ToolPath will then display the job in millimetres once it has been loaded.

HPGL (PLT) Import Settings

It is important to be able to calibrate the HPGL settings within ToolPath because there is more than one standard for the number of Units/Inch.

Resolution:
True HPGL standard is 1016 units/inch, but many software design packages will use only 1000 units/inch. This would result in a job that would end up cutting smaller that it was designed for if imported at the 1016 HPGL standard.

File extensions:
Here you can associate up to three different file extensions with HPGL files. Some design packages may default HPGL plots with extensions other than PLT, so you can synchronize ToolPath's HPGL importer to those file extensions.

Tool Mapping:
The final enabled dialog section allows you to co-ordinate pen colours of the original HPGL plot to separate Groups within ToolPath. The cooperation between the designer and the ToolPath operator will save much of the time required to separate the imported job into the individual groups required for any particular job.

G-Code Importer Settings

Resolution:
As with the DXF importer, you can select the import measurement system, or set custom units per inch setting.

File extensions:
You can associate three file extensions with G-Code files that ToolPath will recognize.

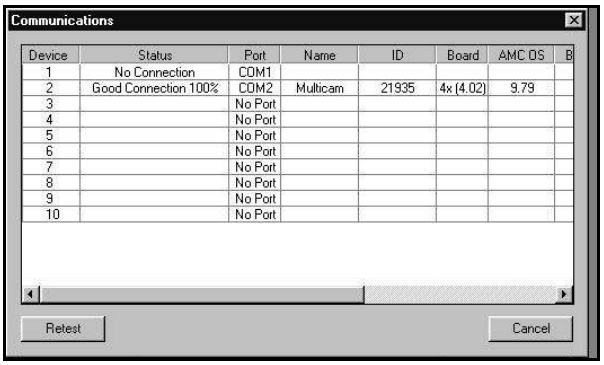
Tool Mapping:
Co-ordinate layer colours and tool assignments of the original G-Code plot to separate Groups within ToolPath.

Refer to page 101 for more information Tool Mapping and configuring import filters.

Test

Use the **Test** button to check the communication between your computer and your Multicam machine. From the Menu Bar select either **Communication**, **Reboot Machine**, **Main Menu** or **Help**.

Communication
Click on communication to check the communication to the machine. The following diagnostics screen will appear showing the results of the Communications test.



RS485 Communications Test Screen

If No Connection appears, then the computer and your machine are not communicating. This can signify several situations:

1. The data cable may be unplugged at either end. Check the connection at the computer and at the machine.
2. The machine power may not be on. Check that there is power connected to your machine .
3. If power is on at the machine, check that power is getting to the AMC board. The left hand most LED light in the upper right hand corner of the board should be lit. If there are no lit LEDs, then check you the fuses.

If the status reads Bad Connection, move the data cable to the other LRC port (there are two on every card). A Bad Connection is usually caused by a flaky connection in the communication system. If the problem persists, check the continuity of the data cables and check the cable is properly seated at both ends. Also, reset the LRC card in the computer to ensure that it is properly seated in you system.

Reboot Machine
Clicking on **Reboot Machine** will retransmit the machine operating system to the Multicam machine. This will overwrite the existing operating system only, it will not delete any jobs that may happen to be resident at the machine.


You must not perform communications tests or reboot your machine whilst you have jobs running. Doing so may damage your machine and any jobs you are running!

Main Menu
Clicking **Main Menu** will exit you from the Communications screen and return you to the ToolPath Main screen.

EDIT

The **EDIT** function allows for the modification of some parameters of your job, including erasing shapes, sequencing the cuts, changing start points, etc. When you select **EDIT,** the main menu will be replaced by the edit menu:

You will notice that one of the shapes on the screen will be highlighted in a brighter colour than the remaining shapes. This is referred to as the "selected" shape. The number in the top left corner of this menu indicates the shape number. You can select any shape by toggling through them in sequence using the **Next** and **Previous** commands or simply pick the desired shape by pointing to its outline using the cursor and clicking on it. You can also use the keyboard shortcuts **N** and **P** for next and previous shapes.

A small yellow “icon” will appear at the start point of the selected shape. The “icon” also indicates the direction of the cut, as well as the side of the cut that the compensation of the end mill will be on.

Zoom

Commands in the **Zoom** menu allow you to get a closer look at your job. You can use the various commands to inspect shapes, or to get an overall view of your job. By selecting **Zoom**, the **EDIT** menu is replaced with the **Zoom** menu. Your options in this menu are as follows:

You can pick an area to **Zoom** in on by clicking the left mouse button once and dragging a “bounding box” around the area you wish to zoom in on. Releasing the mouse button will cause the selected area to be zoomed in on.

By selecting;

All you will zoom the shapes and material out to display everything, including the full size of the design, or the full size of the material, depending on which is larger.

Extents will zoom the picture to the size of all the shapes in the job.

Shape will zoom the selected shape to fill the screen.

Material will zoom the screen to the limits of the material, as defined in the layout.

Left, Right, Up, Down are used to pan around the screen

3D View will allow you to enter angles for the X and Z axes thereby tilting and rotating the job on the screen. This is useful for viewing 3D NC (G Code files) to verify that there are no apparent errors in the Z axes of the job. Refer to page 53 for more detailed information on how to use the 3D View function.

You can also use the following keys on the keyboard to execute the zoom commands;

A	All	**R**	Right
S	Shape	**D**	Down
W	Window	**U**	Up
L	Left	**ESC**	Exit

Reverse

The direction of cut can be reversed by selecting the desired shape and choosing **Reverse** or pressing the **R** key on the keyboard. On open shapes reverse also changes the start point from one end of the shape to the other.

Erase

A selected shape can be erased by selecting **Erase** from the edit menu. This can be useful when used in conjunction with the **Move** command if you need to re-machine some parts of a job, but not others.

To erase an item, select **Erase**, then "pick" a drag point on the selected shape. The selected shape will turn White. Clicking on **Erase Shapes**, or pressing the **E** key on the keyboard will delete the selected shape from the screen. You can also click your right mouse button to erase selected shapes.

If, after selecting a shape, but before deleting it, you change your mind, select **Undo 1**, and the last selected shape will be deselected.

To erase multiple items, you should click and hold your left mouse button in an open area, beyond the extents of what you want to erase and as you drag your cursor, you will see a white box open up. Only the shapes that are completely enclosed by the white box will be picked. You can otherwise select individual shapes by clicking on the ones you want. You can then click on **Erase Shapes**, and the selected shapes will be erased. You can also click your right mouse button to erase selected shapes.

Undo 1 works the same way for multiple shapes as it does with individual shapes. If you used the box to select your shapes, the last shape from the right to the left on the screen will be deselected. If you picked individual shapes, they will be deselected in the reverse order to which you picked them.

You can get back to the main menu any time by pressing "**Esc**" on your keyboard, or clicking on **Main Menu** with the left mouse button.

Sequencing Shapes Within a Job

The **EDIT** menu also provides a sequencing function to control the actual sequence of the shapes to be cut within a job. This is a very useful, but simple function that allows you to optimise the machining process and so speed up production.

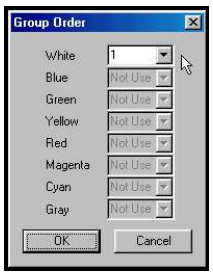
Select **Sequence** from the edit menu. Now select the shapes using the cursor in the order you want them to cut. The shapes will be highlighted as they are selected. If you make a mistake, select **Undo** and the last selected shape will be deselected. Once you have set the sequence select **Forward**.

It is not necessary to select all the shapes. Any shapes not selected will cut in random order after the selected shapes. There is also a **Reverse** command. This will cause the first selected shape to cut last, the second shape second last, etc. Shapes that were not selected will cut before selected ones.

If you wish to select more than one shape at a time click and hold down the left mouse button in a blank area of the screen and drag a box to surround the desired shapes then release left mouse button. All the shapes **completely** enclosed by the window box will be selected.

This is a useful function if you are in a situation where you need to clamp your material from one side of your machine only. Select the shapes along the edge where the clamps are going to be located. If you **Reverse Sequence** these pieces, they will be the last ones to be cut, ensuring your material will stay in place.

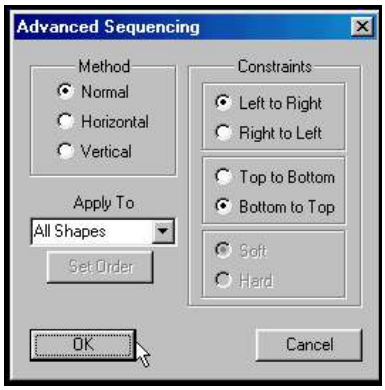
You can also sequence by groups. If you choose this option a menu will appear where you can select number the groups 1 through 8 in the order you wish to cut. This is advantageous if you wish to cut different shapes at different depths. If you want to engrave some shapes at a certain depth, then cut out around the entire shape, you can set the engraved shapes to a particular group (e.g. White) and set the cut depth to your desired amount, then set the final cut out to another group with it's cut depth all the way through the material. Sequencing by group is also useful if you are using multiple tools within the one job and wish to minimise the number of tool change operations.



The final option for Sequence is to choose **Advanced**. This option allows you to sequence your job based upon criteria you select in the dialogue box.

You can choose to sequence horizontally, vertically, left to right, right to left etc. You can even have ToolPath apply the desired sequencing to the entire job, or if your job contains groups, to specific group colours.

If you make a mistake pressing the **U** key on the keyboard will undo your last command.



Altering Start (Plunge) Points

The start point is the point where a shape begins and is represented on screen by a yellow tag. The start point of any shape can be moved by selecting the desired shape then choosing **Start** from the edit menu. A new start point can be designated by pointing to the desired start point and clicking on it. You can also increment through the shape vertices using the **Next** and **Previous** commands. You may wish to alter the start points in order to avoid conflict with other shapes.

The direction of an open (green) shape can only be altered by selecting **reverse**.

Outline

This will outline a selected shape with a line at the distance you specify. It allows you to assign a specific outline or inline to a shape. If the object appears blue on the screen, it will be an outline. If the object appears red on the screen, it will be an inline. You must click on OUTLINE and type in a numeric value and press ENTER. To create an OUTLINE on a red shape you must enter a negative value.

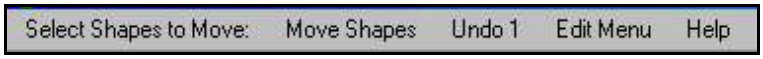
Pause

This command allows you to select a shape for the machine to pause on. The machine, when seeing a PAUSE POINT, will automatically lift the head and pause with a message on the LCD Display **"Pause On Shape #"**, you can now change the tool, re-do depth and continue cutting, The START button on the Sub console must be pressed before the job will continue. The PAUSE command is useful for single head machines that may require a tool change during a job or on automatic tool change machines that also require the use of manually inserted tools. More detailed instruction on the use of pause points can be obtained from our technicians.

After selecting **Pause**, and clicking on a shape a small pause point icon ⊠ will appear at the start point of that shape. If you click on the shape again you can move the pause point anywhere you want. The pause will always occur just before the selected shape.

Move an Entity or Shape

This feature allows you to move shapes around within a job. It is very useful if you find the tool bit compensation is causing some overlapping of parts.



To move an item, select **Move**, click on the shape to move. The selected shape will turn White. Clicking on your right mouse button will bring up a white box that represents the extents of the selected shape. As you move the cursor the shape's frame will move with the cursor, "drop" the shape by picking a drop position and clicking your left mouse button.

To move multiple items, you should click your cursor in an open area, beyond the extents of what you want to move. As you drag your cursor, you will see a white box open up. Only the shapes that are completely enclosed by the white box will be picked. By clicking your left mouse button again, all shapes that were completely enclosed by the box will be selected. You can otherwise select individual shapes by clicking on the ones you want. Again, clicking your right mouse button will bring up the extents box for the shapes you have selected. You can then move the box to where you want the shapes, and click the left mouse button to drop the shapes.

When shapes are moved, if your material size has not been set, ToolPath will display a message then adjust the material size to the new extents. If the material size has been set, it will stay at the selected size even if a shape is moved over the edge of the material.

This function is useful because it will allow you to nest shapes manually to make better use of your material, or to fill spaces left when you delete a shape. You can get back to the Main Menu any time by pressing "**Esc**" on your keyboard, or clicking on **Edit Menu** with the left mouse button..

VIEW

Commands in the **View** menu allow you to get a closer look at shapes in your job. You can use the various commands to inspect shapes, or to get an overall view of the job. By selecting **View**, the **Main Menu** is replaced with the **View** menu. Your options in this menu are as follows:

You can pick an area in your job to **View** in on by clicking the left mouse button once and dragging a "bounding box" around the area you wish to zoom in on. Releasing the mouse button will cause the selected area to be zoomed in on.

By selecting;

All you will zoom the shapes and material out to display everything, including the full size of the design, or the full size of the material, depending on which is larger.

Extents will zoom the picture to the size of all the shapes in the file.

Shape will zoom the selected shape to fill the screen.

Material will zoom the screen to the limits of the material, as defined in the layout.

Left, Right, Up, Down are used to pan around the screen

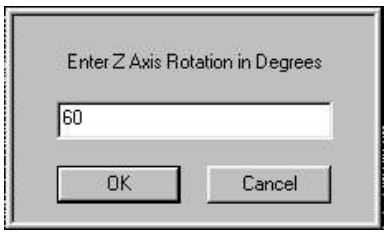
3D View will allow you to enter angles of rotation for the X and Z axes by selecting the required axis from the menu bar.

This allows you to tilt and rotate the job on the screen which is useful for viewing 3D NC (G Code files) to verify that there are no apparent errors in the Z axes of the job.

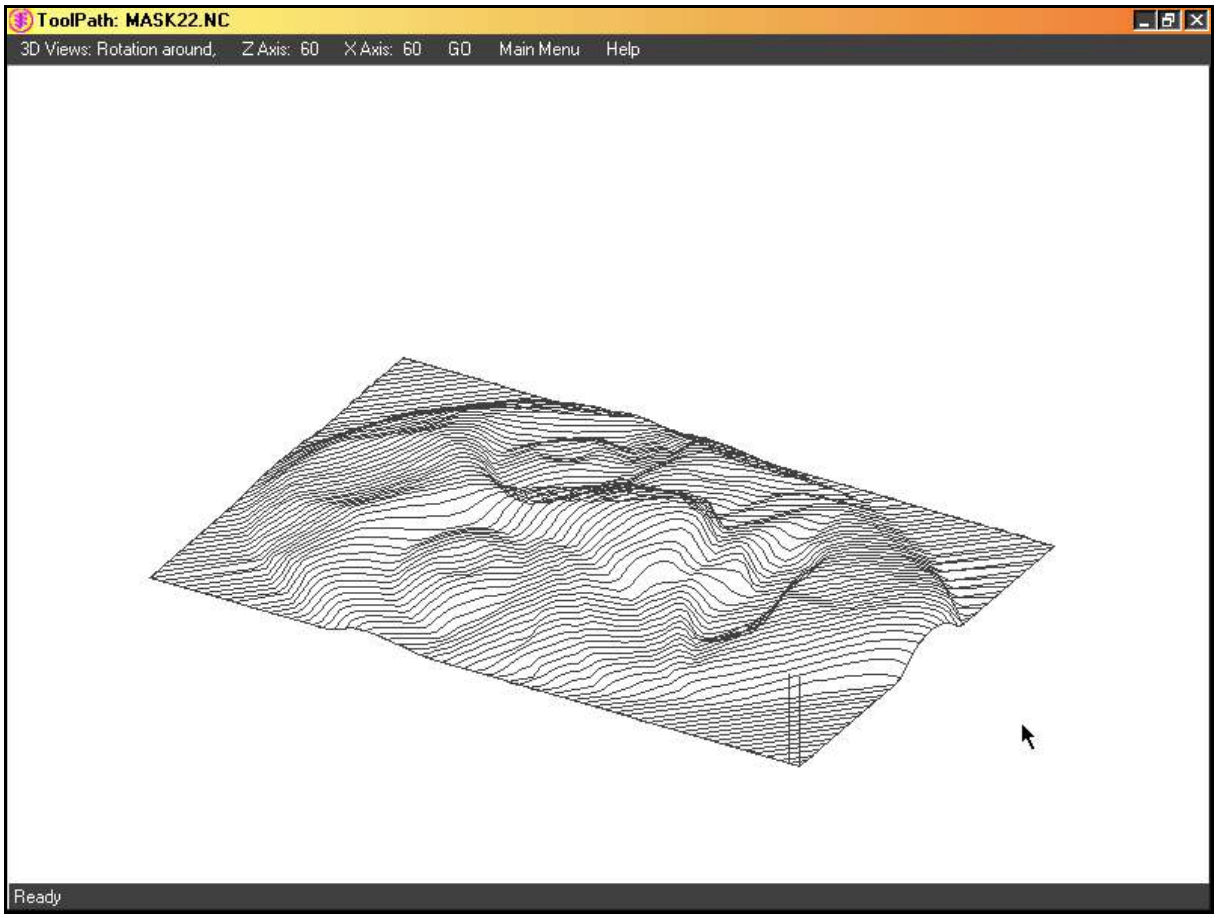
Click on the axis that you wish to rotate and enter the rotation angle in the dialogue box, then click **OK** to close accept the value. Do the same for the other axis if required.

Once you have entered your rotation angles click on **GO** on the menu bar to have your job rotated on the screen. If you are not happy with the resulting view you can re-enter either of the rotation angles.

Note: The values for the rotation angles are measured from absolute points. They are not cumulative.

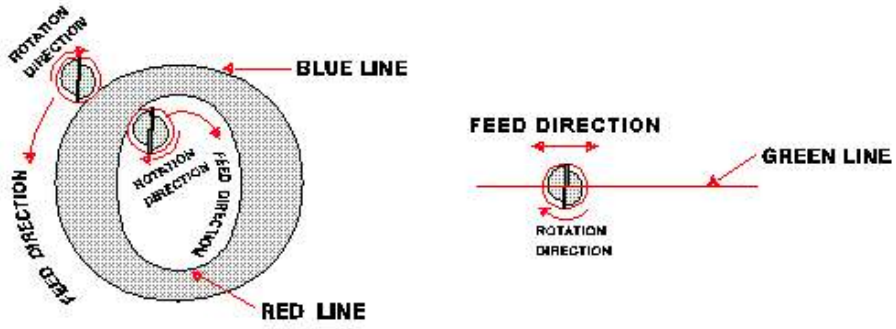


The following page shows a screen shot of a typical 3D NC file.


ToolPath: MASK22.NC
3D Views: Rotation around,
Z Axis: 60
X Axis: 60
GO
Main Menu
Help
Ready
3D View

ANALYSIS

There are certain conventions that are important to know when creating tool paths for CNC Routers. The Analysis menu will identify and take care of the following issues.



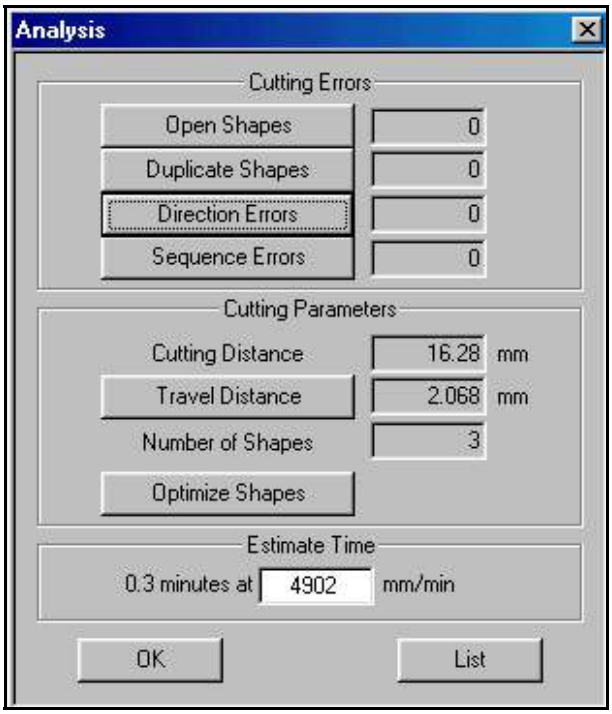
- Normally all cutting paths should be closed
    - ToolPath identifies those paths that are not closed, and offers the ability to close open shapes that have a short distance between the start point and the end point.
- Duplicate Shapes (two identical shapes superimposed on each other) will cause havoc when doing 2D and 3D Engraving. This occurs sometimes in stock clip art, and can be corrected by ToolPath by automatically erasing the excess multiple copies until only one is left.

  Hint: Over the counter clip art has been known to be peppered with duplicate shapes. Always make sure you eliminate duplicates if you plan to engrave, or to simply route out these designs.
- Inside shapes must be cut before the corresponding outside shapes. For example the inside part of an "O" must be cut before the outside part otherwise there will be nothing holding the material together as it cuts the inside
    - By correcting the sequence errors in a job, you will eliminate any possibility of outside shapes cutting before inside shapes.
- Inside shapes usually are cut in a clockwise direction, outside shapes in a counter clockwise direction. There are two reasons for this:
    - The radius of the router bit must be compensated for.
    - Compensation is calculated to the right of the cut.

  This means that positive compensation values will give correct cutouts provided the shape directions are correct. Use the Direction Errors correction in ToolPath to ensure proper direction of cut.

  Router bits turn clockwise which means the material to the left of the bit is being "front cut" and the material to the right is "back cut". It is normally preferable to front cut the "good" edge.

The **Analysis** command provides a quick and simple way to ensure that the generated tool path will cut correctly. When the job is correctly set up all the inside shapes will appear **Red** to indicate that they are cutting clockwise. All the outside shapes are indicated in **Blue** and will cut counter clockwise. **Green** shapes (open lines) will be cut on the side of the line indicated by the yellow icon at the start point of the shape.



ToolPath will automatically correct direction, sequence and duplicate shape errors when you click the corresponding buttons in the dialogue box. You should always ensure that there are no direction or sequence errors in the job before transmitting the job to the machine.

Open Shapes

The Analysis dialogue provides a quick and simple way of ensuring the processed tool path will cut out properly at the machine. This will reduce waste of both time and material when setting up jobs.

Open shapes are represented on the Toolpath screen by green lines. The number refers to the total number of open shapes (green shapes) in your job. Ideally any unwanted open shapes should be corrected in your design software before proceeding. However you can use this command to correct some open shape errors in Toolpath. The command acts globally within the job.

Clicking on this button brings up a dialogue prompting the user to enter a search radius. Once entered, ToolPath will attempt to close all shapes whose endpoints are close enough together to fit into a circle with the given radius.

Note: Shapes whose endpoints cannot fit into the given radius will remain open shapes. If a shape remains open or is closed with an unwanted line, then use your design software to ensure the shapes are closed.

Duplicate Shapes

The number of duplicate (excess) shapes is shown to the right of the button. Clicking on this button will remove duplicate shapes. If it is intended to have duplicate shapes in your job, ensure that the shapes exist on separate groups. For instance, you may want to cut a shape at half depth with a wide drill bit and then cut the same path again at a deeper depth with a smaller drill bit. Since each tool requires it's own group, the duplicate shapes are essentially correct.
Note: If duplicate shapes are found even though they exist in different groups, they will be removed from the job. If it is the intention to keep duplicate shapes in separate groups, do not use this feature. Manually ensure that the job is correct with regards to duplicate shapes.

Direction Errors

Clicking on this button will correct both Direction and Sequence errors. This will ensure that inside shapes are processed first at the machine and then outside shapes will follow.

Sequence Errors

Clicking on this button will eliminate sequence errors. All inside shapes will be cut first.

Cutting Parameters

These parameters are informational values for you. The intention of the values is to give you an idea of the jobs cutting time and process distance. This will allow you to rationalize tool usage and cutting performance.

Cutting Distance

This is a status field that indicates the total amount of cutting expected at the machine.

Travel Distance

The status field at the right of this button indicates the total amount of distance the tool travels between cut out shapes. This is the distance where the tool must lift out of the material and move to the next shape. The lift and plunge distances are not included in this total, only the difference of travel between one shape's endpoint to the next shape's start point in the sequence.

Clicking on Travel Distance will activate the Sequencing menu, which will allow you to optimize the sequence of the shapes in your job.

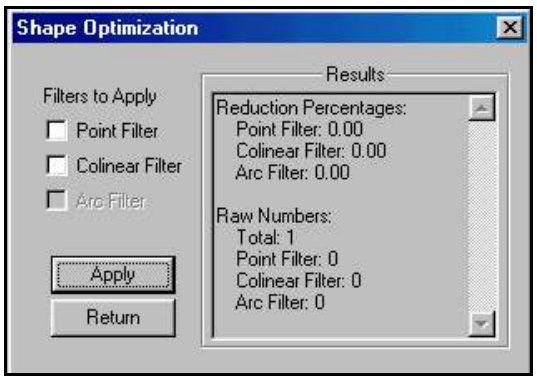
While the machine is travelling between shapes, it will move at the Maximum Speed. Thus, positioning is generally faster than cutting. This minimizes processing time, but it is desirable to optimize the shape sequencing in order to minimize travel time. Refer to page 51 for information on sequencing shapes.

Number of Shapes

This status field indicates the total number of shapes in the job. All shapes, drill points, and pause points are included in this total.

Note: A good indication if an error has occurred in the job is if the number of shapes is much higher or lower than the expected number of shapes.

Optimize Shapes



Shape optimization is used to help reduce the number of vertices (or control points) in you job. This is particularly useful when importing three-dimensional shapes from machining programs. Some design packages can produce unnecessary points.

In brief, the development of a "shape optimization" routine was instigated to help reduce the size of certain jobs being downloaded to a machine. For instance, three points are used to define two consecutive line segments. The middle point can be removed if the lines are co-linear (the angle between the two lines is 180 degrees). The resulting line segment does not alter the job's cutting path. This can reduce job size without altering the job itself.

Two filters have been developed to facilitate shape optimization.

Point Filter
The Point filter removes points that are closer together than the resolution of ToolPath. This resolution is known as the minimum line length, and it is the minimum distance that is required to uniquely identify a line segment. If you design package generates drawings of a greater resolution, these extra points are removed by ToolPath because they are considered extraneous.

Co-Linear Filter
The co-linear line filter, as described earlier removes extra points that need not describe co-linear lines. The tolerance of this filter is set in the TPWin.axy file to ensure that it is not intentionally or accidentally changed from within ToolPath. If the tolerance is set too high, your shapes will come out distorted as many line segments will become one line segments. The current tolerance is by default 0.001 inches.

Arc Filter
A third form of filter (not currently developed) removes points unnecessary in defining a 3D arc. A 3D arc is defined by a centre point, a start point, an end point (or angle), and a radius. Defining an arc by individual line segments is both wasteful and imprecise. Eventually, ToolPath will automatically find arcs and define them using the minimum amount of information required.

Estimated Time

ToolPath can estimate the amount of time your job will take to process. This value is calculated by taking into account the processing speed, cutting distance, and travel distance of your job. Lift and plunge distances, multipass settings and tool changes are not taken into consideration in the estimated time. The processing speed can be set in the edit box contained in the Estimated Time section of the dialogue. By entering the same processing speed you expect to process the job with, you can get a fairly accurate indication of job cutting time.

Note: Job time will be shortened if time is taken to optimize the sequencing of the job and minimize travel time. A few minutes extra at the computer will save you time and money in your plant. At the machine, minimizing the lift height of the tool will also save on processing time as the Z lift is slower than X or Y travel.

List

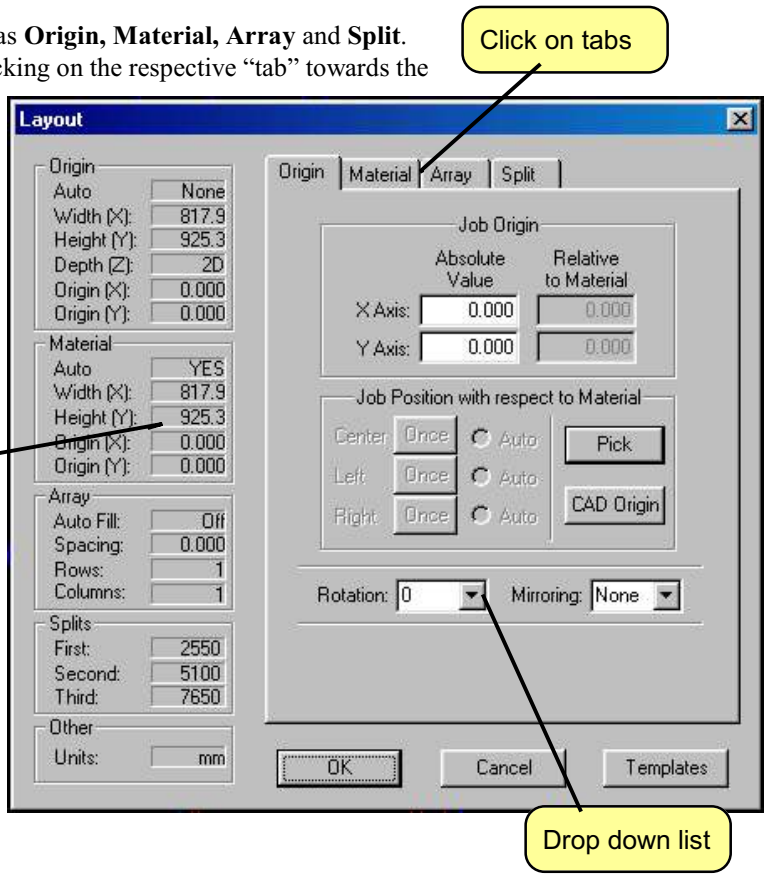
Pressing the **LIST** button will generate a text file called *LIST.SHP* in the last directory folder that ToolPath accessed. This file contains information about job on the screen such as number of shapes, minimum and maximum extents, group information and machine setup information.

LAYOUT

The **Layout** feature provides functions to position the job on the material, rotate or mirror it, split the job if it is too large for your table, or array the job if multiple copies are required. After selecting **Layout** the following dialogue will appear:

The dialogue is broken into four main areas **Origin, Material, Array** and **Split**. You can move between these areas by clicking on the respective "tab" towards the top of the dialogue (as indicated on the screen shot) or by clicking in one of the fields under the area you want to move to on the left side of the dialogue.

You will notice the use of drop down lists to select values for things such as Rotation and Mirroring.



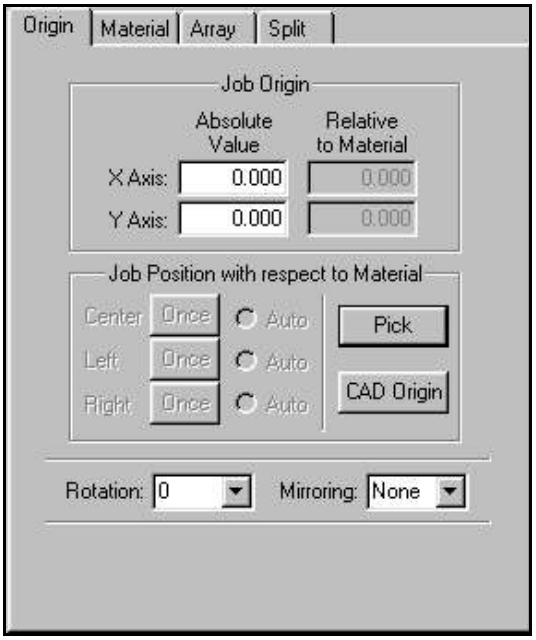
You can save *Templates* in Layout for commonly used settings such as Material Size and Job Origins. This will help those that use a number of different layout settings on a regular basis.

Note that the Templates in Layout are in no way related to the Templates in Groups.

The job width and height are determined by the file that is imported. An important thing to remember is that the units of measurement are set in the **SETUP** menu. **ToolPath** defaults to MM. If you want to change this to CM or INCHES, refer to the section on **SETUP.** It is vital that the values that are exported from your design program are the same as those you are working with in **ToolPath**.

Origin

Job origin is the positioning of a job or layout on your material. Some of the options on the Origin tab will not be available until information about your material has been entered on the Material tab. Specifically the **Centre, Left** and **Right** buttons will not work until the **Auto-adjust material to fix job extents** check box is *unchecked* on the Material tab and a size for material is entered.



Job Origin is used in conjunction with the material size to set the exact placement of the start of you job. By default, the Job Origin is set to the lower left corner of the material, or the (0,0) point. The origin point is designated by a small white cross at the (0,0) point. This is also known as the Left Origin. You can also set the starting point to any position on the material.

By setting the Job Origin to the upper right corner of you machine, you can start and finish your job with the tool head moving out of the way of your material loading. If you change the origin, the white cross in ToolPath remains at (0,0), but your shape offsets have changed to reflect the new origin position.

Note: Be aware that if you change the origin in ToolPath from (0,0) to another point, Function 16 at the sub-console will become disabled. Auto Home will still function, which will return the tool head to the machines sensors.

You can also align you job in respect to the material size. Centre, Left and Right are useful alignment features especially if you are using pre-cut blank stock and wish to line up your jobs shapes and text with respect to the material. You may notice that the Centre, Left and Right alignment buttons are disabled upon opening the Layout Dialogue. To enable these features, click on the Material tab and ensure the "Auto-adjust material to fit job extents" option is turned off (i.e. no check mark in the box). Returning to the Origin tab will show enabled alignment buttons.

> **Centre** justify will position your job into the geometric centre of your material.
> **Left** justify will position the job to the lower left corner of the material. This is the default setting.
> **Right** justify will position to the lower right corner.

*If you find that you are always using the same justification you can set this justification as a default for ToolPath by clicking the button labelled **Auto** next to the required justification value.*

Or you can specify alternate origins by clicking the **Pick** or **CAD Origin** buttons;

> **Pick** You can pick your own Job Origin by "eye-balling" a position on the job screen. By clicking on Pick, the layout dialog will disappear allowing you to "pick" a point in your job to set the origin point. After clicking somewhere on the screen, the Layout Dialogue will appear again with the adjusted point.
>
> **CAD Origin** will position the (0,0) coordinate from the original CAD drawing to the lower left corner of the material. The job itself will then be placed on the material relative to the CAD origin. Note that not all design packages export a CAD origin. This is a convenient method of setting up your jobs for use with a jigged clamping or vacuum hold down system.

The machine itself will always start at the lower left corner of the material (the lower left of the yellow box indicated on the screen).

Rotation
You can rotate the job in 90-degree increments from the lower left corner of the job extents. Select one of the 4 rotation values from the Rotation drop-down list.

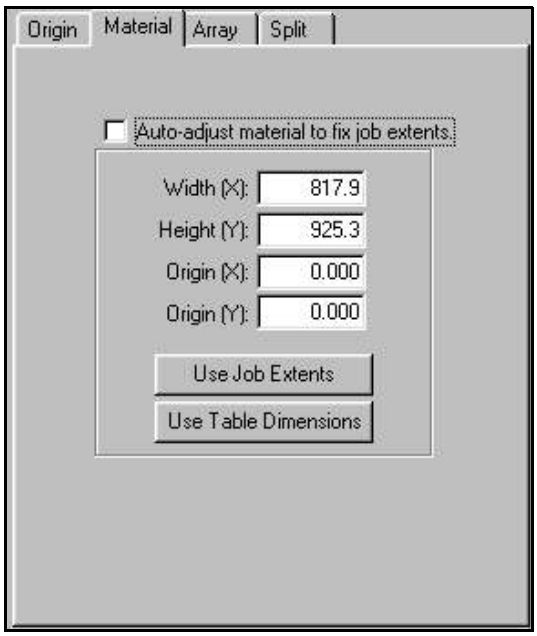
Mirroring
You can mirror your job about the X (horizontal) or Y (vertical) axes.

Note: Mirroring is useful since it allows you to cut jobs from the back side of the material if you are using a laminate that you do not want to pull apart, or if you want to engrave in the back side of a clear material such as acrylic.

If you use location brackets on your machine to locate sheets of material when loading you may want to set an offset amount in th X and Y axes to prevent the cutter from hitting the corner brackets. The amount you enter for the X axis and Y axis absolute values must be greater than the diameter of the largest cutter that you intend to use. Typically 20mm offset is a good safe amount.

Material

The material tab is used to supply information about the size of the material from which you will be cutting your job.



The material size is represented in ToolPath as a dashed yellow line. This shows the material extents with respect to your job.

You can have ToolPath automatically set your material size to suit your job by clicking the check box next to **Auto-adjust material to fix job extents**. This will result in the material size being defined as the same as the size of your job.

To define a set material size, make sure that **Auto-adjust material to fix job extents** is unchecked. You can then type in the size of your material. You can also type in the X and Y Origins of the job in relation to the lower left corner of your material if you so wish.

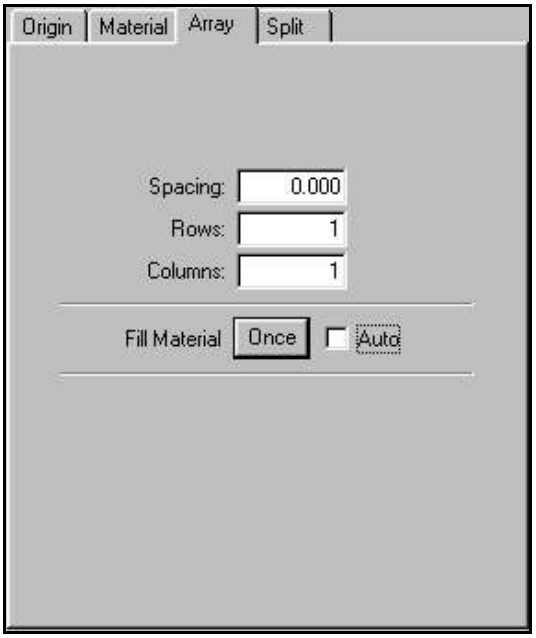
Clicking on the Use Job Extents button will result in the material size being defined as the same as the size of your job.

To set your material size as the same size as your table click on the Use Table Dimensions button.

Note: If you rotate your job, the material size will automatically reset itself to the rotated job extents if the dimensions change.

Array

The array feature will repeat the job in designated rows and columns. The number of rows and columns may be selected or you can have ToolPath automatically fit as many repeats on to the material as possible by checking the **Auto** check box or clicking on the Once button.



The **Spacing** command determines the space left between the rows and columns. This should normally be set to three times the bit diameter you are using.

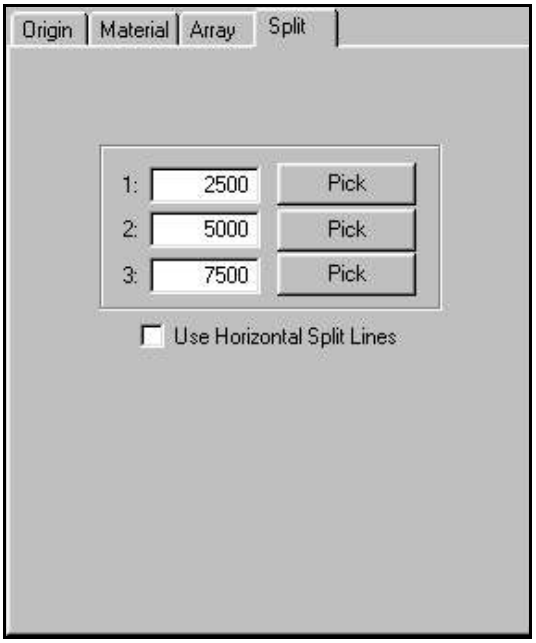
If you rotate the job, only the job copies will rotate, not the rows and columns. Thus a 3x5 grid will remain a 3x5 grid, not rotated to 5x3.

On the screen, Toolpath will display the original item of the array with the remaining member of the array being represented by a white box.

*Note: If you check the **Auto** check box, this will become the default setting for ToolPath until you un-check it, even if you exit and then re-enter ToolPath*

Split

Sometimes a job is too large for your machine. If this is the case, you can have ToolPath split the job up into manageable pieces. Three split positions may be entered to divide your job up into a maximum of four parts. The split positions 1 through 3 can be simply entered as desired or can be "picked" by clicking on the **Pick** button. Picking the split position is normally the better way as you can visually position the split lines so as to avoid cutting shapes in half. You can only pick a position to the left of the split line.



The split positions will be indicated on the screen as vertical magenta lines in your design

Once the job is transmitted the machine will cut the split sections in succession stopping after each section. At each split, the Sub Console will then indicate to the operator exactly how far they are to move the material, i.e. **"Page Material 1675.0 MM**. Once the material has been shifted the operator presses **START** and the machine will then cut the next section.

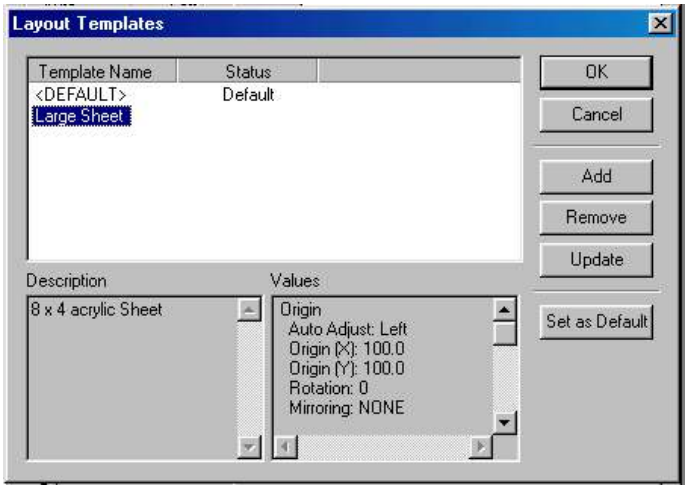
Note: If accuracy is important in your job (i.e. the position of the shapes cut must remain within tolerance) use splits with extreme care. Moving the material after a first section of the job be cut may cause the rest of the job to cut inaccurately. Ensure that the material is moved the exact width as displayed on the sub console, and that the material remains square to the table.

Layout Templates

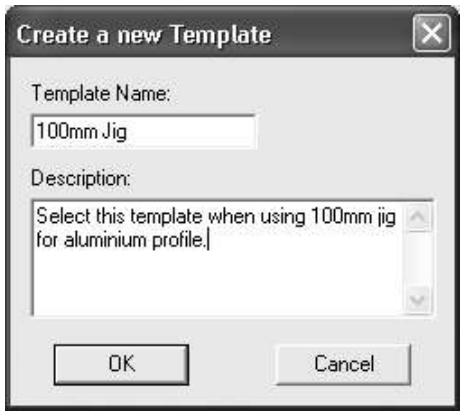
If you find that you frequently use the same settings for material size, job origin, rotation or any other variables in Layout then you can speed up the output process even further by utilising **Templates**.

Templates allow you to save the settings that you have entered in the layout dialogue box and the recall them for later use with a different job.

To create a template click on the **Templates** button and, in the **Layout Templates** dialogue box that appears.



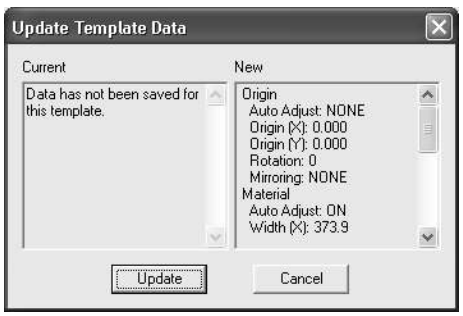
Click on **Add**. Type in a name for the template you wish to create and a meaningful description, then click on **OK**



Your template will now appear under the column *Template Name*. At this point you will note that the *Status* of your template will say "Not Available". This is because the current values set in the **Layout** Dialogue box have not yet been assigned to your newly created template.

To assign the values click once on your template in the templates window so that it is highlighted, then click on **Update**. Your template has now been assigned the values you entered in the **Layout Templates** dialogue box.

To use your template later on with a new job, in the **Layout** dialogue box click on **Templates**, select the template you wish to use, and click on **OK**. The values contained in your template will be assigned to the current job.



Other functions of the **Layout Templates** dialogue box are;

Remove Clicking on this will remove your template from the template list. This process is irreversible, however you will be asked to confirm the action.

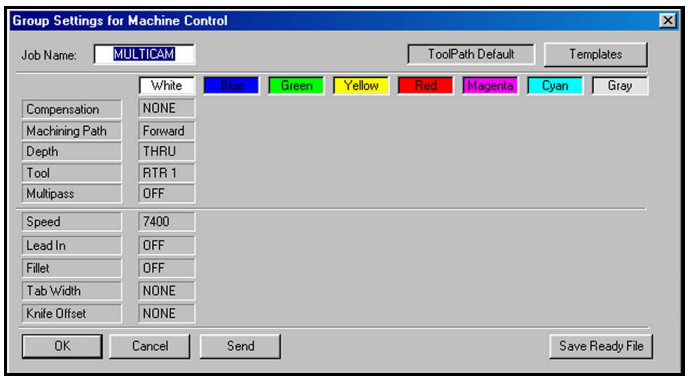
Update Use this to add values to a newly created template or to change the values in an existing template.

Set As Default Clicking this will set the template you select as the *Default* template so that next time you enter ToolPath the **Layout** dialogue box the setting of this template will already be loaded.

OUTPUT

The Output dialogue and the Groups dialogue are the basically the same. The Output dialogue displays only the groups that have shapes within them. The Groups dialogue shows all groups. The Output dialogue is designed to setup the final tooling configuration before sending the job to the table. Nevertheless, both the Groups and Output dialogues are interconnected.

Once **ToolPath** has completed preparing the job, you will download it to the machine using the **Output** command. This command provides the tool path settings to accommodate the tool being used on the machine:



Functions that affect the way the job is cut can all be set in the **Output** dialogue box by clicking on the value on any one of the group fields (i.e Compensation, Machining Path, Depth etc..)

In the above screen shot only one group (white) is shown. This is because the job being sent to the machine only contains a single group. If you assign parts of your job to more than one group then those groups will also be shown in this dialogue box.

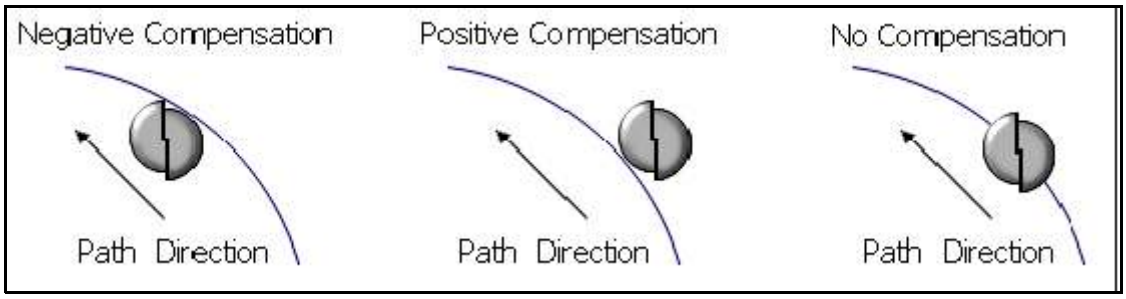
*If you use multiple groups you can change any of the job settings for any of groups in the **Output** menu without having to go back to the **Groups**|**Groups Control** menu.*

Job Name

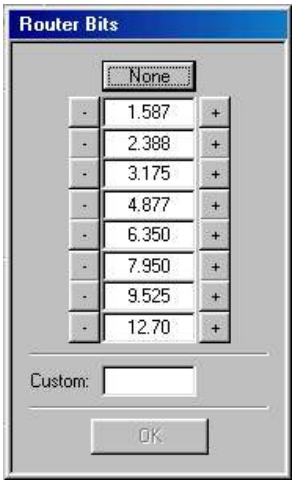
The job name is normally defaulted to the name of the file it was created from. This is the name which will appear on the Sub-Console, and the name of the file if it is saved as a ready file. If a different name is required, select this item and type a new name in its place.

Compensation

Compensation is used to tell the machine what diameter of router bit or end mill will be used to process the job with. You can set one of 10 preset sizes (the most popular bit sizes) or enter a custom bit size. When a job is compensated, the bit is shifted to the left or right of the shape outline so that the edge that is cut lies exactly on the shape outline. If the job is not compensated the bit is centred on the shape outline and you will find the parts cut will be exactly the bit diameter.



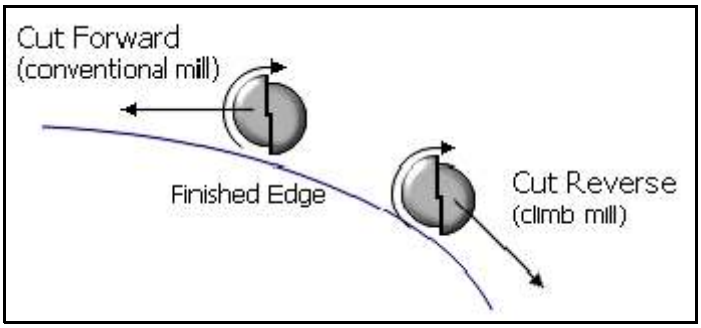
The radius (or half the diameter) of the bit is used as the compensation amount. Select the appropriate compensation by clicking on either the + button next to the required value for positive compensation or the **-** button for negative compensation. You can also enter a custom compensation value by entering the value in the Custom box and clicking the **OK** button



NOTE: *It is also possible to set the compensation at the machine using the sub console's Function 28. Use one or the other* ***not both*** *or the result will be double compensation! Using Function 28 at the sub console is not as accurate for complex shapes.*

Machining Path

The Machine Path parameter is used to set the direction of travel to the group of shapes as the end mill processes the material.



A **Forward** machine path is referred to as "Conventional Milling" in the industry. This means that outside shapes cut Counter-Clockwise and inside shapes cut Clockwise. This is the preferred path direction when cutting wood and most plastics.

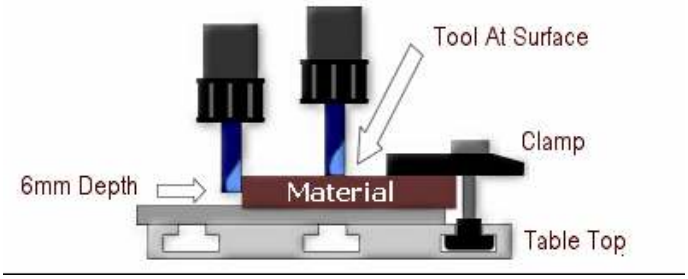
A **Reverse** machine path will process in the opposite direction of the Forward path. This is referred to as "Climb Cutting". Outside shapes will cut Clockwise, inside shapes cut Counter-Clockwise. This is preferred for cutting non-ferrous metals and acrylics.

Setting the Machining Path to **Off** will make the group inactive. When the group is inactive, it still is part of the job, but its paths are hidden. These paths will not be processed at the machine until made active again (by setting the Forward or Reverse machine path).

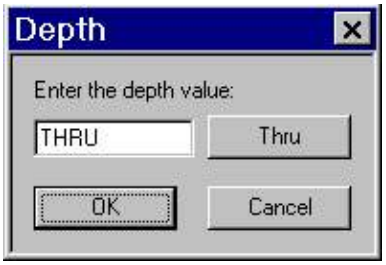
NOTE: *If you attempt to set your machining path to reverse whilst you have multiple tabs set you may encounter irregular machine behaviour where the machining path direction may be overridden. You will be able to see any inconstancies in the output screen prior to transmitting the job to the machine. It is recommended that you do not set your machining path to reverse when using multiple tabs.*

Depth

Setting the depth of cut determines how far into the material the tool will plunge.



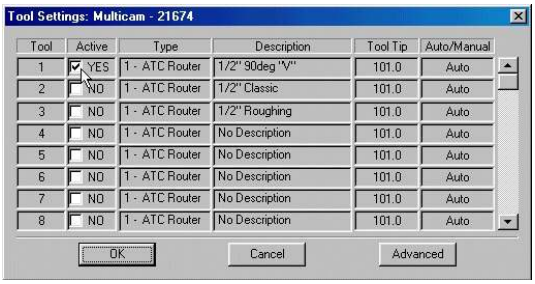
Here you can enter the depth at which the machine should cut the job. **THRU** indicates that the machine will cut right through the job to the bottom of your Function 4 setting. This is the setting mostly used for cutting out shapes. If you do not want to cut right through your material enter the depth to which you want the machine to cut and then click the **OK** button.



To reset your depth to Thru simply click on the **Thru** button or enter the letter **T** for your cutting depth and then click the **OK** button.

NOTE: *For safety it is always preferable to enter the thickness of your material (plus a small cut through allowance of say 0.2mm) in the depth setting field. This will prevent you from plunging in to your machine table top in case you accidentally incorrectly set your Function 4 setting.*

Tool

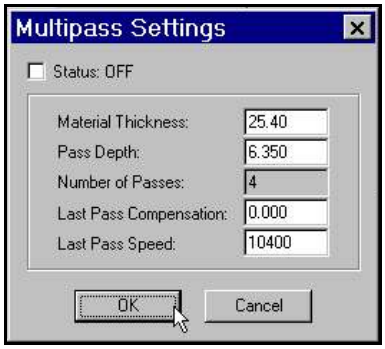


The **Tool Settings** dialogue box allows you to specify which tool is to be used to machine a particular tool path. You can only assign one tool to each group in your job.

You will usually only use the **Tool Settings** dialogue if your machine is fitted with more than one routing head, is fitted with an HSD Quick Change motor, is fitted with an Automatic Tool Changer or if you have other devices attached to your machine such as pneumatically controlled drill heads.

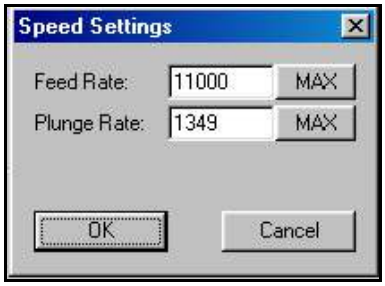
For each head position you can specify a head type (Router, Drill etc...) Unless you have set up multiple tools in the **Setup** menu you will not be able to activate or select different tools for your job. Refer to the section on **Setup|Configure Tooling** on page 95 for instructions on how to do this. For a detailed explanation of multiple tools and their setup and use refer to Appendix F commencing on page 161 and Appendix G commencing on page 165.

Multipass



This feature may be turned "ON" or "OFF" to enable a job to be cut in several passes by clicking the check box labelled **Status**. When status is ON the values you enter for *Material Thickness, Pass Depth etc.*. Will be enabled. Refer to the "Multipass" sub-section in the "**Groups**" section of this manual commencing on page 87 for a closer discussion of all the variables and their effects.

Speed



Enter the Feed Rate at which you want the job to be cut or you may click on **MAX** to automatically select the maximum feed rate for your machine.

Feed Rate is the speed at which the machine will move whilst cutting, not the rate at which it travels between shapes (which is known as the Travel Rate)

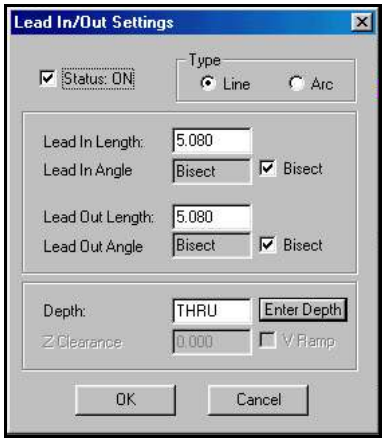
The Plunge Rate is the speed at which the tool is driven down in to the material. Again you may type in a value or you may click on **MAX** to automatically select the maximum plunge rate for your machine.

The figures that you enter here will directly affect cut edge finish, tool life and machine wear. Entering incorrect figures here can lead to poor edge finish, tool breakage or excessive machine wear.

Speed setting are overridden if the speed at the machine entered by Function 1 (Feed Rate) or Function 15 (Plunge Rate) is *slower* than those set in this dialogue box, and also by the Last Pass Speed setting in the Multipass Settings if Multipass is ON.

As a safety precaution the Multicam machine will always run at the slower of the speeds set with Function 1 or the speed setting in this Output menu.

Lead In



For Lead In/Lead Out to work you must have the Status check box ticked ON.

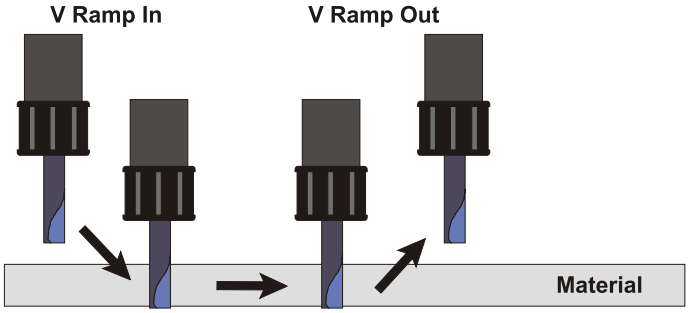
There are some instances where the use of Lead In and Lead Out may be desirable.

Firstly some router bits do not plunge very well and may have a tendency to damage the material in the vicinity of the start and end points. Secondly you may want to have the tool enter the material away from your shape to preserve a good quality edge.

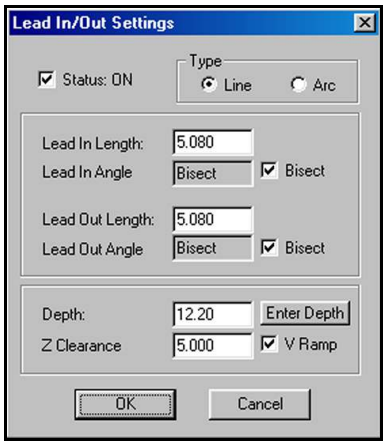
The Lead In/Lead Out may be set in various ways. You can choose from 2 different Lead In/Lead Out types (Line or Arc). You can select the lead in length and angle as well as the lead out length and angle for each style. Instead of selecting a specific angle you may choose **Bisect** which stands for bisecting angle. This is usually the best setting as it allows the controller to calculate a lead in angle that will not interfere with the final cut.

Vertical Ramping (Plunge Ramping)

The Lead In/Out Settings dialogue box also allows you to set a **Vertical Ramp** (V Ramp). Vertical ramping causes the machine to plunge in to the material at an angle rather than using a purely vertical plunge. At the end of the cut the tool then ramps out of the material. Vertical Ramping is especially useful for cutters that do not plunge well or for large diameter cutters that tend to also have trouble plunging. Another benefit of using **V Ramp** is that it acts like a **TAB**. Cutting forces toward the finish of the job are reduced because the cutter is not removing the full thickness of material; a useful feature for small pieces that can move slightly once the cutting process is finished.



To enable V Ramping you must firstly enter a depth in the Lead In/Out Settings dialogue box, then check the V Ramp check box. The depth that you enter here will automatically be transferred to the depth setting in the Group Settings dialogue box. If there is already a depth set (other than THRU) in the dialogue box then you will notice that this depth is already placed in the Lead In/Out Settings dialogue box, so all you need to do check the V Ramp check box to enable the function.



The Depth setting should correspond to the *Material Thickness* plus a small through allowance.

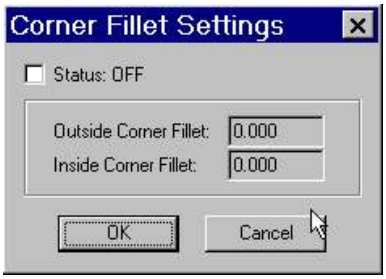
The **Z Clearance** is the distance *above* the material that the V Ramping will commence and finish. It should always be set to the same setting as the lift top setting you set in Function 4.

Once you OK the settings in the Lead In dialogue box with V Ramp enabled you will notice that the Group Setting dialogue box will have the values for Depth and Lead In updated.

NOTE: *If you are using lead ins,* ***the compensation must be set in ToolPath*** *as Function 28 at the controller will be disabled.*

IMPORTANT: If you are using more than one Group in your job you must also set the depth for these groups. Do not leave the depth set at THRU. This is because when V-Ramp is selected Toolpath now treats the job as a full 3D job and must have absolute depths to work with.

Fillet

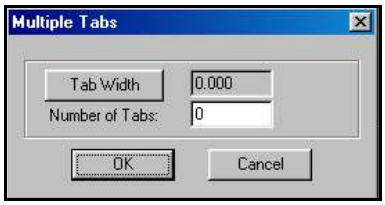


This is a useful feature if you are running an infill job (cutting shapes and cutting a corresponding face to fit the shapes into) The usual problem is that router bits cut outside corners square and inside corners with the bit's radius. The solution is to fillet all the corners with the bit's radius. Setting this value to the bit radius will automatically adjust the tool path as required.

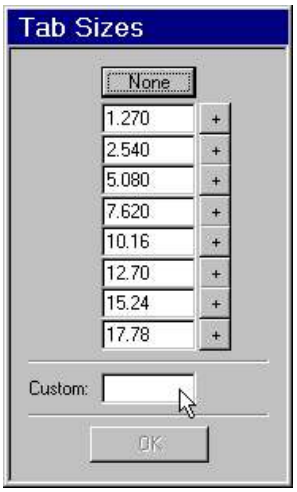
From the **Corner Fillet Settings** dialogue box check the Status check box and click on either **Outside Corner Fillet** or **Inside Corner Fillet**.

From the **Fillet Sizes** dialogue box choose the size of the fillet you wish to apply or type in a custom value.

Tab Width



The Tab Width command will allow you to automatically add tabs to your job. Clicking in the Tab Width field will present you with the Multiple Tabs dialogue box. Click on the Tab Width button to select and enter the width of the tabs to be automatically added to your job. Then click in the Number of Tabs field and type in the number of tabs required.



Tabs are useful if you are cutting lots of small shapes that threaten to pop out and jam the cutting head or get damaged by fouling the router bit once they are cut. It is also useful if you are cutting thick materials such as aluminium and perspex.

Using this feature with a small tab width, i.e. 1 mm will leave a tiny tab of material holding the job together. When the job has finished, simply break the parts away and file the unfinished section.

When a tab or tabs are added to the job the Tab Width field on the Group Settings for Machine Control dialogue box will be updated. For example if the Tab Width field displays 2(1.000) this means that there a 2 tabs of 1mm on each shape of the job.

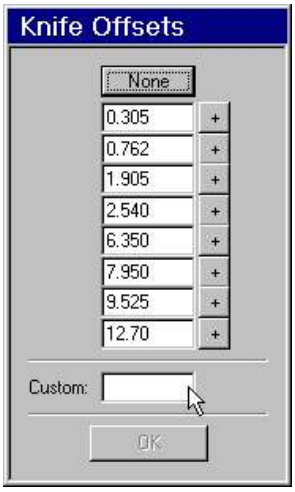
You can visually check the Tabs you have added to the job in the output screen. Each break in the job represents a tab location.

Tabs are placed evenly around the perimeter of the shape commencing from the Start Point of the shape. If the tab placement does not seem correct then you can re-position the tabs by moving the Start Point of the shape. Refer to page 51 for more information about Start Points.

NOTE: *If you attempt to set your machining path to reverse whilst you have multiple tabs set you may encounter irregular machine behaviour where the machining path direction may be overridden. You will be able to see any inconsistencies in the output screen prior to transmitting the job to the machine. It is recommended that you do not set your machining path to reverse when using multiple tabs.*

Knife Offset

The most commonly used knife is a swivel or trailing knife. The point of the knife is slightly offset from the centerline of the cut. This will normally round the corners slightly. To eliminate this problem the knife offset must be set appropriately and the program will adjust the tool path for the offset of the knife. Standard Roland Digital knives have an offset of **0.25 mm**. Most Mimaki knives have an offset of **0.3 mm** You can also enter a custom Knife Offset if you wish.



Send

Once all the values for your job are set, click the **Send** button to send the job to the machine. The program will redraw the job with the above adjustments and display the memory requirements of the job on the Menu Bar;

If all is OK the display should be in Cyan. If the display is in Yellow this means that there is a problem.

If your design contains any errors, especially compensation settings, you can view the file now one last time before transmitting. What you see on this screen is what the machine will cut!

Click on **Transmit.** The job will be transmitted to the machine. Once sent the computer is free to continue designing work. Job files can be transmitted to the machine while it is running.

Click on **Main Menu** or press the ESC key on the keyboard to abort sending the job to the machine.

Save Ready File

Once a file has been prepared as a machine tool path, it is often desirable to save it as a ready or **.rdy** file for future use. Unlike .**DXF** or **.PLT** files, **.RDY** files preserve all the settings exactly as programmed in **Toolpath**. When they are re loaded, they can be sent immediately to the machine without additional editing.

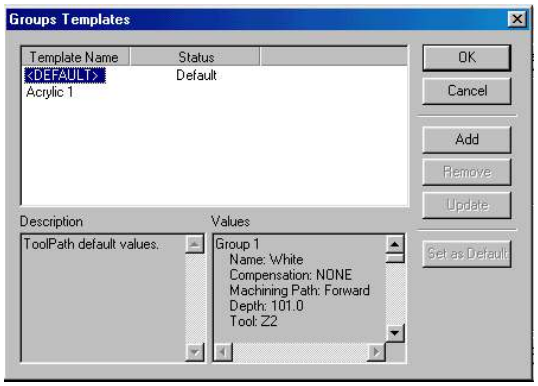
To create a **.RDY** file select **Save Ready File** from the **Output** dialogue box. A file manager menu will appear which will allow you to choose the disk, and the directory in which you wish to save the **.RDY** file. Select **OK** when you have set the correct file path.

A recent modification to the **Export Ready File** now allows you to save the file as any type, by selecting **File Type** from the menu, and selecting your desired type. Again, select **OK** when you are ready to save. This can be useful if you have modified the file by deleting shapes, or if you have used **ToolPath** to manually nest shapes.

NOTE: *Files other than .rdy files do not maintain the job settings such as groups and compensations, but will maintain moved shapes, or deleted shapes.*

The ready file, or any other kind of file, can be recovered using the **Files** command.

Group Templates



If you find that you frequently machine the same types of materials with the same tools and the same speed settings then you can speed up the output process even further by utilising **Templates**. Templates allow you to save the settings that you have entered in the output dialogue box and the recall them for later use with a different job.

To create a template click on the **Templates** button and, in the **Groups Templates** dialogue box that appears, click on **Add**. Type in a name for the template you wish to create and a meaningful description, then click on **OK**

Your template will now appear under the column *Template Name*. At this point you will note that the *Status* of your template will say "Not Available". This is because the current values set in the **Output** Dialogue box have not yet been assigned to your newly created template. To assign the values click once on your template in the templates window so that it is highlighted, then click on **Update**. Your template has now been assigned the values you entered in the **Groups Templates** dialogue box.

To use your template later on with a new job, in the **Output** dialogue box click on **Templates**, select the template you wish to use, and click on **OK**. The values contained in your template will be assigned to the current job.

Other functions of the **Groups Templates** dialogue box are;

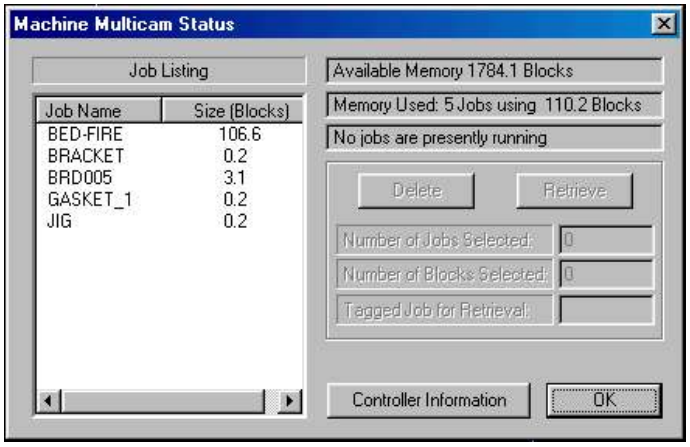
Remove Clicking on this will remove your template from the template list. This process is irreversible, however you will be asked to confirm the action.

Update Use this to add values to a newly created template or to change the values in an existing template.

Set As Default Clicking this will set the template you select as the *Default* template so that next time you enter the **Output** dialogue box the setting of this template will already be loaded.

STATUS

The status menu allows you to query the machine from within ToolPath to see which jobs are present at the machine, which job (if any) is currently running and the size (in blocks) of each job at the machine. The machine must be turned on and connected to your computer for status to work.



Machine Status also tells you the amount of memory currently in use and the amount of free memory available at the machine.

From within this dialogue box you can also delete jobs from the machine controller and retrieve jobs back from the machine into ToolPath.

Finally you can also verify information about the controller including versions of the software currently installed.

Flashram

The Multicam controller utilises Flashram technology to store jobs and controller information at your machine. Flashram is a dynamic form of memory that allows you to store, retrieve and delete jobs on the machine in a similar way to hard disk drives in a standard computer. The most desirable feature of Flashram technology is that it allows you permanently store jobs at the machine controller even after the machine has been turned off.

Because of the way the memory on the Flashram module is handled you may occasionally see a situation where you are not able to transmit a job to the machine even though the size of the job is less than the amount of available memory. You may also notice that, at times, memory does not seem to be “freed up” after you have deleted jobs from the controller. The reasons for this are quite technical and are described briefly below, however it should be noted that this is not a phenomenon of the Multicam controller, all Flashram acts in this way on all types of equipment.

The memory of the Flashram module is divided into 32 sectors. This enables the controller to erase sections of memory allowing greater flexibility than would normally be possible with Flashram modules. When you delete a job from the controller the job is not actually deleted. Rather it is marked for deletion and is physically removed when the entire sector that that job occupies is marked for deletion. This means that sometimes the amount of available memory is not updated until a full sector is freed up. Note that if you delete a job from a sector that contains other jobs, the memory for that sector is not made available until all the jobs in that sector are deleted. These situations usually only occur when you are working at close to the full capacity of the Flashram module.

The Flashram module will automatically “optimise” when it decides the time is right, however sometimes you can try to force an optimisation by performing a Function 18 at the machine.

Deleting Jobs from the Machine Controller

To delete a job from the machine click once on the name of the job you wish to delete, with the left mouse button, to highlight it and then click on the **Delete** button. You will be prompted to confirm the action before the job is deleted.

You can select more than one job by clicking once on the first job name with the left mouse button then pressing and holding down the **Ctrl** key on the keyboard and then selecting the other jobs you wish to delete by clicking on them with the left mouse button. The selected jobs will be shown highlighted in the *Job Listing* window. You can select and deselect the required jobs in any order. Click on the **Delete** button. You will be prompted to confirm the action before the jobs are deleted.

To select a number of consecutive jobs select the first job by clicking on it once with the left mouse button then press and hold down the **Shift** key on the keyboard and click on the last job you wish to delete. Jobs between the first and last selected job names will be highlighted. Click on the **Delete** button. You will be prompted to confirm the action before the jobs are deleted.

Retrieving Jobs from the Machine Controller

If for some reason you wish to retrieve a job from the machine controller, click on the job you wish to retrieve, then click on the **Retrieve** button. The selected job will be retrieved from the machine and once retrieval has completed will be displayed in the ToolPath main screen.

You can observe the retrieval process by watching the progress bar on the status bar at the bottom of the ToolPath main screen.

Note: The file that is retrieved will include all the settings that was applied to it, such as compensation. If the job contains plunge or drill points it may not be able to be retrieved.

Controller Information

To check version of the operating system is currently loaded on your machine and the version of ToolPath you are currently using click on **Controller Information**. The dialogue box you see to the right will be displayed showing version numbers of the respective software. Click **OK** to close the dialogue box.

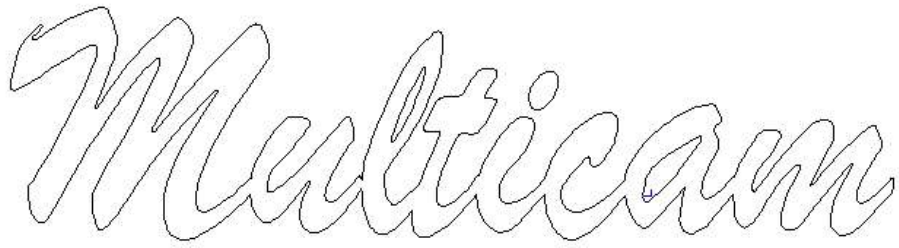
SPECIAL

The special menu is where you perform your **2D** and **3D** engraving as well as **Projections** over shapes, and **Drill Patterns**. You can also use the **Move to Group** command to move parts of your job to different groups to aid in setting up these special functions.

Engrave 3D

3D Engraving creates a three dimensional ToolPath from a two dimensional image. The depth of the 3D ToolPath is calculated by ToolPath to maintain the correct stroke width of the 2D shape using a "V" cutter.

For example script and brush type fonts have varying widths depending on the vertical and horizontal stroke of the letter. The 3D module creates a tool path that will maintain the correct widths by raising and lowering the "V" cutter as the head moves about its X and Y axis. This effectively maintains the correct widths of the stroke.



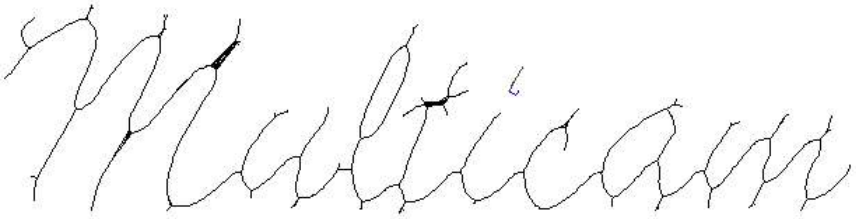
3D design before the tool path is calculated as imported into ToolPath.

In this example, the software will create a tool path in the centre of the letters that will maintain the width of the letters by altering the depth of the cut but not exceeding the **Max Depth** setting set in the 3D dialogue box. Where the depth is not deep enough to allow the required width to be reached, an "island" will be created which may need to be hatched out.

3D design after the tool path has been calculated showing the actual tool path within the lettering and some small areas of hatching.

If you are satisfied with the result click on **Enter** on the Menu Bar at accept the job. If however you wish to change the 3D effect click on **Undo** to try again.

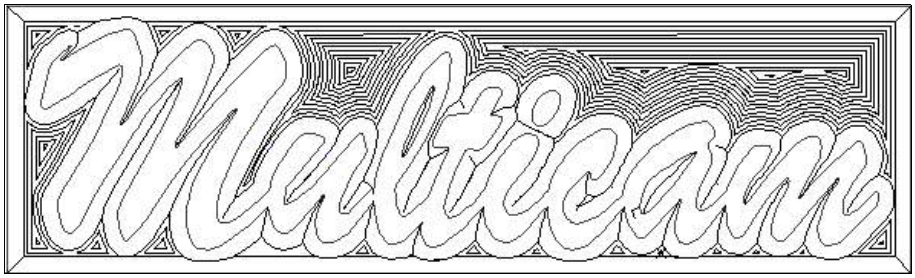
Below is the resulting tool path that will be created and transmitted to the machine.



The resulting 3D tool path that has been generated. The tool path shown represents only the path the point of the 3D tool will travel.

Reverse 3D is where a tool path is generated that will result in the design being raised from the background material and can be accomplished by simply placing border around the design you wish to engrave. This must be done in your design program. When reverse 3D engraving is performed you will need to hatch out the background (i.e. the space between the outside of the lettering and the border as well as the centres of letters such as "A" or "O")

This hatching is performed by using the **Fill Type** and **Fill Space** commands. The maximum depth of the fill for the reverse 3D is set by the **Max Depth** command.



Reverse 3D effect creates raised lettering.

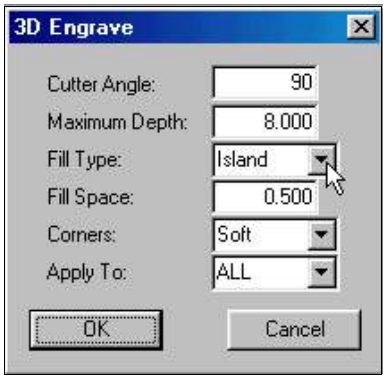
As discussed, ToolPath must work out the depth of the cut to maintain the correct width of the shapes. You may however set the maximum depth the cutter will travel to suit the material thickness or the maximum cutting depth of the cutter used by using **Max Depth**. Remember this is the **Max**imum depth that the tool will travel to, not the actual depth of the cut. It is possible that during calculations, the program will determine that the tool need not even go to the maximum depth.

3D Engraving Procedure

Bring your design into ToolPath as you normally would. Select **Analysis** to correct any sequence and/or direction errors.

Note: Open shapes (which appear green) cannot be engraved.

From the Menu Bar select **Special**, then **Engrave 3D** and the 3D Engrave dialogue box will appear.



Enter the required values for your 3D engrave. Note that some of the settings can be selected by clicking on the downward pointing arrow and choosing from the available options.

Once all the values have been entered click on **OK** to have ToolPath calculate the tool path for you. This process may take some time depending upon the complexity of the design and the number of shapes in the design.

It is possible to enter some parameters in this dialogue box that will cause

ToolPath to generate an error message. The most common cause of this is choosing a **Fill Type** for your design, but ToolPath finds that no fill (or hatching) is required because of your **Maximum Depth** setting. The error message simply indicates that some of your settings will be ignored.

Following is more detailed information on each of the settings in the 3D Engrave dialogue box.

Cutter Angle: The most common "V" cutter form is 90°, but other angles are available. You may select an angle from 1° to 135°. The greater the angle, the shallower the 3D depth will be. The angle you set must exactly match your true cutter angle for correct results. The angle entered must also be the *included* angle of the cutter.

Max Depth: This sets the maximum depth the tool will be allowed to travel. Set this depth bearing in mind your material thickness and the maximum cutting depth of the cutter in use. If this restriction creates islands in the centre of shapes, set a **fill type**.

Fill Type: There are two types of fill that may be selected. These are **None** and **Island Infill**. If the **Fill Type** is set to **Island** and ToolPath determines that none is required, an error message will be generated in this event select "OK". This does not affect the job.

You do not need to set a fill type if you do not wish to do so. If a fill type is required but not set the finished job will contain a number of "hills" where the fill would have normally been hatched away.

Fill Space: This is the overlap for the **Island Infill**. For a "V" cutter, 1mm is a good general selection. A smaller **Fill Space** will generate closer hatching and larger files. The job will also take longer to run.

Corners: You can select **Soft** or **Hard** corners. **Soft** is the default setting. The **Hard** corners option is a lot slower in operation. On most jobs, **Soft** corners is the best choice.

Apply To: It is also possible to engrave parts of a job and not engrave other parts. Use the **Move to Group** command on the Menu Bar to assign parts of your design to different groups and then the Apply To parameter to choose which group you wish to apply the current 3D engraving settings to. Using the **Groups** command it is also possible to engrave various parts of jobs using different parameters.

Refer to the section on Groups in this manual, commencing on page 85 for instruction on how groups work and are assigned.

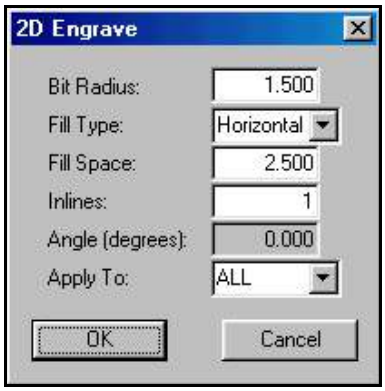
Engrave 2D

Two Dimensional Engraving is used to hatch out a design. As with 3D engraving reverse hatching can be accomplished by placing a box or border around the design you wish to engrave. ToolPath will hatch out between the border and your design, leaving your design raised.

2D Engraving Procedure
Bring your design into ToolPath as normal. Select **Analysis** to correct any sequence and/or direction errors. Select **Special** from the Main Menu bar, then select **Engrave 2D**

NOTE: *Open shapes (which appear green) cannot be engraved.*

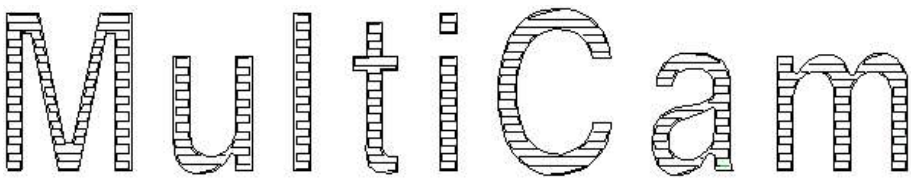
Enter the required values for your 2D engrave. Note that some of the settings can be selected by clicking on the downward pointing arrow and choosing from the available options.



Once all the values have been entered click on **OK** to have ToolPath calculate the tool path for you. This process may take some time depending upon the complexity of the design and the number of shapes in the design.

It is possible to enter some parameters in this dialogue box that will cause ToolPath to generate an error message. The most common cause of this is choosing a **Fill Type** for your design, but ToolPath finds that no fill (or hatching) is required. The error message simply indicates that some of your settings will be ignored or cannot be applied.

Once the engraving has complete you may accept the result by selecting **Enter**. If it is not satisfactory, select **Undo** and try a different bit size, fill space, or fill type, or a combination all three.



2D design after the tool path has been calculated showing the actual tool path within the lettering and hatching.

You can output your design to your machine by using the standard procedures. You do not need to do an analysis of the tool paths because the **Engraving** procedure does it automatically prior to doing the **Engrave**.

When you output to your machine you should set the required depth for the engraving. All other variables in the **Output** menu except **Job Name** are ignored by the engraving procedure. This is done to prevent any changes in the path or tool offsets that may result in an incorrect cut.

NOTE: *The quality of the final result is very dependent on the bit size used. If you attempt to use a bit that is too large for the detail of the job the results can be erratic!*

NOTE: *Once a job is engraved the bit compensation has already been calculated into the formulae. The engraved group of shapes will not accept a compensation value, however the operator **must** ensure that compensation is set to 0 at the sub console.*

Following is more detailed information on each of the settings in the 2D Engrave dialogue box.

Bit Radius: This is **half the diameter** of the end mill you are using. If the radius you choose is not the same as the cutter that is in your machine, you may end up with some unexpected results.

Fill Type: There are four types of fill that may be selected. **Horizontal**, **Vertical**, **Diagonal** and **Island Infill**. The type of fill you select is largely a matter of personal preference, however consideration should be given to the overall shape of your job. Selecting the correct fill type for your job can hale reduce processing and cutting time.

For complex designs, **Horizontal**, and **Vertical** are the recommended hatching styles. The **Island Infill** is more effective on simple designs, and designs without tight corners.

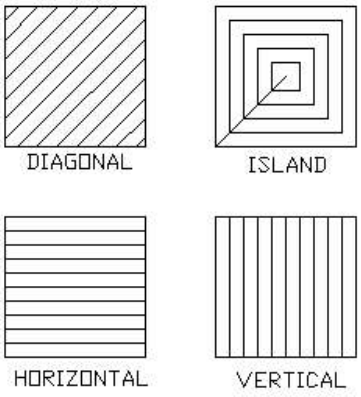
Fill Space: This is the space between the lines of the fill. For **Vertical** and **Horizontal** and **Diagonal** fill types the fill space should be approximately two thirds (66%) of the cutter diameter. This will allow for an overlap to ensure a smooth cut out. If you choose **Island Infill**, the Fill Space is set automatically for you.

Inlines: This will create a cleanup pass around the perimeter of each of the shapes in your design to remove any dent or gouge marks caused by the fill type you have chosen . For **Vertical**, **Horizontal** and **Diagonal** hatching, **Inlines** default setting is 1 but you can enter other values. If you choose **Island Infill**, Inlines is set automatically for you.

Angle (degrees): Available only if you choose a **Diagonal** fill type, you can enter an angle that you wish the hatching to be carried out at.

Apply To: It is also possible to engrave parts of a job and not engrave other parts. Use the **Move to Group** command on the Menu Bar to assign parts of your design to different groups and then the Apply To parameter to choose which group you wish to apply the current 3D engraving settings to. Using the **Groups** command it is also possible to engrave various parts of jobs using different parameters.

Refer to the section on Groups in this manual, commencing on page 85 for instruction on how groups work and are assigned.



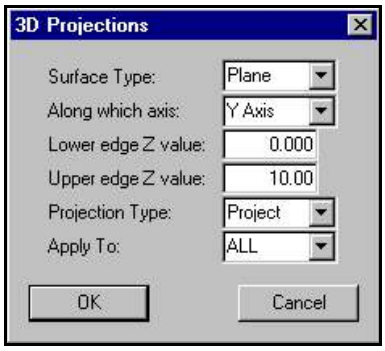
Representation of different Fill Types.

Projections

The projections command allows you to "project" your design onto a number of predefined surfaces. These are **Plane**, **Cylinder**, or **Globe**.

Plane: This is probably the most common type of projection you will use as it allows you to place your design on a "slope". This is useful for such things as drain patterns in bench tops. You can apply the slope to either the X or Y axis. The slope is entered as a difference in the Z depth between the left edge of the design and the right. It is not an angle in degrees.

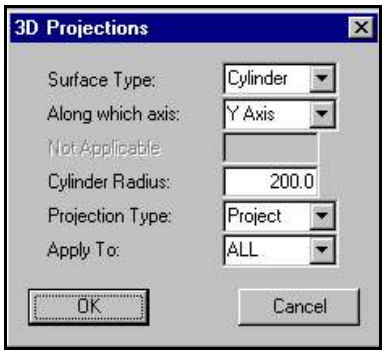
If your design makes use of Groups, you can choose to have the projection applied to a single group or the entire design by making your choice in the **Apply To** drop down list.



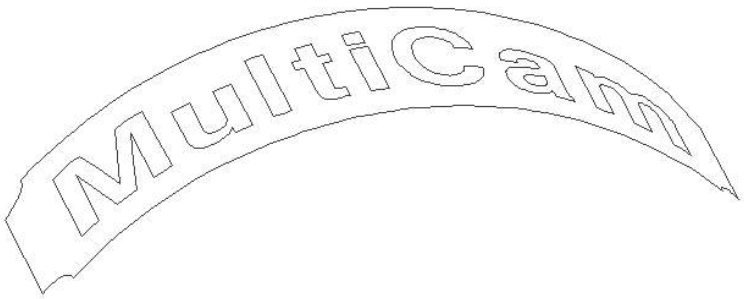
Cylinder: You can also project your design over a cylinder which may be useful if you need to engrave or machine a piece of curved extrusion (such as a pipe).

It is especially useful if you need to cut a pocket or window in a curved extrusion as it will maintain the correct dimensions of the cut out. In this case you should set the **Projection Type** to **Wrap**.

Below is a 3D view of a design that has been projected over a cylinder.

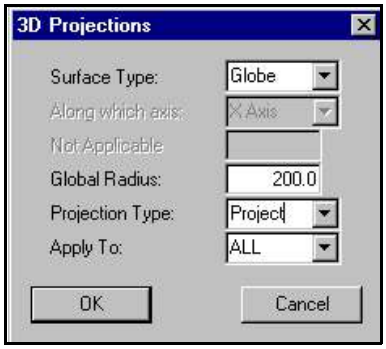


Note: When engraving over a Cylindrical or Spherical surface you should use a ball nose cutter that will allow machining over a curved surface. Flat bottomed cutters will cause gauging in the bottom of your job.



3D view of design projected over a cylinder.

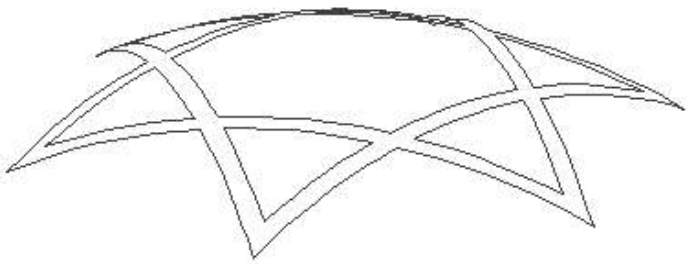
Globe Projecting a design over a Globe is basically the same as projecting over a Cylinder, however the projection is applied in all directions of the design and uses the radius of the Globe to determine the new shape of the job.



You may choose to Project your design over the globe or Wrap it around the globe.

If your design makes use of Groups, you can choose to have the projection applied to a single group or the entire design by making your choice in the **Apply To** drop down list.

Below is a simple design that has been Projected over a globe.



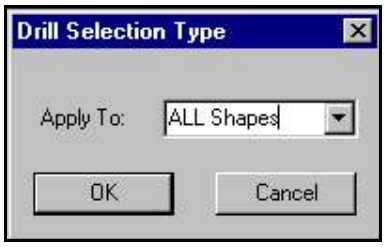
3D view of a design projected over a Globe.

Drill Patterns

The **Drill Pattern** feature has three options; **Centre**, **Outline** and **Fill**. They are used to create a series of drill holes either in the centre of the design, around the outline or to fill the objects on the screen..

You may apply the drill pattern to ALL shapes, a single GROUP or USER select. If USER select is chosen click on the parts of the design that you wish to convert to drill points and then click on **DRILL** on the **Menu Bar**.

Centre Will place a plunge point at the centre of the objects on the screen. You can make use of the **Groups** (Groups command from the Main Menu) function to assign certain parts of your job to a different group and apply the centre drill pattern to that group only. This is very handy for such things as gaskets or cover plates that have bolt holes in them.

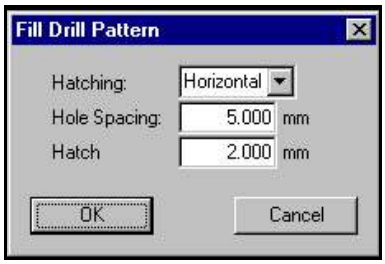


Outline Will place a series of holes around the perimeter of the shapes in your design. Once again you can apply the drill pattern to selected groups in your design or apply the drill pattern to the entire design.

You will then be prompted to enter the hole measurement, ToolPath will then calculate the correct positions of the holes based on this size. If you are using a 3mm drill and want to have a 2mm space between the holes you must enter 5mm in this dialogue box. The dialogue does not take into account the size of the tool that you actually use.

Remember that these holes are just plunge points. If you require 3mm holes you must use a 3mm diameter tool.

Fill Will outline your design with holes as well as fill your design with holes. Once again you can apply the drill pattern to selected groups in your design or apply the drill pattern to the entire design.



You will then be prompted for the required *Hatching, Hole Spacing and Hatch*.

You can choose from either Vertical or Horizontal Hatching.

For your Hole Spacing *remember that the hole spacing must allow for the hole diameter.* If you require a drill hole of 3mm with 2mm spacing a 5mm *Hole Space* would be entered.

The *Hatch* figure is the distance between the holes used in the fill.

Remember that these holes are just plunge points. If you require 3mm holes you must use a 3mm diameter tool.

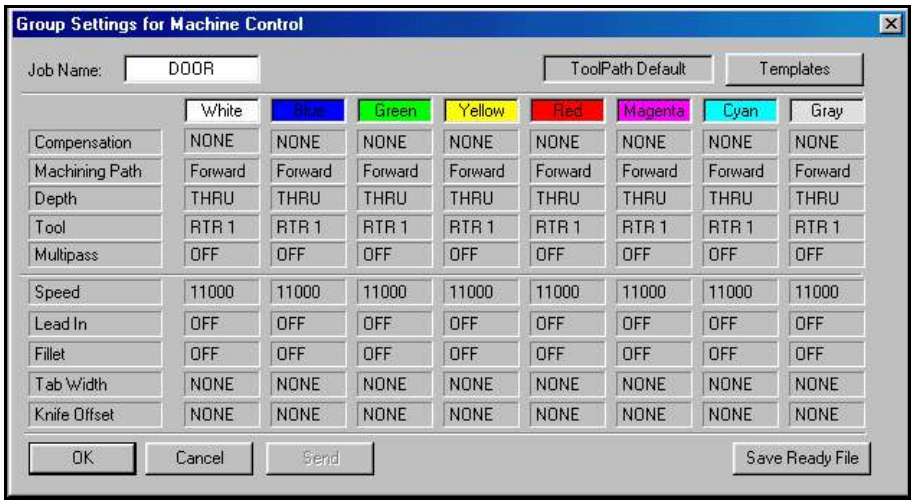
GROUPS

The **Groups** function is a useful tool in setting up more complex jobs, in terms of using different tools, and different heads. An example would be, you need to engrave a design with your engraver, drill holes and use the router to cut the job out of the material. The Groups commands will enable you to do this easily.

When you click on **Groups** from the main menu, all the shapes in your design will now change colour to indicate their corresponding group. Normally when you first use this command, all the shapes will be in the White group. The menu bar at the top of the screen will be:

Group Setup

Choosing **Setup** from the **Groups Menu** will allow you to activate the specialized group features on a group by group basis. The **Group Setup** dialogue box will appear:



For each group you may set the Compensation, Machining Path (direction), Depth, Tool, Multipass, Speed, Lead In, Fillet, Tab Width or Knife Offset.

To change a setting for any of the groups simply click on the item you wish to change and enter the new value in the dialogue box that appears.

N.B. The default group colour for a normal job is the WHITE group.

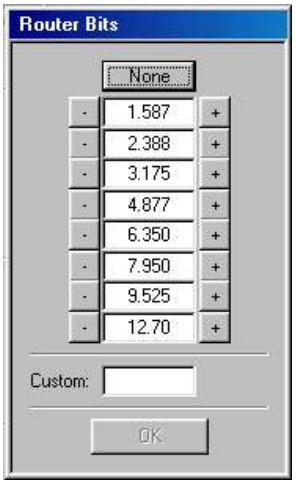
Job Name

The job name is normally defaulted to the name of the file it was created from. This is the name which will appear on the Sub-Console, and the name of the file if it is saved as a ready file. If a different name is required, select this item and type a new name in its place.

Templates

You can use the **Templates** function to save commonly used settings in Groups for later use. Refer to page 74 for details on how to activate and use templates.

Compensation



If a router bit is being used this value is the radius (or half the diameter) of the bit. You can set up ten standard compensations for the cutters most commonly used, and select the appropriate one as required by clicking on either the + button next to the required compensation value for positive compensation or the **-** button for negative compensation. You can also enter a custom compensation value by entering the value in the Custom box and clicking the **OK** button

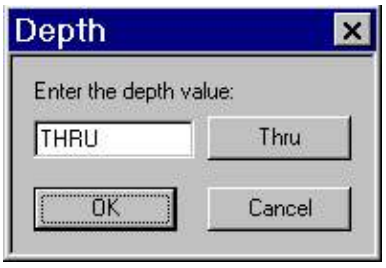
NOTE: *It is also possible to set the compensation at the machine using the sub console's Function 28. Use one or the other* ***not both*** *or the result will be double compensation! Using Function 28 at the sub console is not as accurate for complex shapes.*

Machining Path

Normally the direction is set to **forward.** This ensures that all cuts are conventionally milled: outside shapes are cut counter clockwise, inside shapes are cut clockwise. If you set this value to **reverse** the machine will climb or back cut all the shapes. To change the direction of the Machining Path click on the direction currently displayed. You can toggle between Forward, Reverse or Off.

NOTE: *If you attempt to set your machining path to reverse whilst you have multiple tabs set you may encounter irregular machine behaviour where the machining path direction may be overridden. You will be able to see any inconstancies in the output screen prior to transmitting the job to the machine. It is recommended that you do not set your machining path to reverse when using multiple tabs.*

Depth



Here you can enter the depth at which the machine should cut the job. **THRU** indicates that the machine will cut right through the job to the bottom of your Function 4 setting. This is the setting mostly used for cutting out shapes. If you do not want to cut right through your material enter the depth to which you want the machine to cut and then click the **OK** button.

To reset your depth to Thru simply click on the **Thru** button or enter the letter **T** for your cutting depth and then click the **OK** button.

NOTE: *For safety it is always preferable to enter the thickness of your material (plus a small cut through allowance of say 0.2mm) in the depth setting field. This will prevent you from plunging in to your machine table top in case you accidentally incorrectly set your Function 4 setting.*

Tool

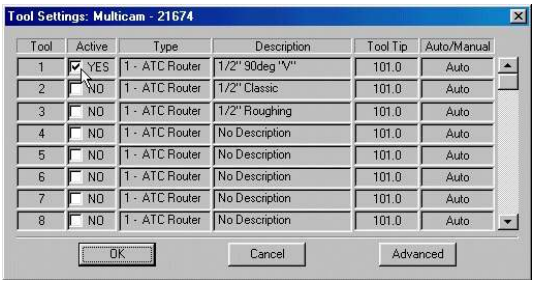
If you have an Automatic Tool Change Unit fitted to your machine, or if you have it set up as an Automatic Tool Change Unit you may be presented with the dialogue box pictured on the right when you click on Tool in the Group Settings for Machine Control.

For now you should just click on the Standard Tool button to bring up the Tool Settings dialogue box.

The External Tool and Profile Tool buttons are reserved for future expansion of Toolpath and are currently not fully implemented.

Click on Cancel to close the dialogue box.

The **Tool Settings** dialogue box allows you to specify which tool is to be used to machine a particular tool path. You can only assign one tool to each group in your job.



You will usually only use the **Tool Settings** dialogue if your machine is fitted with more than one routing head, if you have a quick change motor, automatic tool changer or if you have other devices attached to your machine such as pneumatically controlled drill heads.

Unless you have set up multiple tools in the **Setup** menu you will not be able to activate or select different tools for your job. Refer to the section on **Setup|Configure Tooling** on page 95 for instructions on how to do this. For a detailed explanation of multiple tools and their setup and use refer to Appendix F commencing on page 161 and Appendix G commencing on page 165.

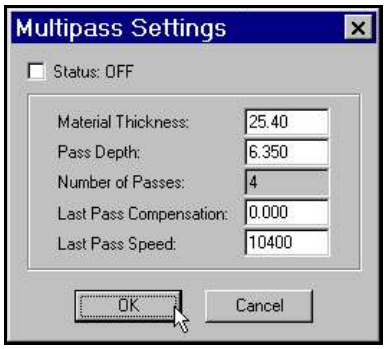
The desired tool is chosen from the list between 1 and 16. The default tool is tool number 1.

Tools may be set Active or Inactive by clicking on YES/NO in the active column.

The Multicam controller supports a number of different head types: Routers, Drill, Plasma, Airtool, Spare, Engraver, Studweld, No Tool.

Multipass

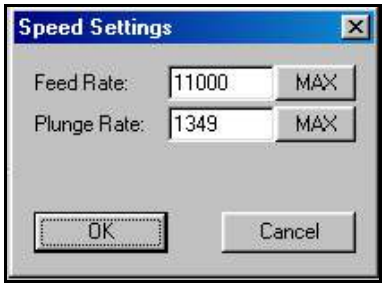
This feature may be turned “ON” or “OFF” to enable a job to be cut in several passes by clicking the check box labelled **Status**. When status is ON the values you enter for *Material Thickness, Pass Depth etc.*. Will be enabled.



Material Thickness and **Depth per Pass** may be set as required by using the pointer to select the appropriate fields and setting suitable values. The **Num. of Passes** field will automatically be adjusted. The **Last Pass Compensation** is used to "clean up" the layering effect that is caused by Multipass cuts. It should be set to be a small negative number so that the bit will move slightly into the cut on the last pass to clean up the edge. Sometimes a positive value is entered to create a shoulder.

Multipass may be set ON or OFF by selecting desired group in the Machine Control (Groups) window.

Speed



Enter the Feed Rate at which you want the job to be cut or you may click on **MAX** to automatically select the maximum feed rate for your machine.

Feed Rate is the speed at which the machine will move whilst cutting, not the rate at which it travels between shapes (which is known as the Travel Rate)

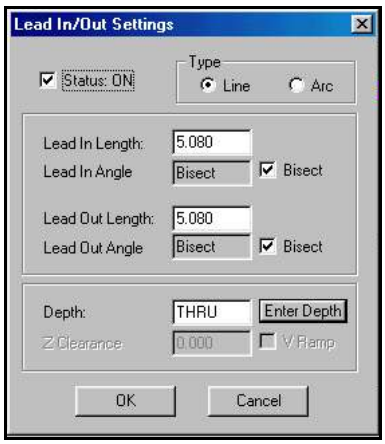
The Plunge Rate is the speed at which the tool is driven down in to the material. Again you may type in a value or you may click on **MAX** to automatically select the maximum plunge rate for your machine.

The figures that you enter here will directly affect cut edge finish, tool life and machine wear. Entering incorrect figures here can lead to poor edge finish, tool breakage or excessive machine wear.

Speed setting are overridden if the speed at the machine entered by Function 1 (Feed Rate) or Function 15 (Plunge Rate) is *slower* than those set in this dialogue box, and also by the Last Pass Speed setting in the Multipass Settings if Multipass is ON.

As a safety precaution the Multicam machine will always run at the slower of the speeds set with Function 1 or the speed setting in this Output menu.

Lead In



For Lead In/Lead Out to work you must have the Status check box ticked ON.

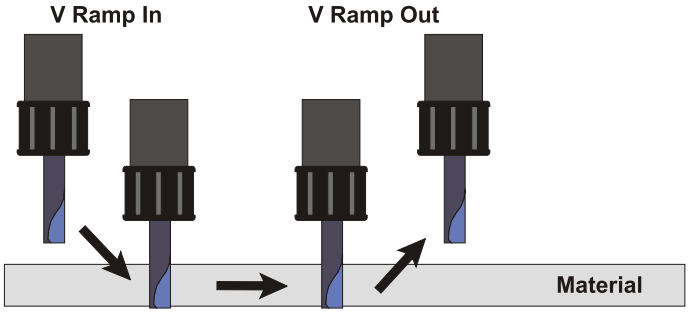
There are some instances where the use of Lead In and Lead Out may be desirable.

Firstly some router bits do not plunge very well and may have a tendency to damage the material in the vicinity of the start and end points. Secondly you may want to have the tool enter the material away from your shape to preserve a good quality edge.

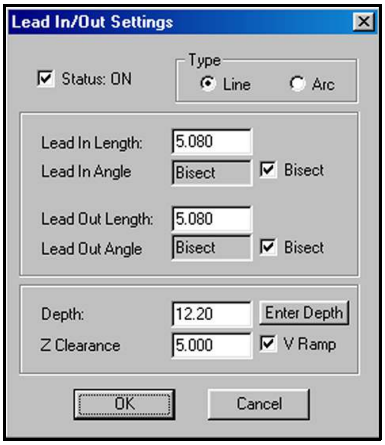
The Lead In/Lead Out may be set in various ways. You can choose from 2 different Lead In/Lead Out types (Line or Arc). You can select the lead in length and angle as well as the lead out length and angle for each style. Instead of selecting a specific angle you may choose **Bisect** which stands for bisecting angle. This is usually the best setting as it allows the controller to calculate a lead in angle that will not interfere with the final cut.

Vertical Ramping (Plunge Ramping)

The Lead In/Out Settings dialogue box also allows you to set a **Vertical Ramp** (V Ramp). Vertical ramping causes the machine to plunge in to the material at an angle rather than using a purely vertical plunge. At the end of the cut the tool then ramps out of the material. Vertical Ramping is especially useful for cutters that do not plunge well or for large diameter cutters that tend to also have trouble plunging. Another benefit of using **V Ramp** is that it acts like a **TAB**. Cutting forces toward the finish of the job are reduced because the cutter is not removing the full thickness of material; a useful feature for small pieces that can move slightly once the cutting process is finished.



To enable V Ramping you must firstly enter a depth in the Lead In/Out Settings dialogue box, then check the V Ramp check box. The depth that you enter here will automatically be transferred to the depth setting in the Group Settings dialogue box. If there is already a depth set (other than THRU) in the dialogue box then you will notice that this depth is already placed in the Lead In/Out Settings dialogue box, so all you need to do check the V Ramp check box to enable the function.



The Depth setting should correspond to the *Material Thickness* plus a small through allowance.

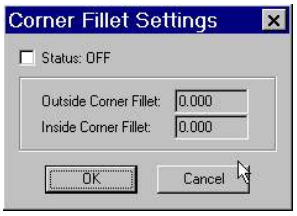
The **Z Clearance** is the distance *above* the material that the V Ramping will commence and finish. It should always be set to the same setting as the lift top setting you set in Function 4.

Once you OK the settings in the Lead In dialogue box with V Ramp enabled you will notice that the Group Setting dialogue box will have the values for Depth and Lead In updated.

NOTE: *If you are using lead ins,* ***the compensation must be set in ToolPath*** *as Function 28 at the controller will be disabled.*

IMPORTANT: If you are using more than one Group in your job you must also set the depth for these groups. Do not leave the depth set at THRU. This is because when V-Ramp is selected Toolpath now treats the job as a full 3D job and must have absolute depths to work with.

Fillet

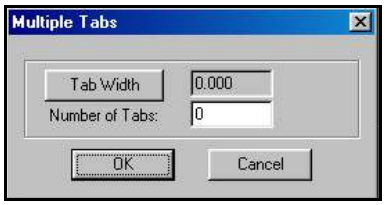


This is a useful feature if you are running an infill job (cutting letters and cutting a corresponding face to fit the letters into) The usual problem is that router bits cut outside corners square and inside corners with the bit's radius. The solution is to fillet all the corners with the bit's radius. Setting this value to the bit radius will automatically adjust the tool path as required.

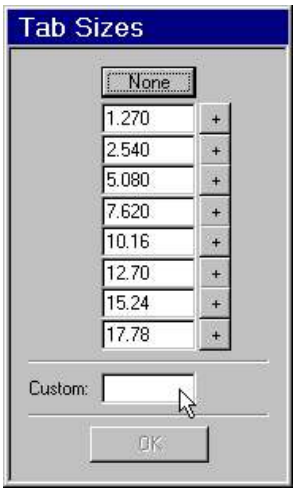
From the **Corner Fillet Settings** dialogue box check the Status check box and click on either **Outside Corner Fillet** or **Inside Corner Fillet**.

From the **Fillet Sizes** dialogue box choose the size of the fillet you wish to apply or type in a custom value.

Tab Width



The Tab Width command will allow you to automatically add tabs to your job. Clicking in the Tab Width field will present you with the Multiple Tabs dialogue box. Click on the Tab Width button to select and enter the width of the tabs to be automatically added to your job. Then click in the Number of Tabs field and type in the number of tabs required.



Tabs are useful if you are cutting lots of small shapes that threaten to pop out and jam the cutting head or get damaged by fouling the router bit once they are cut. It is also useful if you are cutting thick materials such as aluminium and perspex.

Using this feature with a small tab width, i.e. 1 mm will leave a tiny tab of material holding the job together. When the job has finished, simply break the parts away and file the unfinished section.

When a tab or tabs are added to the job the Tab Width field on the Group Settings for Machine Control dialogue box will be updated. For example if the Tab Width field displays 2(1.000) this means that there a 2 tabs of 1mm on each shape of the job.

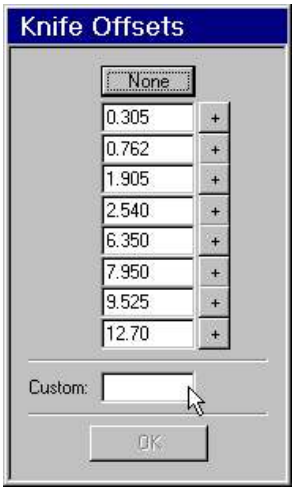
You can visually check the Tabs you have added to the job in the output screen. Each break in the job represents a tab location.

Tabs are placed evenly around the perimeter of the shape commencing from the Start Point of the shape. If the tab placement does not seem correct then you can re-position the tabs by moving the Start Point of the shape. Refer to page 51 for more information about Start Points.

NOTE: *If you attempt to set your machining path to reverse whilst you have multiple tabs set you may encounter irregular machine behaviour where the machining path direction may be overridden. You will be able to see any inconsistencies in the output screen prior to transmitting the job to the machine. It is recommended that you do not set your machining path to reverse when using multiple tabs.*

Knife Offset

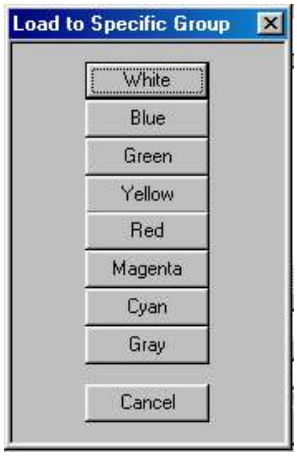
The most commonly used knife is a swivel or trailing knife. The point of the knife is slightly offset from the centerline of the cut. This will normally round the corners slightly. To eliminate this problem the knife offset must be set appropriately and the program will adjust the tool path for the offset of the knife. Standard Roland Digital knives have an offset of **0.25 mm**. Most Mimaki knives have an offset of **0.3 mm** You can also enter a custom Knife Offset if you wish.



Moving or Copying Groups

To move a shape from one group to another select **Move to Group** from the above menu bar. A new menu bar will appear:

With your mouse click to select the desired shape or shapes that you wish to assign to a different group or groups. You can also click and drag a bounding box around the desired shapes using the mouse. As you pick the shapes they will be highlighted in bright white. Once all the desired shapes have been picked, select **to Group** from the menu bar. The Group Colour dialogue box will appear:



Use your mouse to select a group by its colour. Once the group has been picked the dialogue box will disappear and the selected shapes will now show the corresponding colour of their new group. The menu bar will return to the **Group Menu.** You may divide up the shapes in your design into any combination of groups.

Copying to groups works exactly the same way except that the shape will be copied to instead of moved to the chosen group. This will effectively give you the same shape but in two different groups. You would use this in cases where you wish to engrave a step that has one side as the perimeter of your job (such as the recess for the glass in a picture frame.

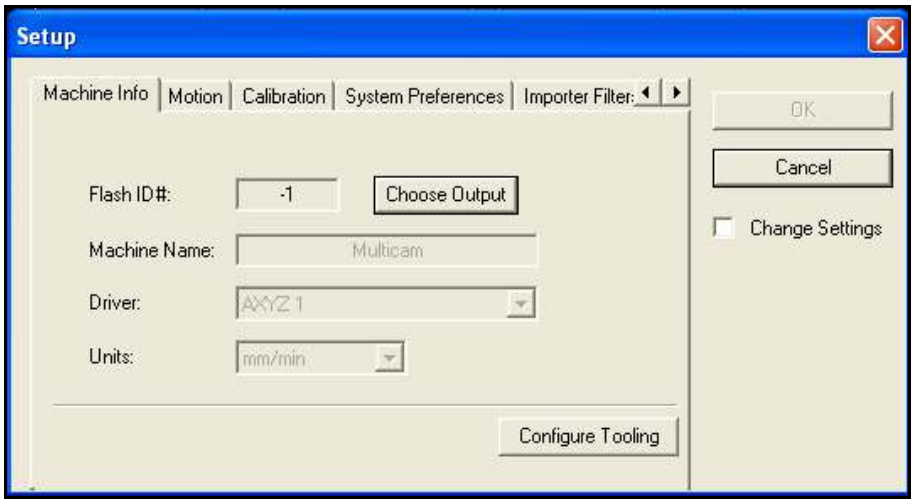
Clicking on **Return** will close the Copy/Move to groups menu and return you to the Group Control Menu.

Once you have finished setting up the group controls for your design click on Main Menu to return to the Main Menu. You will notice that if your job make use of different groups these groups will be shown in the **Output** menu of ToolPath.

SETUP

Setup is used to set up the standard parameters for your Multicam machine and the ToolPath software

ToolPath is designed so that you may operate several Multicam machines from a single PC. Click on **Setup** to open the Setup dialogue box.



The dialogue box has a number of tabs that are used to enter information and settings for the Multicam machine and ToolPath software. Click on each of the tabs to bring them to the front so that you can make any changes. These tabs are labelled **Machine Info, Motion, Calibration, System Preferences, Importer Filters** and **Wizards**.

In order to allow changes you must first click on the Change Settings tick box.

Once you have made changes in the Setup dialogue box and the dialogue box is closed you will be given the opportunity to update the machine controller. This will transmit the new settings to the currently selected machine and also store them on your hard drive. If your machine is not online, the changes will be saved to the hard drive and transmitted to your machine next time you establish communication.

Changes to the Setup of your machine must not be made whilst your machine is running. If you are in doubt as to the effect changing a setting will have, leave it alone!

If you already have jobs present at the machine changing the machine settings *will not* alter the changes that were transmitted with those jobs. For the new changes to affect jobs at the machine you must re-transmit the jobs to the machine.

The following pages describe in detail each of the Setup Tabs.

Machine Info

The first tab is the Machine Info tab. It contains basic information the Multicam machine currently active in ToolPath

Flash ID# This is the serial number of the machine and is used to differentiate between different machines in a multiple machine setup (ie if you have more than one machine connected to your computer). In these circumstances you may choose a different machine to the one currently active by clicking on the **Choose Output** button and selecting a different machine from the list of available machines.

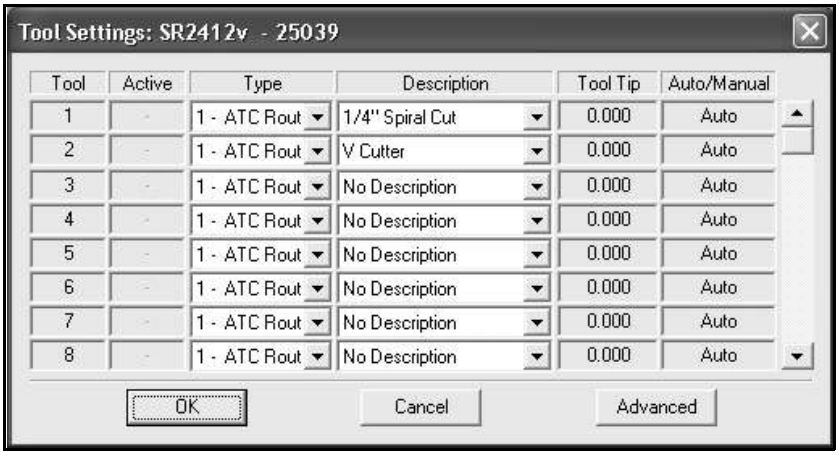
Machine Name You can designate a more descriptive name for your machine rather than relying on the Flash ID# to identify machines.

Drives Select the driver type for your model machine. This is usually taken care of during installation of your machine. If you change the drive type for you machine you may find that it will not run correctly, if at all.

Units Choose your preferred units to use. You may choose between metric (mm or cm) and imperial (inches) measurements. It also allows one to choose whether speeds are measured per minute, or per second for both metric and imperial.

Tool Settings

With the **Configure Tooling** button a dialogue box is activated where you can set up the tooling for your machine. This is mainly of use to those machines fitted with multiple heads, quick tool change motors or automatic tool changers. You only have access to these functions if your machine is communicating with the host computer.



Selecting the Configure Tooling button on the bottom right corner of the Machine Info tab, you can set up the number and type of tools your machine is equipped with. These settings allow you to set a different tool for different groups in your job. You can configure up to 16 tools mapped at up to 4 physical tool positions. A physical tool position corresponds to an actual tool on your router table (i.e. Router, Drill, Automatic Tool Changer (ATC), etc).

When the Tooling Settings Dialogue is brought up, it will attempt to read the current bit descriptions stored at the machine. If a bit description at the machine is not included in the computer's list of bit descriptions, you will be prompted to add the description to the list, or ignore the description. See **Advanced Tooling** for more information on how to add/remove bit descriptions.

Tool Column
The first column lists the tools that can be assigned in ToolPath. This provides a logical way of assigning multiple tools for use in a job. If you have an ATC, you will want to setup 8 of your most commonly used cutting bits for automatic tool changing. If you infrequently use additional bits then you can setup 8 additional MANUAL tools. So, when a job calls for a Tool 9, which is a Manual tool change, simply replace the current bit in the router with the tool specified for Tool 9.

Active Column
Indicates which tool is currently active for a given Group. This column is only enabled when you click on the Tool field in the Group Options dialogue, See Groups.

Type Column
Assign one of the 4 physical tool types to the current tool number.

Description Column
Select on of the pre-defined tool bit description to associate with the current Tool number. See Advanced Tooling for more information on how to add tool descriptions.

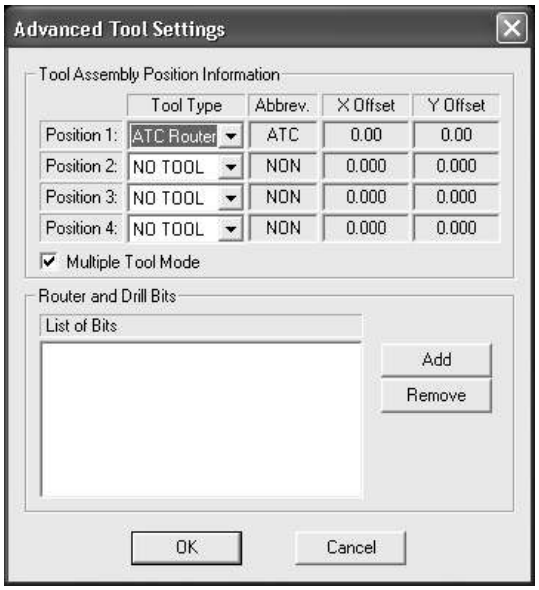
Tool Tip
These values are obtained from the table and indicate the setup value for the Tool Tip. Tip values are set when a Function 25 is performed at the sub-console.

Auto / Manual
When using the ATC, this indicates if the given tool will be automatically changed when needed in a job, or requires user intervention (manual change) during a job. Set up your most commonly used tool bits as automatic changes and less frequently used bits as manual changes.

Advanced Tooling Settings

Pressing the "**Advanced Tooling**" button brings up the Advanced Tooling setup dialogue

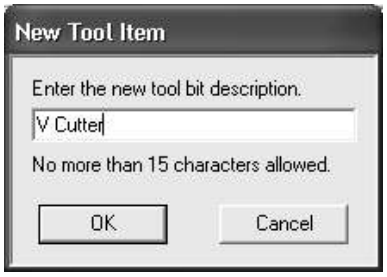
.



Two major pieces of information can be configured here.

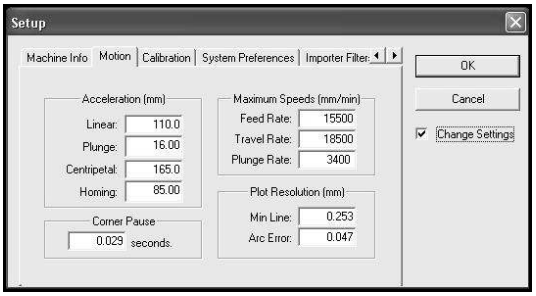
Tool Assembly Position Information
First, you can adjust the type of tool available at one of the physical positions given in the top portion of the dialogue box. As you can see in the above dialogue, Position 1 is set to a ATC. Machines without an ATC can have multiple tool assemblies (such as dual head machines): if your machine does not have an ATC, ensure that the "Multiple Tool Mode" check box is checked. If you have an ATC, then turn off the check mark. Tables with ATC and only have one possible tool, the ATC itself.

Router and Drill Bits
The second portion of the Advanced Tooling dialogue allows you to add a list of descriptions for various drill and router bits. Add the names and sizes of the bits you use in your tool array. Setting the description facilitates the ease of setup for the ATC and organizing your shop tools. Click "Add" to describe a new bit (dialogue shown to the right). Highlight an existing bit in the list and press "Remove" to delete the description from the list.



For further explanation of multiple tools and their setup and use refer to Appendix F commencing on page 161 and Appendix G commencing on page 165.

Motion



The motion tab is used to enter information that controls the motion of the machine. It is recommended that you do not change these settings without first consulting your Multicam technician. Incorrect settings will cause the machine to act in an unpredictable manner and can lead to poor cutting results.

Motion control is broken into four main areas, Acceleration, Plot Resolution, Maximum Speeds and Corner Pause.

Acceleration:

Linear: This is the distance that it will take the machine X and Y axes to go from a standing start to full speed.

Plunge: Sets the distance over which the Z axis accelerates to its programmed speed. Setting too small an acceleration distance may cause the Z axis to stall. Setting too large an acceleration distance means that the Z axis will take a long time reach its programmed speed.

Centripetal Acceleration: This value determines the speed of the machine around a **tight radius**. The set value is the minimum radius at which the machine will go around a line at the Maximum Speed. For example if the value is set to 25mm the machine will allow itself to go around that radius at full speed. If the radius is 12mm it will go around the curve at one half linear max speed or the set feed speed, whichever is less. A good setting to start with is 8mm for large machines and 12mm for smaller ones. This will also depend on the size of machine you have. If your machine is wobbling around curves increase the centripetal acceleration, if you think it is too sluggish decrease this value.

Homing: This setting tells the machine to start decelerating toward the homing sensors (X, Y and Z axes) when it is the programmed distance away from the sensor.

Maximum Speeds:

Feed Rate: This is the maximum speed at which the machine will actually cut and is usually set lower than the Travel Rate. The value you set here is the default value that will appear in your Output Menu and your Groups Menu. Set a value of about 10000mm/ per minute

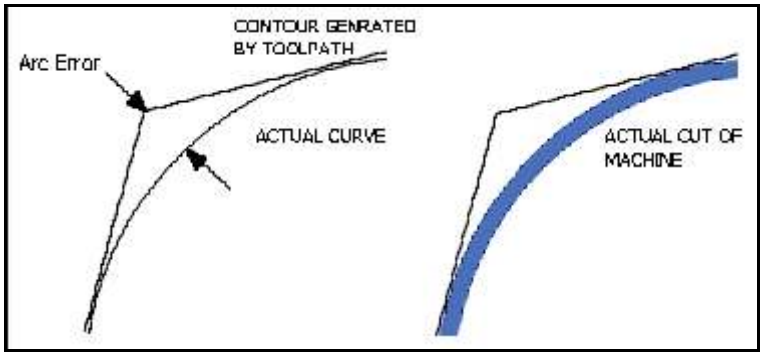
Travel Rate: This is the maximum speed you will allow the machine to go when it is not cutting (e.g when jogging or moving between shapes when cutting a job). Normally you set this value as high as possible. There are two limiting factors: The speed of the controller and the mechanical limitations of the machine. Generally it will be the mechanical limitation of the machine that will determine the setting. Start out with a value of about 12000mm/ per minute and adjust from there as you use your machine. If your machine is acting erratically and stalling easily, you must reduce this setting.

Plunge Rate: This is the same as the linear max speed except in the vertical direction. Normally you do not wish this to be nearly as fast as the linear max speed as you will destroy a lot of router bits by slamming them into your material too fast. Generally a setting of **1800mm/ per minute** is desirable.

Plot Resolution:
Plot Resolution allows the machine to decide what is a line or line segment, and what is actually part of a curve.

Min Line: This controls the shortest line that the program will recognize as an independent line. This should be set to about 0.25 mm.

Arc Error: This controls how ToolPath distinguishes corners from curves. It is usually set to 0.051 mm. This is an important setting and must be considered carefully.



ToolPath approximates a complex curve with a series of straight lines as shown on the left. The contour generated is determined by the minimum line length and arc error settings (i.e. The maximum deviation from the actual curve is the arc error). The smaller the arc error the more contour points will be required to approximate the curve, larger arc errors will generate few points and smaller files. Once the machine runs the contour is fitted with a spline curve as shown on the right, this will recover your curve once you do the actual cut.

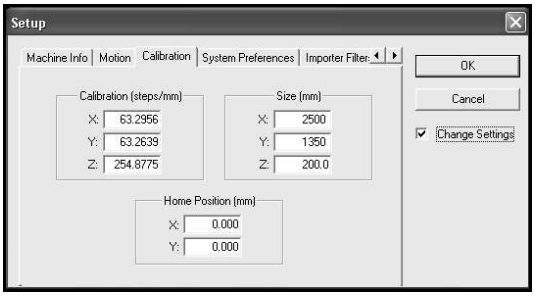
Note: Altering the arc error will NOT result in smoother curves. Your curves will be smooth regardless. The arc error is simply a measure of the maximum deviation between the true curve and what is actually transmitted to the machine. The actual deviation from true curve is about 20% of the arc error. Normally the arc error is set to 0.051 mm. Smaller values will tighten the maximum deviation and increase file size, while larger values will increase maximum deviation and decrease file size. It is also important to note that the machine itself can only process a single curve up to a maximum of 500 points. Any curve larger than 500 points will be split into two curves, this will cause the machine to take a momentary pause between the two curves. This pause may create a problem with edge finish at that point. The solution is to increase your arc error, reducing the number of points forming a complex curve. Remember this will not alter your smoothness, only your arc deviation. In most applications it is almost impossible to tell the difference in accuracy between a 0.05mm and 0.1mm arc error. Now that we have made a simple concept like resolution complicated we can give you a simple guideline:

Resolution is equal to the lesser of:
1. Step size setting in the calibration.
2. Arc Error divided by 5.

Corner Pause:
This is a momentary pause that the machine will do at each sharp corner. If you set this too short your machine will wobble at the corners as it physically cannot change direction fast enough. Too long and the machine will seem sluggish. A setting of about 0.040 seconds is normally adequate.

Calibration

This tab is used to store information about the accuracy of your machine, process area of your machine and you may even set a home position for your machine (as measured from the absolute lower left corner of the machine table top).



Calibration:
Machine Calibration is the designated number of steps the motors on your machine need to take to move the machine one mm or cm. It is basically what ensures the accuracy of your machine when cutting shapes. The settings applied when your machine was installed are quite accurate, however over a period of time it may be necessary to re-calibrate your machine. Refer to page 27 for details on how to do this.

You can actually calculate the calibration in steps/ mm if you know the mechanical gearing of your system. There are two parts to the calibration calculation: First, the number of steps per rotation of the motor. This can range from 200 to over 50,000 depending on the setting of the drive. Usually there are DIP switches on the drive that sets these values. You should never change these settings. Secondly the number of rotations of the motor per unit length of motion. For example if we have the drive set to 800 steps/revolution and have geared the machine to 2 revolutions per mm the calibration is:

800 steps/rev x 2 revs/mm = 1600 steps/mm

We can set the calibration of an axis at an arbitrary value, plot a line of a known size, then measure the actual length of the line. Use the following formulae:

New calibration = Old calibration x correct line length / actual line length

Machine Size:
This is the actual process area of your machine (limits of travel) not the physical size of the machine. Simply enter your table's size in all three axes. These settings are pre set for you during installation and should never need to be changed.

Home Position:
You can set a position on your machine that the controller will consider as the Home Position. This is the point to where your machine will home to if you perform a **Function 12** at the machine. Setting a home position that is not the same as the absolute lower left corner of your machine may be useful for jigging. The distance is measured from the point where the machine homes to when you perform a Function 12 when the values for X and Y are set to 0.

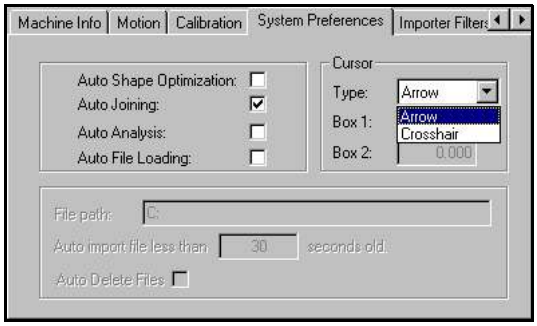
Refer to Appendix B on how to set a Datum position using the **Home Position** and **Function 12**.

Be aware that entering amounts in this section will reduce the process area in the X an Y axes by these amounts.

System Preferences

The System Preferences tab allows you to enable or disable automatic functions in ToolPath by clicking in the relevant check box next to the feature you wish to activate.

These functions only affect the way that ToolPath interacts with your files as they are imported or with the programmer.



Auto Shape Optimization
Enabling this will cause ToolPath to look at your file as you are importing it and automatically sequence it in such a way as minimise the travel and cutting distances when the job is being processed. This may be OK for simple jobs but may not be desirable where more complicated jobs are involved and you need to cut your job in a particular manner to suit your material.

Auto Joining
Some design packages do not correctly close shapes (such as boxes or triangles) when they are drawn, or if you have drawn a polygon manually you may not have properly closed the start and end points of the shape. Auto Joining will cause ToolPath to automatically close these open shapes. The distance that this will be effective over (the distance between the start and end points of the polygon) is set in the value for **Min Line Length** which is in the **Motion Tab.**

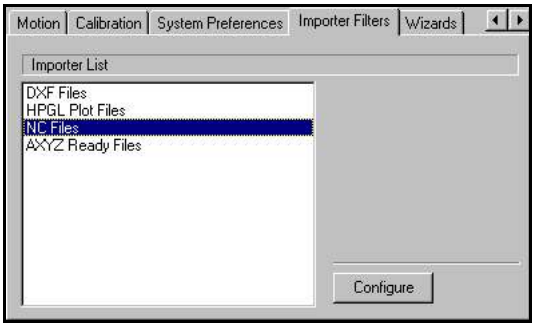
Auto Analysis
Enabling Auto Analysis will cause ToolPath to analyse the file as it is being imported and automatically correct and direction, sequence and duplicate shape errors.

Auto File Loading
You can have ToolPath automatically load a file when you start it by enabling this function. Once enabled you will need to specify the location of the file on your hard drive including the file name. This feature may be useful if your company makes many one off jobs and does not wish to save them all. You can give these jobs the same name as you export them (such as temp.dxf) and have ToolPath load this job on open. Checking the Auto Delete Files box will automatically delete the file from your hard drive. It is also useful if you have a design program that can save a compatible file and then automatically start Toolpath. If the file name is specified in the Auto File Loading field then Toolpath will open with this file on the screen.

Auto File Loading can also be used in conjunction with other command line parameters as described in the section on File commencing on page 38

Cursor
If you don't like the standard ToolPath cursor you can change it to a crosshair type with pick boxes. You may also specify the size of the pick boxes by entering values Box 1 and Box 2.

Importer Filters



The Importer Filters tab allows you to specify parameters for the various types of files that ToolPath will accept.

For each of the Importer Filters (DXF, PLT, NC, PCD, and RDY) you can define Resolution (the units in which the files are imported), File Extensions (valid extensions for this file type) and Tool Mapping (which tool in the file is assigned to which group).

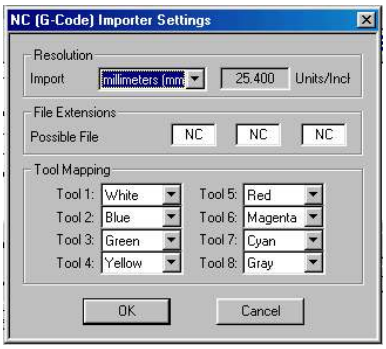
Some of these parameters may not be able to be defined for some file types.

If you use a design package that allows you to design in millimetres but will only export files in inches, you can set the ToolPath import filter Resolution to inches. ToolPath will then display the job in millimetres once it has been loaded.

Configure

Some of the Importer Filters allow you to assign Tool Mapping. This can be handy if your design package will allow the export of tool information with their files. ToolPath can be told to automatically assign the coloured layers to specific tools as it loads the file.

This can result in considerable time saving when setting up complex jobs.



QUICK TOOLPATH HOW-TO

This is a simple step by step overview of how to bring your job file into **ToolPath**, and how to manipulate it once you get it there.

Step 1 - Starting ToolPath

Click on the ToolPath Icon on your Windows desktop or go to the **Start** button and select **Programs|ToolPath|ToolPath for Windows**

Step 2 - Importing Your File

After ToolPath has been started, the next screen you will come up to is the **File Manager** screen. From this menu you can import a .DXF, .PLT, .NC, or .RDY file into the program. Included on your **ToolPath** diskette was a sample .DXF file that was copied to your hard drive during installation.

Go to your TPWin|Samples directory and select the sample DXF file by double clicking on it, or by highlighting it and clicking **Open**.

If you wanted to see which HPGL files were in a particular directory, you would change the file type in the **Files Of Type** section of the dialogue box and selecting **.PLT** from the drop down list. You can see from this menu that you could also see **.NC**, **.RDY**, or you could select **All** and see all files on a disk, or in a directory.

If you do not have a communications device plugged into your computer you cannot output any jobs. All other functions work as they are supposed to.

Step 3 - It's here, what do I do?

What you should now see on the screen is the sample design, along with the Main Menu across the top of the screen.

You can see the actual size of the design by looking at the lower left corner of the status bar. The display will show the overall size of the job as well as the number of shapes contained within it. If you click on an individual shape the size for that shape will be displayed.

The next step is to click on **Analysis** from the Main Menu. This will allow **ToolPath** to correct any errors in cut sequencing and / or in cutting direction. You can then see how far the cutter is going to have to travel, and the estimated time for completion.

Click on **OK** to get back to the Main Menu.

After clicking **OK** to exit **Analysis**, you may notice that some of the shapes will have changed colour from blue to red, or vice versa. This is because **ToolPath** decided that they should be cut in the opposite direction. If you would like to see the actual order the shapes will be cut, select **Edit** from the Main Menu.

By clicking on **Next** and **Previous** in the **Edit** menu, you can see the order of cutting by which shape is highlighted. Notice that inside shapes are cut first, before outside shapes. If the outside shape was cut first, nothing would be able to hold the inside from moving when it was being cut.

You will see a small yellow "L" shaped icon hanging off each shape as it is highlighted. This is the directional key. It tells you, by it's placement, the direction of the cut (clockwise or counter-clockwise), and the side of the cut that the cutter compensation will be justified to (this is the side the tail end hangs on). It also designates the start point for each cut.

If you wish to change the start point for any of the shapes, select **Start** from the **Edit** menu. You can increment around the highlighted shape by clicking next or previous, or you can manually pick a location by choosing the location you want the start to be, and clicking the left mouse button. Most simple jobs will not require the operator to change the start points of the shapes. **ToolPath** is pretty good about setting good start points.

If you wish to make any other changes, such as 2D or 3D engraving, do them now by following the directions set out in the **ToolPath** reference section on **Engraving**.

Step 4 - Output to Your Machine

At this point, redo the **Analysis**. Click on any direction or sequence errors to correct them. The correct tool paths are now set, and we are ready to output to our machine. If you are running **ToolPath** without a communications device, you will not be able to output. If you are plugged in and ready to go, the **Output** menu will appear.

Make sure you set all the appropriate values in the output menu. The main one in particular is compensation. Without setting this value, the job will be sent to the table with no compensations. This seems rather obvious, but what will occur is that the actual size of the pieces cut out will be off by the size of the router bit. This could result in a real waste of material, as well as time and energy on your part. Set the Compensation to half the diameter of your router bit.

Diameter of Cutter	Compensation
1.0mm	0.5mm
2.0mm	1.0mm
3.0mm	1.5mm
6.0mm	3.0mm
12.0mm	6.0mm

In the Job Name section, you will see the name of the file that you are sending to the machine. This will be the name that appears on the Sub Console. If you wish to change it, simply click on it and type in a new name.

If you are happy with how you have set up this cutting job, and expect to need to cut it again in the future, you can now click on **Save Ready File**. This will bring you back to the file manager screen. You can then select the directory you wish to save your router ready file in. You can simply click OK if saving it to your current directory is OK. The filename of the ready file will be whatever the Job Name was that you selected, with the suffix **.RDY**.

You can now select **Send** and your job will redraw itself, taking into account compensations, tabs, and whatever other changes you have made in ToolPath. If you have not already sent the operating system to the machine, you will see that the operating system is now being transmitted. Your machine will not run without an operating system. You will also be informed as to how large your job file is, as well as how much available memory is remaining at the machine controller. If everything meets with your approval, simply select **Transmit**, and the job is on it's way.

Job Recovery

The Multicam CNC controller incorporates features which enable the operator to recover the exact start position of a job in the event of sudden power loss or if the machine loses position for some other area. In order for this feature to work the operator must do two things;

- When the machine is first powered up a Function 12 (refer to page 115 for more information on Function 12) must be executed, this will enable "soft limits" as described on page 120 as well as enable the job recovery features.
- Before a job is started a Function 3 (refer to page 112 for more information on Function 3) must be performed which records the job start position in relation to the Function 12.

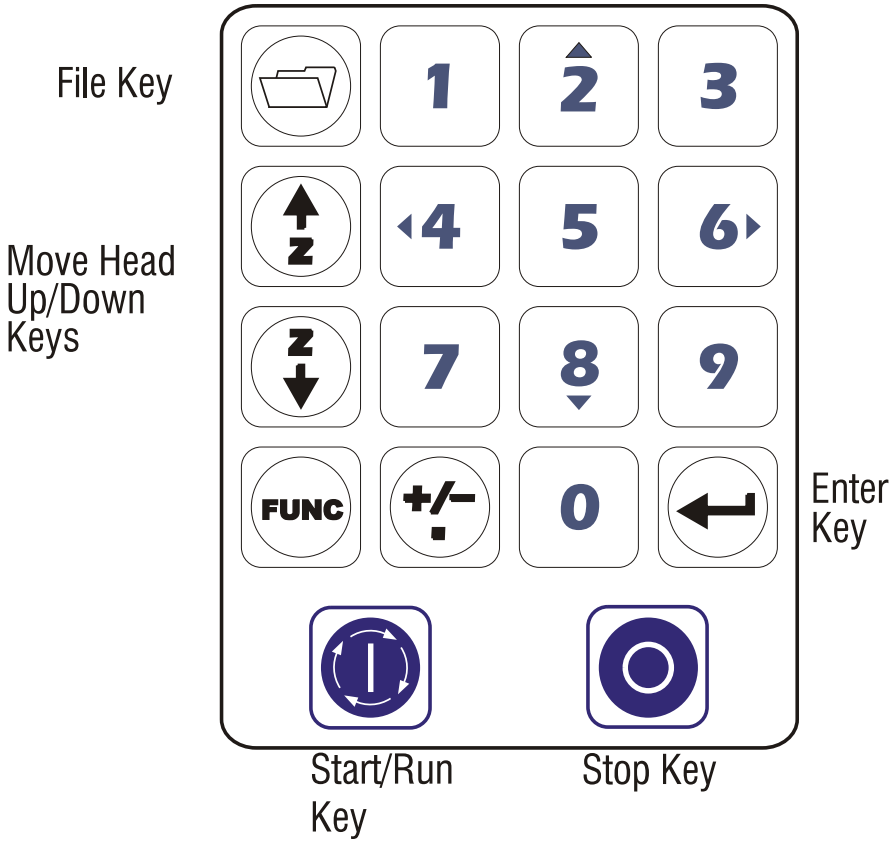
If a job fails and loses position for any reason the original job start position can be easily recovered by following the steps below;

- Perform a Function 12. The machine will seek the soft limits.
- Perform a Function 13. The machine will move to the position stored by the Function 3 setting. Refer to page 116 for more information on Function 13.

The job can now be re-run in its' entirety or you can start the job from some other point by commencing from a different shape number as described on page 143.

MULTICAM MACHINE OPERATION

The sub console is the black box attached to your Multicam machine. It is really the lifeline between the operator and the **Multicam Controller**. Treat the Sub Console with respect. It may appear tough but it contains a number of small electronic parts which may malfunction if the Sub Console is abused. It has a 40 character display and a 18 button membrane keypad. The display prompts you for operation commands. The keypad appears as follows:



NOTE: If your display reads:

Error 92 No O/S Installed

then no operating system has been downloaded to your machine. You must first download the operating system from **ToolPath** to your machine. Once the operating system is successfully downloaded the LCD will display the following message:

Set X, Y Origins? YES

Press the [ENTER] key and the machine will proceed to find its limit sensors.. The display will then show the following message until you send a job from **ToolPath**.

AMC 3D SERIAL NO. XXXXX

Sub Console Key Board Operation

Most of the keys on the Sub-console have more than one function depending on what you happen to be doing at the time:

Numeric Keys: These are the ten keys marked **0** through **9**. Note the small blue arrows on the **2**, **4**, **6**, and **8** keys. These keys are used to move in the X and Y axes. The machine head will jog, left/right, or diagonally etc. depending on which key is pressed. If you are in a function that calls for a numeric entry these "jog" keys become number keys as marked.

The [FILE] key. Will select the next file resident in the controller and prepare it to run on the machine. Pressing [FUNC] then [FILE] will allow you to enter a job *number* to find. This is useful if your jobs are transmitted as numbers instead of names. Hold the [FILE] key down to have the files automatically displayed one after another. Release the [FILE] key when the desired job appears. If you scroll past the job you want press the [+/-] key within 5 seconds then press the [FILE] key and the files will reverse the direction of the job scrolling moving back to the previously displayed job. The display will always indicate which job is ready to go.

The [START/RUN] key is used to start the job that is indicated on the display. To start a job you must press [START/RUN] twice for two dimensional jobs, three times for three dimensional jobs.

The [STOP] key is straight forward. Anytime it is pressed the machine will stop! If you are running a job, the machine will stop and the file will go into pause mode and await further instructions.

The [FUNC] key allows you to access the various functions. This key must always be followed by a numeric entry reflecting the desired function number. The only time this key does anything else is if we are in **NC mode** and using function 3 or function 4. Pressing the [FUNC] key will allow the numerical entry of a Z position.

The [+/-] key has quite a few uses: If we are making a numeric entry and press [+/-] before entering any numbers it will switch the sign of the numeric entry. If it comes after a numeric key has been pressed it will place a decimal point. For example:

[+/-] 0 [+/-] 5 [ENTER]

Will enter the value **-0.5**. If we are in the start sequence of a job file the [+/-] key will move the job start from one shape to the next. See the section entitled: Starting a Job Part Way Through a File. The [+/-] key is also used as a toggle switch in functions that require a selection of a mode. For example function 10 sets the job to run forward or reverse, use the [+/-] key to toggle between these choices. Finally the [+/-] key is used to select FAST, MEDIUM, SLOW and INCREMENT JOG while the controller is in jog mode. On dual headed machines this key will alternate between active heads.

Using the [+/-] key within 5 seconds of using the [FILE] key will enable you to step back through the list of files in machine memory.

Finally the [ENTER] key also has several uses depending on when and where you are using it:

- If you are making a numeric entry within a function it will terminate and store the entry, similar to the way a calculator works.
- If the display is showing a job ready, i.e. **job TEST ready**, pressing the [ENTER] key will display the job's size. Pressing the [ENTER] key again will once again display the job name.
- If the display is showing the X, Y, and Z co-ordinates the [ENTER] key will allow you to enter a co-ordinate that you wish the machine to move to.
- Finally if you are within a start sequence of a job, the [ENTER] key will allow you to enter the shape number that you wish to start with. NOTE: This function is designed for use after the first shape has been started. You must not start a job then seek a shape number until shape 1has been started.

Sub Console Function List

There many functions available to the operator through the **Sub Console**. The following is a list of the more common functions, note that most functions are valid while a job is running:

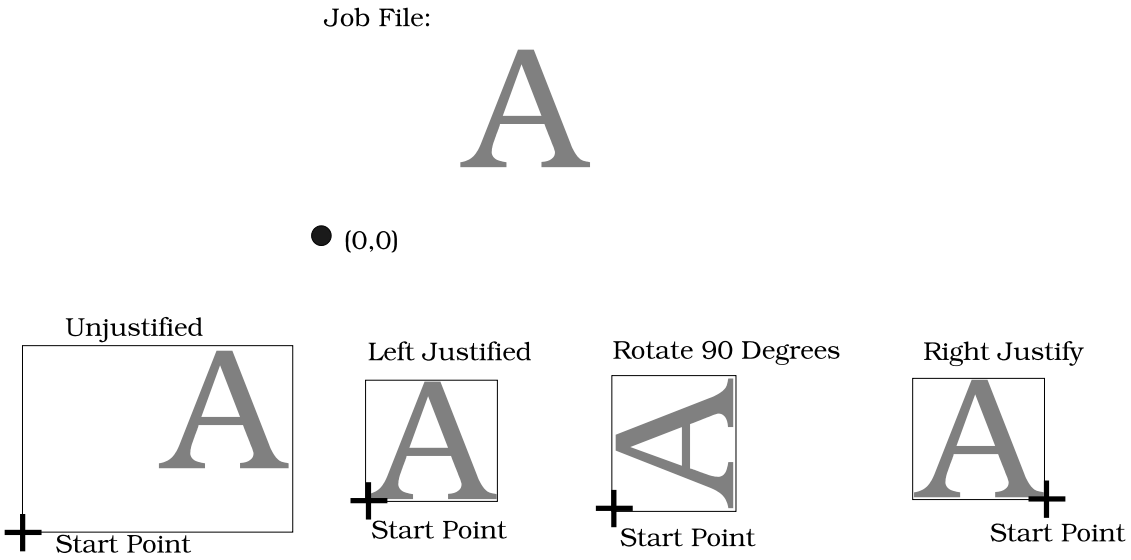
Function	Without Job Running	While Job Running
0	Justification (Left, Right, None, Rotate 90°)	No
1	Feed Speed Setting	Yes
2	Spindle Control (Auto/On/Off)	Yes
3	Reset X, Y and Z origins	No
4	Lift Top and Lift Bottom Setting	Yes
5	Activate Appliances	No
6	Measured Move	No
7	Adjust Lift Bottom	Yes
8	Set Material Surface/Job Datum 0	Yes
9	Abort running job	Yes
10	Set Cutting Direction forward/reverse	Yes
11	Set job mirror (None, X mirror Y mirror)	No
12	Seek absolute origin	No
13	Go to Job origin	Yes
14	Set up a Job Array	No
15	Set Plunge Feed Rate/Peck Retract	Yes/No
16	Set Job Finishing Position	No
17	Run a measured box	No
18	Machine Parameter Settings	No
19	Toggle between NC and normal display mode	No
20	Adjust pause time before a shape cut	Yes
21	Set up job repeats	No
22	Digitise a job	No
23	View job settings	Yes
24	Disable machines' soft limits	Yes
25	Set tool tip offsets (tool change only)	No
26	Reset all tool tips (tool change only)	No
27	Job process time (min:sec)	Yes
28	Tool bit compensation adjustment	Yes
29	Show Image File Version	Yes
31	Select Individual Tool for Single Operation (ATC only)	No
32	Enable/Disable Second Carriage	No
33	Unload Tool from spindle (ATC only)	No
35	Save settings for functions 1, 4, 8, 15 and 84	No
36	Recall function 35 setting	No
38	Set up Material Off Loader Arm (Scraper Arm)	No
39	Activate Material Off Loader Arm (Scraper Arm)	No
61 - 64	Preset Measured Moves	No
91	Delete Job file	No
92	Erase all files in memory	No
121 - 124	Preset home positions	No
159	Erase operating system from AMC	No
181	Seek Sensor and Plunge acceleration distance	Yes
330	Set up "S" mouse	No
350	Set up/Bypass automatic tool change	No
352	Tool change automatic tip sensing	No
370	Activate Tangential Knife Mode	No
FILE KEY	Search for a job in memory.	No

There are some basics that need to be covered first before delving too deeply into the various function commands of the Sub Console. The first is the two different operating modes that are available. These are **Display Mode** and **NC Mode**.

Sub Console Functions

Function 0 - Justification

There are four possible justification settings; unjustified, left, right, and rotate 90 degrees. Once you have selected function 0, push the +/- key to sequence through the selections. Whatever selection you stop at will be the one executed. There is no need to push the [ENTER] key.



Unjustified This means that the starting position of the cutting head is the (0,0) co-ordinate of the job.

Left Justify The cutting head starts at the lower left position of the job.

Right Justify Opposite to left justify - the cutter head starts on the right edge of the job.

Rotate 90 Selecting this mode will rotate your job 90 degrees counterclockwise.

Function 1 - Feed Speed

The feed speed is altered by using function 1:

[FUNC] 1 [ENTER]

The display will show: **SPEED 10000 mm/min**

You can alter this value to anything you wish by using the numeric keys to type in a new value and then pressing [ENTER].

NOTE: *Depending on the program you used to generate the NC file in the first place or the speed controls you may have used in ToolPath you may have feed speed information in the file itself. The controller will always compare the programmed speed in the file to the feed speed setting in Function 1 and use the* ***SLOWER*** *of the two. If there is no feed speed information in the NC file the speed used will be the Function 1 value.*

The feed speed can be altered using Function 1 at any time. You can pause a job part way through its run, change the feed speed and resume at the new speed. This feature is extremely convenient as you may adjust the feed speeds as you go depending on how well the job is progressing.

Function 2 - Spindle Control

With this function you can set the machine controller to either automatically activate the spindle when a job is commenced (**AUTO** setting), have the spindle permanently on (**ON** setting), or have the spindle permanently off (**OFF** setting)

If the operator selects **ON** the spindle will turn on as soon as [ENTER] is pressed and will stay on until one of the following conditions occurs;

- The operator has pressed [STOP]
- Enter [FUNC] 2 and select either **AUTO** or **OFF**
- The machine has started and ended a machine cycle. The system automatically stops the spindle at the end of a cycle and defaults the spindle to **AUTO**
- Emergency stop is pressed.

If the function 2 setting has been set to **OFF** the spindle will never come on until function 2 has been reset to **ON** or **AUTO**. If the operator attempts to start a job or enters function 6 or function 17 with function 2 set to **OFF** the sub console will prompt the operator to set function 2 to **AUTO**. The operator can choose to either set the function 2 to **AUTO** or leave it **OFF**, in which case the job will run with the spindle off.

AUTO is the default mode. The machine controller decides when the spindle should be on or off and will only activate the spindle during a machine cycle, i.e running a job, performing a manual move (function 6) or cutting a rectangle (function 17). The spindle is always turned off after a cycle is completed.

If you set the spindle to **ON** using function 2 you can stop the spindle by pressing the [STOP] key on the keypad. The spindle will stop and the function will be re-set to **AUTO**

Function 3 - Resetting the X, Y and Z Origins

To set the machine origins back to 0,0 or to where the head is positioned enter Function 3:

[FUNC] 3 [ENTER]

The X and Y co-ordinates will automatically be set to (0,0). It will also set the Material Surface to machine top.

If you are in NC mode: The LCD readout will show:

X = +/- 0.000

You are able at this point to enter a new co-ordinate value for X or you can press Enter to accept this as the X origin. Once you press enter, you are asked the same thing for Y, and then the same thing for Z.

Once Function 3 is completed the tool head will return to the machine top position.

Function 4 - Setting Lift Top and Lift Bottom (Material Thickness)

Setting a function 4 is very important. This function controls the depth to which the machine will cut.

NOTE: If your machine is fitted with an Automatic Tool Changer (ATC) please also refer to the following page.

The **lift top** setting is the height above your material surface the tool tip will lift when moving between shapes. It is normally set 5-10mm above the material surface. You must ensure that this setting will also clear any clamps or jig fixtures.

The **lift bottom** position will limit how deep a program can drive the tool head. You should set it so that it is higher than the machine bed.

The lift bottom position has slightly different purposes in two dimensional and three dimensional jobs.

Two Dimensional. Two dimensional files do not include any Z dimensions so the lift bottom position will become the cutting depth. The lift top position is the travel height between cuts.

Three Dimensional. Three dimensional files do specify Z dimensions so the lift bottom position becomes a safety level. Regardless of how the file wishes to drive the tool head it will not move lower than the lift bottom position. The lift top position is still used as the travel height between cuts.

To set the material surface, lift top and bottom use Function 4:

[FUNC] 4 [ENTER]

The display will show:

Set Surface: +0.000

Using the [↓] key jog the head to the material surface and press [ENTER]. Generally this position is the surface of the material to be cut, however in 3D engraving you may wish to set this slightly above the material surface (say 0.5mm) so that the tool fully exits the material when lifting out of the cut.

If you are in NC mode: You can type in a position by pressing [FUNC] then typing in a numeric value and pressing [ENTER].

Once you have entered the material surface position you will be prompted to enter the lift bottom. This is generally the thickness of the material *plus* an allowance for the tool to go through the material (say 0.5mm):

Lift Bottom: -0.000

Type in a value for the material thickness plus a cut through allowance (for 10mm thick material you would enter 10.5) and press the [ENTER] key.

The final prompt will ask you to set the lift top setting;

Lift Top: +0.000

This is the distance above the material that the tool tip will move to when travelling between shapes in the job. It is normally set 5-10mm above the material surface. Type in the desired clearance amount and then press [ENTER]. You must ensure that this setting will also clear any clamps or jig fixtures.

Once the settings have been entered the head will return to the machine top position. Refer to page 135 for more detailed information on Z settings.

Function 4 and Automatic Tool Change Machines

On machines fitted with Automatic Tool Change units the function 4 setting is performed using TOOL 1. If tool 1 is not loaded in to the spindle when you perform the function 4, the machine will first return the currently loaded tool to the tool changer and then pick up tool 1. It will then carry on as described below. On ATC machines the relative length of tools recorded during the function 352 is referenced from tool 1 in the ATC unit.

It is possible to perform a function 4 on individual tools by selecting the tool number using th +/- key during the function 4.

Let's say that you are in the middle of a job currently cutting with tool 3 and find that you need to change the tool due to wear.

After you have changed the tool perform the function 4 using the method below.

[FUNC] 4 [+/-] 3 [ENTER]

This will allow you to set your depth as described on the previous page using tool 3.

Function 5 - Appliance Activation

This function is used to enable Vacuum Hold Down, Monitor Vacuum and Dust Collector. Vacuum Hold Down and Dust Collector can be set to either ON, OFF or AUTO. Monitor Vacuum can be set to either OFF or ON

Activating **FUNC 5** will bring up the following options;

VACUUM HOLD: AUTO

this is the default setting. Press the [+/-] key to cycle between ON, OFF and AUTO Press the [ENTER] key to accept your selection. The ON setting will turn your vacuum table pump on immediately. OFF will turn it off immediately and AUTO means that the vacuum table pump will come on automatically when you start a job.

MONITOR VACUUM: NO

this is the default setting. Currently this option is NOT implemented. Press the [+/-] key to cycle between YES and NO. Press the [ENTER] key to accept your selection. Setting to YES will monitor the vacuum table pressure through a special monitoring device if one is fitted to your machine. If the pressure drops below a pre-set level that machine will stop and display an error message. Setting to NO disable vacuum monitoring.

DUST OFF DELAY: 5sec

set a delay before the dust collector turns off at the completion of a job. Use the number keys to enter a delay value then press the [ENTER] key on the keypad. The value you enter is stored on the machine memory. If you wish to stop the dust collector after a ob has completed before the time-out delay simply press the [STOP] key on the sub console unit.

DUST COLLECTOR: AUTO

this is the default setting. Press the [+/**-**] key to cycle between ON, OFF and AUTO. Press the [ENTER] key to accept your selection. The ON setting will turn your dust extractor on immediately. OFF will turn it off immediately and AUTO means that the dust extractor will come on automatically when you start a job.

Function 6 - Measured Moves

This allows you to jog the cutting head by precise distances. There are three steps to this function:

- Enter the X direction distance, press [ENTER]
- Enter the Y direction distance, press [ENTER].
- **Move with head up** will then appear.
  Press [ENTER]**.**

Function 7 - Adjust Lift Bottom

This function allows you to alter the lift bottom position by a positive or negative increment. If you require your lift bottom to be 1mm higher select function 7 and enter 1mm. It will adjust the lift bottom position 1mm higher. Negative values will move the position lower.

Note: Function 7 will ADD to your Function 4 Lift Bottom setting making it lower or higher depending upon whether you enter a positive or negative number.

Function 8 - Set Material Surface/Job Datum Zero

This function is used to preset material surface. In order to enter the material surface select [FUNC] 8, drive the Z axis down to touch material surface with tip of tool (using the [↓] key), press [ENTER] when correct. This will set the material surface for 3D or multi-depth 2D jobs. This will allow the operator to bypass the step of setting material surface as they start their jobs.

Function 9 - Abort a Job

If you are running a job and wish to abort it, first press [STOP] then select function 9. The job will be aborted, the head will lift to the topmost position and return to the job origin.

Function 10 - Set Cutting Direction

You may cut a job forward or reverse by selecting the desired direction with the [+/-] key. If you select forward all the shapes will cut exactly as specified by the ToolPath program. Reverse will reverse the cutting direction of every shape. Cutting direction can make a significant difference in cut quality which will be discussed in a later section again.

Function 11 - Mirroring

This is a handy function. You may select: **no mirroring**; **X mirroring**; or **Y mirroring** by pushing the [+/-] key. This results in the same type of mirroring that is attainable in **ToolPath**.

Function 12 - Seek Absolute Origin

Function 12 will cause the controller to seek the X, Y and Z limit sensors. This is the mechanical home position. Function 12 may also be used to seek a datum point. Refer to Appendix B for information on setting a datum point.

Function 13 - Go To Last Job Origin

Selecting this function will return the cutting head to the start position of the last job executed or cut. This Origin point was created when [START] was entered or function 3 was entered.

Function 14 - Setting a Job Array

This function allows you to set up a continuous cutting array of a specified number of columns and rows. Select function 14 and the present array setting will be displayed, probably 1 row and 1 column. Press [ENTER] and enter the number of rows you wish to cut. Press [ENTER] and enter the number of columns you wish to cut. Press [ENTER]. You will also be prompted to enter the distance between the rows and columns. This value should take into consideration tool diameter etc. Normally a distance of three times the tool diameter is a good value.

If you enter 3 rows and 4 columns the machine will repeat the job 12 times, cutting the desired pattern in 4 columns of 3 rows each.

Function 15 - Plunge Feed Rate/Peck Retract Settings

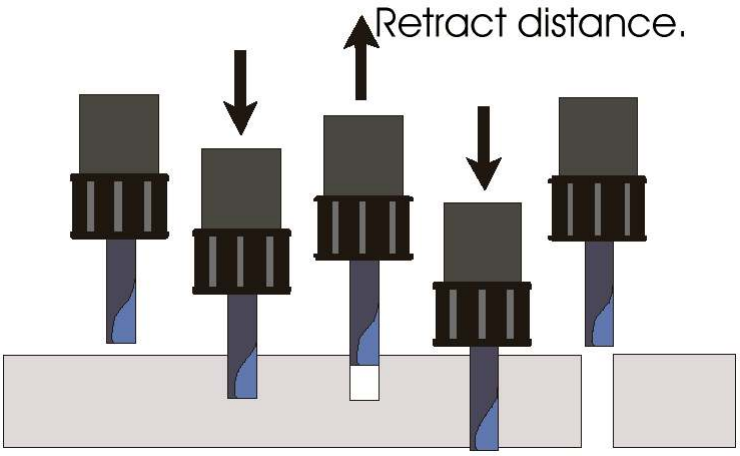
This function allows you to adjust the vertical plunge speed exactly like function 1 sets the feed speed. This is an important feature when cutting metals as end mills must plunge slowly into the material or you will damage the bit.

Peck Retract is only available for jobs that have multi-pass set. It allows you to set a value in the Z axis that the head will retract after each plunge before going on to the next depth, hence giving a "pecking" action. It is useful in cases where a tool is required to plunge a long way into the material and so preventing the cutter from clogging up with swarf.

The retract distance must be a positive value. If it is set to zero (default setting) pecking is disabled and the machine will function as if function 15 was not enabled.

If you set retract to a very large number it will always retract to the lift top position after each peck.

The controller will never go higher than the lift top setting.



Function 16 - Set Job Finishing Position

Use this function to enter an X and Y offset value from the current home position. You may enter a positive or negative offset value. The offset is applied to the X and Y axes and will move the machine the specified distance away from the job start position when the job is completed.

Function 17 - Cut Rectangle

This function will allow you to cut a rectangle. Enter the X size then the Y size at the appropriate prompts. Enter the cutter compensation for the tool you will be using. Press [ENTER] and the router will cut a rectangle of the specified size.

Function 18 - Machine Parameter Settings

This is a multi-purpose function and should only be used by the installer. It is used to provide information to the machine controller about your machine.

Lift On Pause Lift on Pause tells the machine to go to the machine top (upper limit of the Z axis stroke) whenever a pause point is encountered in your design. This may be set to either Yes or No. Use the [+/-] key to toggle between the two and press [ENTER] when your desired option is displayed. The default setting is YES

Display Mode May be set to either Job Name or NC Co-ordinates. Use the [+/-] key to toggle between the two and press [ENTER] when your desired option is displayed. Refer to page 134 for more detailed information on Display Mode.

Drive Type Different machines may have different drive types depending upon their configuration. The choices are AXYZ1, AXYZ2, and Hybrid drive types. Use the [+/-] key to toggle between the choices and press [ENTER] when your desired option is displayed. The default setting AXYZ1. Selecting the incorrect drive type for your machine may result in rendering your machine inoperable.

Axis Mapping Axis mapping allows you to swap the direction in which the machine moves when the jog key is pressed on the sub console unit. Basically you can "flip" the X, Y or Z axes. Axis mapping may be set to either Yes or No. Use the [+/-] key to toggle between the two and press [ENTER] when your desired option is displayed. The default setting is NO.

Airtool Delay This is a delay that the machine applies before firing of an air tool. The default setting is 15, which appears to be adequate for most applications.

Z Sensor Installed Tells the machine whether a Z sensor is installed. The options are **Z Sensor Installed**, **No Z Drives Installed**, **Z Sensor Not Installed**. Use the [+/-] key to toggle between the three choices and press [ENTER] when your desired option is displayed. The default setting **Z Sensor Installed**.

Auto F25 Sensor Tells the machine that a table mounted Tool Tip Sensor is fitted to the machine. The options are **Yes** or **No**. Use the [+/-] key choose Yes or No. Then press the [ENTER] key to accept your choice. If you select **Yes** the machine will perform a FUNC25 to return to home and then tell you to **Jog to F25 Sensor**. Use the direction keys on the sub-console to locate the tool above the centre of the sensor and then press the [ENTER] key. The position of the Sensor will now be recorded. When a function 25 is performed the machine will move to this position on the table to register the currently loaded tool length.

Operation Mode It is possible to set the machine so that operators can only access/change certain functions. Once a factory user mode has been set you will need a password to disable it. You will need to contact the machine supplier to obtain the password. There are three different User levels;

Normal: The operator has complete access to all functions on the machine.

Level 1:The operator can access functions 0, 1, 2, 3, 5, 9, 12 and 13

Level 2: The operator can access functions 1, 3, 4, 5, 7, 8, 9, 12, 13 and 21

Note: In User modes 1 and 2 when the machine is given the Function 3 command it will actually perform a Function 12. Refer to pages 112 and 115 for information on these functions.

Function 19 - Set To NC Mode

The **AMC** controller can be used in two modes, simple or NC. If you wish to use NC mode simply use the [+/-] key to select your mode of choice. In NC mode the controller display will show the absolute co-ordinate position of the machine head. It also allows numerous features which facilitate the machine head to be controlled by co-ordinate entries.

For normal use NC mode is only an added complexity, since we normally allow the job files to control the entire job, so Simple mode is the more commonly used method.

Function 20 - Plunge Pause

This is altered by using function 20. Enter a time (in seconds) for the machine to pause after each plunge. This feature is used when plunging through very thick materials.

Function 21 - Number of Repeats

You can set the number of repeats of a complete job up to 99 times. This is convenient if you are in production and need to keep your efficiency up by keeping the machine running. This is particularly useful in manufacturing applications. To activate this function, bring your job up on the sub console then press [START]. Then press [FUNC] 21 [ENTER] then type in the number of repeats and then press [START] again.

Function 22 - Digitise a Job

Will enter DMI (Direct Machine Input) mode. This feature allows the operator to directly input, on the machine, a shape by tracing an existing shape by driving the machine head around the shape and entering these points. Any number of points and shapes can be entered. This shape can now be cut out or fed back to the host computer and edited. The procedure is as follows for this function follows;

Press the [FUNC] key on the sub console. Enter 22 then press [ENTER]. The console will display;

DIGITISE JOB NAME:
D1

Press [ENTER] to accept this name or type in a new number and press [ENTER]. Files created in this manner will always have the prefix D. If the message "COULD NOT OPEN *FILENAME*" appears this means that job name already exists.

The console now displays;

ENTER START: SHAPE 1

If the head is not in the correct position jog it to the desired start position and press [ENTER]. The console will now display;

PT: 2 X *co-ordinate*
Y *co-ordinate*

Jog the head to the next position and press [ENTER].

Repeat the above procedure around the shape you are tracing until you reach the last point.

To close this shape and finish the procedure press the [FILE] key or the [STOP] key to finish this shape and create another. The console will display;

CLOSE THIS SHAPE? YES

If you do not want to close the shape press [ENTER]. If you do want to close the shape press [+/-] once and then press [ENTER]

The machine will return to the start point of the shape, and then return to its last home position if it was not the same point as the start point.

Function 23 - Display Job Settings

All the settings from ToolPath output menu for the current job will be displayed. This is handy if a job has been in the machine memory for a month, you recall the job but forget the bit diameter or any of the job settings that were programmed in ToolPath, this will ensure the correct tool known and used. Press [+/-] key to sequence through the settings. Press [ENTER] to end function.
Note: none of these setting can be changed in Function 23, just viewed.

Function 24 - Disable Soft Limits

To disable or enable the soft limits of the machine use this f\unction. Soft limits are the programmed limits of travel of the machine which have been set in the ToolPath Setup menu. It prevents the machine from banging in to the ends of the table whilst jogging or cutting. Default setting is on and we recommend that you leave it on.

Function 25 - Set Tool Tip Offset

NOTE: You must also have set up the tool tip sensor position using Function 18 (if you have a non Automatic Tool Change machine) or with Function 350 (if you have a machine with an Automatic Tool Change).

This function allows you to set up the length of your tools in relation to each other. It negates the need to use setting jigs to set all your tool lengths at the same length. You can set up to 16 different Tool Tip Offsets.

To enter a tool tip offset for tool 1 type in [FUNC] 25 [+/-] 1 and then press [ENTER] This is referred to as function 25.1 indicating tool 1 of function 25.

For Machines with Automatic Tool Change (ATC) Units fitted.

Typing [FUNC] 25 [+/-] *tool number* [ENTER] will cause the machine to home. It will then pick up the required tool (if it is not already in the spindle) and then present the following display

AUTO F25 SENSOR? YES

Pressing [ENTER] will cause the machine to move over to the Tool Tip Sensor and lower the tool until it makes contact with the sensor pad. The machine will then home. Repeat the process for any other tools that you may wish to set up.

See also Function 352 for information on how to set up multiple tools in a tool changer automatically.

For Machines without an Automatic Tool Change (ATC) Unit fitted.

Typing [FUNC] 25 [+/-] *tool number* [ENTER] will cause the machine to home. It will then prompt you to insert the correct tool (if it is not already in the spindle) and then present the following display

AUTO F25 SENSOR? YES

Pressing [ENTER] will cause the machine to move over to the Tool Tip Sensor and lower the tool until it makes contact with the sensor pad. The machine will then home. Repeat the process for any other tools that you may wish to set up.

For Machines without Tool Tip Sensor fitted.

Typing [FUNC] 25 [+/-] *tool number* [ENTER] will cause the machine to home. It will then prompt you to insert the correct tool (if it is not already in the spindle) and then present the following display

AUTO F25 SENSOR? YES

Press [+/-] to change the display to

AUTO F25 SENSOR? NO

then pressing [ENTER] will again home the machine then prompt you to insert the correct tool (if it is not already in the spindle).

Jog the machine to a reference surface (such as the top of your waste board) and press [ENTER]

Repeat the process for any other tools that you may wish to set up making sure that you use the same reference surface.

Function 26 - Reset Tool Tips

Function 26 will reset all your Tool Tip Offsets to 0.

Function 27 - Job Process Time

This function will display the process time of the job based on the current feed rate settings. The display will read Min:Sec, such as 2:13. The function is accurate to within 1% and does not take into account any time that the operator has paused the job.

You can use this function during a job pause, it will show the job running time to that point.

To clear the display press [ENTER].

Function 28 - Bit Compensation

This allows you to compensate for the router bit diameter or to adjust the compensation for fine tolerance work. Compensation is half the tool diameter, also known as the tool radius. If you are using a 6mm bit the compensation would be 3mm.

Positive compensation will compensate to the RIGHT of the cutting direction and negative values will compensate to the LEFT. Normally jobs are always set up so positive compensation will correctly cut out male parts and negative will create female ones.

You may also set the bit compensation in the ToolPath software.

***NOTE**: If you set compensation in Toolpath and at the machine the compensations will add together!*

Generally you would only use this function if you are using re-sharpened tools or do not have a tool of the correct diameter for which the job has been downloaded to the machine. For example a job may have been transmitted to the machine with compensation applied for a ¼" (6.35mm) cutter but the operator finds that they only have a 6mm cutter available. In this case, instead of having to re-send the job to the machine with the correct compensation for the 6mm cutter (which would be 3mm) the operator can use FUNC 28 to correct the compensation.

NOTE: The amount that the operator would alter the compensation by would be half the difference of the cutter on hand and the cutter for which the job was downloaded. In this case the compensation to be added would be calculated as follows;

COMP = (6 - 6.35)/2
= -0.175

On tool change machines and machines set up for multiple tools you may apply a global setting that will affect all tools or set the bit compensation for individual tools.

To enter a tool bit compensation for tool 1 type in [FUNC] 28 [+/-] 1 and then press [ENTER] This is referred to as function 28.1 indicating tool 1 of function 28. To enter a tool bit compensation for tool 3 type in [FUNC] 28[+/-] 3 and then press [ENTER].

Calling Function 28 on its own will result in the following display and apply the compensation to ALL tools;

Set All Tool Comp? No

Press the [+/-] key to toggle to **Yes**. The display will now read;

Cutter Comp +0.000

type in the compensation required and press the [ENTER] key. The display will inform you that the settings have been saved;

Saving O/S to flash mem

To set the compensation for a single tool (say Tool No.1) follow the steps below;

Type in [FUNC] 28 [+/-] 1 and then press [ENTER] the display will show;

Cutter Comp +0.000

Type in the required compensation and press [ENTER] The display will inform you that the settings have been saved;

Saving O/S to flash mem

***NOTE: The compensation settings you enter will stay effective until they are cleared using Function 28 again**.*

Function 29 - Show Image File Information

This function will display the currently installed version of the machine operating systems (Image File). Press the ENTER key to clear the display. The function also shows the release date of the machine O/S as well as the spindle wiring type.

Function 31 - Select Individual Tool for Single Operation

For machines fitted with an Automatic Tool Changer this function allows you to select a single tool from the ATC unit to run a single tool job. Refer to the section on Automatic Tool Changers for more detailed information.

Function 32 - Enable/Disable Second Carriage

This function is used to enable a second carriage or gantry if your machine is fitted with a second carriage or gantry.

Function 33 - Return Tool To ATC Rack

Empties the spindle by returning the currently loaded tool to the Tool Changer rack. The display will read "Dropping Tool *X*" where *X* is the number of the currently loaded tool .

Function 35 - Save User Settings

This function allows you to save a number of settings at the machine for recall at a later time using a corresponding Function 36. With function 35 you can save your settings for functions 1, 4, 8, 10, 15 and 84. You can save up to 16 sets of these settings by using function 35.1, 35.2, 35.3 and so on up to function 35.16

The idea behind this function is to allow users to quickly recall a number of previously saved settings for commonly used jobs.

For instance if you commonly cut 12mm MDF with a particular cutter you might save all the settings for this procedure in Function 35.1. If you also commonly cut 4mm acrylic with a particular cutter you might save all the settings for this procedure in Function 35.2

You can also adopt a procedure where Function 35.1 has all the settings for say a ¼" Spiral Cutter for MDF, Function 35.2 may have all the settings for ⅛" plastic cutter, Function 35.3 may have all the settings for a 90° V cutter for timber and so on up to function 35.16.

To save a set of settings perform your functions 1, 4, 8, 10 and 15 as though setting up a normal job. Then on the sub-console type;

[FUNC] 35 [+/-] 1

You will notice that the display will show

Function 35-T1

This is normal.

Press the [ENTER] key and the display will show;

Save settings? YES

press the [ENTER] key to save the settings or you can press the [+/-] to change the display from YES to NO and then press the [ENTER] key which will abort the save.

If you choose to save the settings the display will read

Saving to flash

Function 36 - Recall User Settings

Use function 36 to recall one of the settings that were saved using Function 35.

For example entering [FUNC] 36 [+/-] 1 will recall the settings that were saved using Function 35.1

Executing the command will result in the following display on the sub-console;

Get saved settings? YES press [ENTER] to recall the settings or you can press the [+/-] to change the display from YES to NO and then press the [ENTER] key which will abort the recall.

Note: Recalling settings will overwrite your current function 1, 4, 8, 10 and 15 settings.

Function 38 - Set Up Material Off Loader Arm

If your machine is fitted with a material off loader use this function to set it up.

Entering [FUNC] 38 [ENTER] will present the following display;

Setup Scraper? NO

Press [+/-] to change the display to;

Setup Scraper? YES

The machine will now home and the display will read;

Set Scraper Start

Jog the machine to the position where you want the scarper arm to lower down. This should be 20mm or so away from the front edge of your sheet of material.

Press [ENTER]

The display will now read;

Set Scraper End

Jog the machine to the point where you want the arm to raise after it has moved the material off the machine bed.

Press [ENTER]

The material will then home and save the settings to memory.

Function 39 - Activate Material Off Loader Arm

To activate the material off loader type in [FUNC] 39 [ENTER]

The machine will home, lower the arm and move the material off the machine bed according to the settings you entered in Function 38.

Functions 61, 62, 63, 64 - Preset Measured Moves

This function works similar to Function 6 in that it allows the operator to enter values for X and Y axes for the machine to move from its current position. this is done by following the steps below;

- Enter [FUNC] 61 or 62, 63 or 64 and press [ENTER]
- Enter the X direction distance, press [ENTER].
- Enter the Y direction distance, press [ENTER].
- Select either **move with head up** or **move with head down** using the [+/-] key. Press [ENTER]**.**

If you selected head down, the head will lower to the lift bottom position and will move the set distance and lift at the speed set by function 1. If you selected head up it will move without lowering at maximum speed.

If Function 61 is used the values are stored on the controller as default values for this function. The same is true if you chose Functions 62, 63 or 64. It in effect gives four programmable set moves for jigging or if the operator constantly uses the same offsets.

The next time Function 61-64 is called the defaults will be displayed. The operator need only accept the defaults by pressing the [ENTER] key.

Function 91 - Delete Job File

To delete a job from the machine controller use Function 91. Once a job has been erased from the controller it cannot be recalled.

Function 92 - Clear Jobs From Memory

This will erase **ALL** the files from the machines memory. It is like doing a Function 91 on every file.

Function 121, 122, 123, 124 - Preset Home Positions

These functions allow you to set extra "Home" positions (up to 4) in any position within the machine process area. This is especially useful if you use a number of jigs or location stops on your machine for cutting components.

When you execute a FUNCTION 121, 122, 123 or 124 the machine will home to the limits at the lower left corner of the bed. This is the reference point from which FUNCTION 121, 122, 123 or 124 are calculated.

The first time you execute a FUNCTION 121, 122, 123 or 124 the display will read; **Offset 0, 0**

This indicates that there has been no position on the machine bed assigned to the function. To assign a position for FUNCTION 121 for the first time follow the steps below;

- On the sub console type in [FUNC] 121 [ENTER] . The machine will home to its absolute position
- The display will read **Offset 0,0** Press the [FUNC] key.
- The display will read **F6 Enter x move** Use the numeric keys to enter a x value for the home position then press the [ENTER] key. This value will usually correspond to a x position of a fixed jig on the machine bed.
- The display will then read **F6 Enter y move** Use the numeric keys to enter a y value for the home position then press the [ENTER] key. This value will usually correspond to a y position of a fixed jig on the machine bed. The machine will then move to position that you have entered.

Note: You could also use the directional keys on the keypad to position the head at the correct co-ordinates and then press the [ENTER] **key**.

You can repeat the process for FUNCTIONS 122, 123 & 124.

Once a FUNCTION 121, 122, 123 or 124 has been set you can move to that position simply by entering the FUNCTION number and then pressing the [ENTER] key or you can overwrite the setting by following the above procedure.

Function 159 - Erase OS

This will erase the operating system from your machine and render it inoperable. You should never perform this function unless specifically instructed to by a technician.

Function 181 - Seek Sensor & Plunge Acceleration

This function works in two parts and can be set at either the machine with this function or through Tooplath|Setup.

Seek Sensor: This setting tells the machine to start decelerating toward the homing sensors (X, Y and Z axes) when it is the programmed distance away from the sensor. When executed the display will show the currently set value. Type in a new value or press [ENTER] to keep the current setting.

Plunge Acceleration: Sets the distance over which the Z axis accelerates to its programmed speed. Setting too small an acceleration distance may cause the Z axis to stall. Setting too large an acceleration distance means that the Z axis will take a long time reach its programmed speed. Type in a new value or press [ENTER] to keep the current setting and save to Flash memory.

Function 330 - Set Up “S” Mouse

On machines supplied with a moveable “S” mouse (Tool Tip Sensor), this function is used to set up the “S” Mouse.

The initial one-time setup of the S-Mouse saves to memory the thickness of the S-Mouse itself, the default depth of cut below the material and the default height of travel above the material. When material to be cut is placed on the machine, it is necessary to program the machine for the height of the material surface and also the material thickness. By placing the S-Mouse onto the material surface and selecting Function 84, the material surface is automatically sensed via the S-Mouse, you are then prompted for a manual entry of the material thickness and you are ready to cut. Function 84 would typically be carried out whenever material is placed onto the table of a different thickness or if a router bit is replaced for any reason. With these two functions, the router table becomes aware of the maximum depth of cut, the upper and lower surface of the material and the default height of travel above the material.

When working with multiple spindles, the S-Mouse operates in a similar fashion utilizing Function 25, 25.2, 25.3 etc... to determine the surface, relative to a particular spindle with great accuracy.

At the machine sub-console enter [FUNC] 330 [ENTER] The display will read

Setup “S” Mouse? YES

The first setting simply initiates the ‘S’ Mouse. Use the keypad [+/-] buttons to toggle this setting on and off.

(T1) Set Surface: + 0.00

With this setting you provide a reference surface to calculate the thickness of the ‘S’ Mouse. With a tool in the spindle touch off on a flat surface. It is important that the surface that you use is clean and flat because you will be using this surface for a reference so it has to be correct. Use the toggle key [+/-] to slow down the head as it gets close to the surface so the you can touch off accurately. Once you have touched off on the surface hit the [ENTER] key. The head will move up to allow you to place the ‘S’ Mouse under the tool.

Position “S” Mouse
Press ENTER to Continue

Now that the head is up, position the ‘S’ Mouse under the spindle, and hit the [ENTER] key.

Below Bottom: + 0.00

This setting will now indicate how far into the waste board to cut. It must be a positive value.

Above Top: + 0.00

This setting indicates how far above the material surface the head has to be when it is traversing from shape to shape. This value is always positive.

Saving to Flash

All the setting that have been setup will be saved to flash memory.
The ‘S’ Mouse it now ready to be used. To setup up the material surface, lift top and lift bottom you can do a Function 4.

Function 330 - “S”Mouse Usage

Once the S mouse has been set up using function 330 you can make use of it in the following ways.

Performing a Function 4
To set up a job you use a function 4 as normal, but the application of function 4 is slightly different when the S mouse is employed.

When you perform a function 4 the display will show;

Position “S” Mouse Press ENTER to Continue

[FUNC] 4 [ENTER]

The head will travel down until the tool touches the S mouse and then return to the top position.

Enter Thickness: + 00.00

Type in the thickness of the material then press the [ENTER] key

When you need to replace the tool perform a function 4 as described to re-set tool lengths.

Performing a Function 8
If you need to re-set the surface of the material use function 8

When you perform a function 8 the display will show;

Position “S” Mouse Press ENTER to Continue

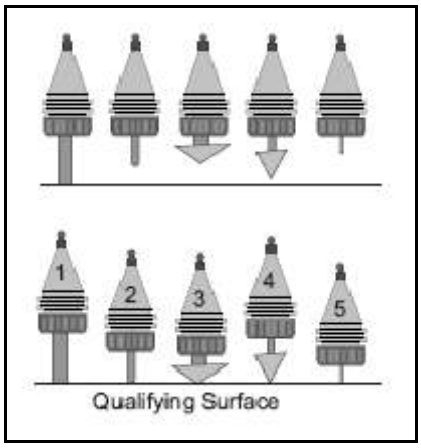
[FUNC] 8 [ENTER]

The head will travel down until the tool touches the S mouse and then return to the top position.

Function 350 - Set Up Tool Changer/By pass Tool Changer

For machines fitted with an Automatic Tool Changer this function allows you to set up the ATC unit or to by-pass it if required. Refer to the section on Automatic Tool Changers for more detailed information.

Function 352 - Toolchanger Automatic Tip Sensing

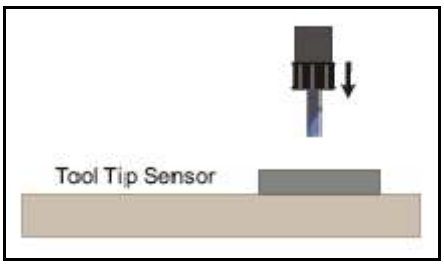


For machines fitted with an Automatic Tool Changer (ATC) this function allows you to automatically have the machine set up the Tool Tip offsets for the tools in the ATC unit.

The upper portion of this diagram shows a typical set of tools, each one has a different length.

When the operator sets the lift top and bottom and the Z datum he uses one tool, usually tool 1, to set the positions. When the machine runs and selects a longer or shorter tool than tool number 1 it must compensate for the differing length of tool.

This is done by touching each tool off the Tool Tip Sensor so the system can record the difference between their lengths. The system will store these values in flash memory and use these settings to correctly adjust for the different length tools.



The machine will pick up tool 1 and move to the tool tip sensor and do a automatic tool tip, for tool 1. This sequence will continue until all tools have been qualified or “tipped off”.

Refer to the section on Automatic Tool Changers for more detailed information.

NOTE: Your tool changer must have a tool in every position prior to initiating this function.

Function FILE - Search for a Job in Machine Memory

Use this function to quickly locate a job in machine memory. This function is only available for job names that are numeric or alpha numeric where the numerals form the start of the file name. For example

12123
12TEST
243TST1

If you store many jobs at the machine, using [FUNC]+[FILE] is a quick way to locate a particular job rather than scrolling through the many jobs one after another until the correct one is displayed.

To use this function, press the [FUNC] key then press the [FILE] key. The display will read;

type in the first few characters of the job you want to find then press the [ENTER]. The first job that matches what you have typed in will be displayed. For example, using the list of job names above;

Type in [1] will locate jobs 12123 and 12TEST.
Type in [12] will also locate jobs 12123 and 12TEST.
Type in [121] will locate job 12123 only

If there is no match then the next job in machine memory will simply be displayed.

Display Mode / NC Mode

The controller will default into “Display Mode”. This mode is designed to make it as simple and straight forward as possible to set up and run a job. It disables many of the co-ordinate features of the controller. To switch to NC Mode which will allow the entire range of features simply select function 19. The display will change to co-ordinate display;

X:0.00 Y:0.00 Z:1.50

Selecting Function 19 again will switch you back to simple mode. In several of the following sections you will note that some of the features available are only active while you are in NC mode.

Jogging and Positioning

You can position the head of the machine anywhere in a 3 dimension space not exceeding the size of your machines bed in the X and Y axis or the stroke range of your tool head in the Z axis.

Depressing and holding the [6] key will move the head in the positive X direction. If you are standing and facing the machine this will be to the right.

If you let go of the [6] key the head will stop. Note that the X co-ordinate on the display will increase as you move to the right.

The other directions are accessed using other keys on the keypad:

2 positive Y
8 negative Y
4 negative X
6 positive X
5 positive Z
0 negative Z

You can also jog in the diagonal directions:

1 negative X and positive Y
3 positive X and positive Y
7 negative X and negative Y
9 positive X and negative Y

In all cases the LCD screen will track the co-ordinates of the head as you jog it around.

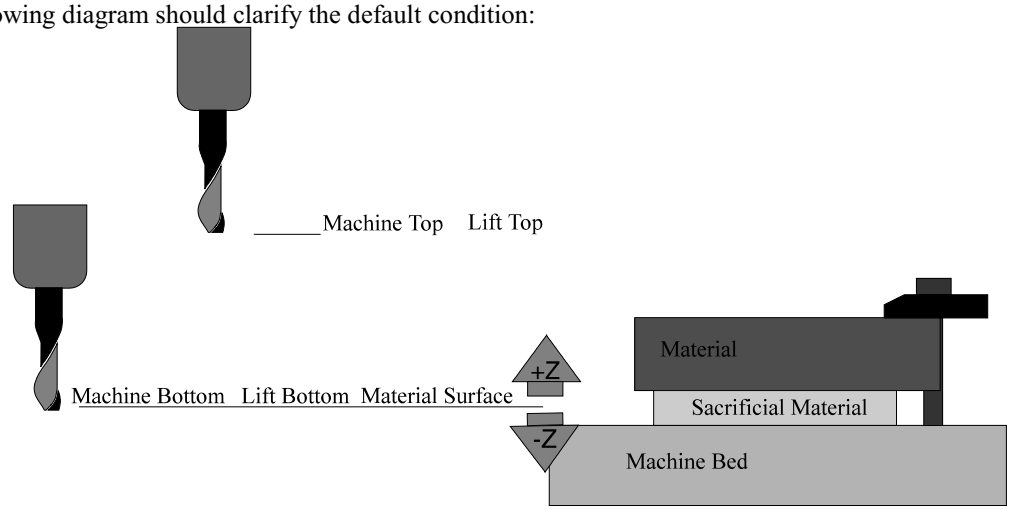
Generally, the machine head will jog at **FAST** speed which is the maximum travel speed of the machine.
By pressing the [+/-] key you can select **MEDIUM** speed which is half the maximum travel speed.
Press [+/-] again and the jog speed will become **SLOW**.
Press [+/-] again and the jog speed will become **INCREMENTAL** where the machine will move 0.02mm at a time.
Press [+/-] again and it will return to **FAST** jog speed.

Setting Origins and Vertical (Z) Positions

When the **AMC Controller** first starts up and receives the program from the design station, the X and Y position that the machine head happens to be in at the time become the X and Y origin or (0,0) position. In the Z direction the head will move up to the Machine Top and stop there. The Z origin or zero position is placed at Machine Bottom.

Machine Bottom is defined as the machine's maximum stroke below Machine Top. The machines maximum stroke is defined in the Setup menu in **ToolPath**, it is a mechanical limitation of the machine itself.

The following diagram should clarify the default condition:



Note that there are 5 defined vertical or Z positions. These positions define the vertical space in which your project will operate:

Machine Top: This is the physical top of the Z or vertical stroke. The controller automatically seeks this position using a sensor. This is the only fixed vertical position so the controller will often seek this in order to calculate all the other Z positions.

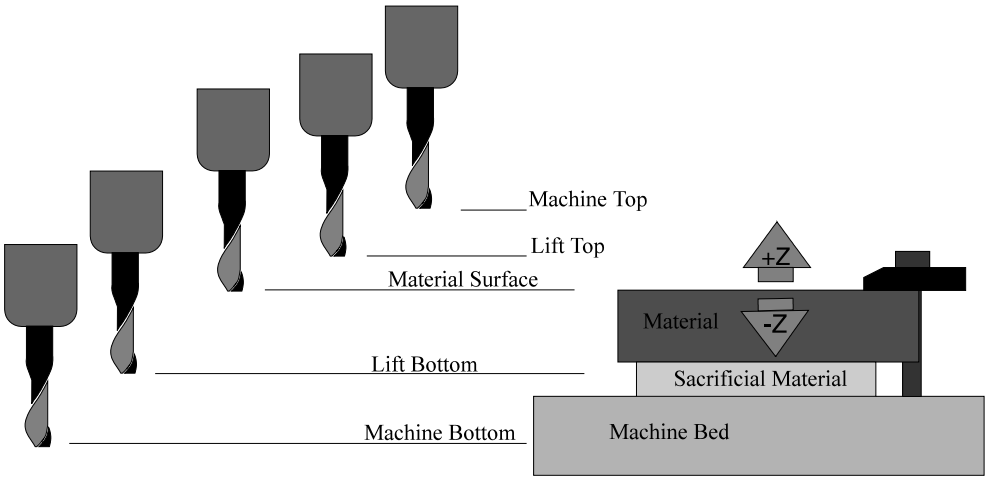
Machine Bottom: This is simply the physical bottom of the Z or vertical stroke. The distance between machine bottom and machine top is the stroke or Z size of the machine. This is defined in the ToolPath setup menu. Both the Machine Top and Machine Bottom positions are defined by the mechanics of the machine, they cannot be altered by the controller.

Material Surface: This is the Z origin or zero position. The vertical or Z co-ordinates are measured from this point. This point is set by Function 8.

Lift Bottom: This value can be defined by selecting Function 4. As you can see from the diagram it is possible for the Machine Bottom value to cause the tool to cut into the machine bed.

Lift Top: Often the Machine Top position is far above the material surface. When the machine is running job and travelling between cuts it will move up to Lift Top before travelling to the next plunge point. Setting Lift Top to be just clear of the material normally saves a lot of process time. NOTE: Do not forget about any clamps that may be in the travelling path, and set the lift top high enough to clear them.

The default vertical positions are not normally the way you wish to run a job as you are in danger of cutting into your machine's bed. The following diagram shows how you normally would wish to set up your vertical positions:



Function 3 Resetting the Origins

To set the origins and material surface enter Function 3:

[FUNC] 3 [ENTER]

The X and Y co-ordinates will automatically be set to (0,0). It will also set the Material Surface to the default position which is at machine top.

Once Function 3 is completed the tool head will return to the machine top position.

Function 4 - Setting Lift Top and Lift Bottom (Material Thickness)

Setting a function 4 is very important. This function controls the depth to which the machine will cut.

NOTE: If your machine is fitted with an Automatic Tool Changer (ATC) please also refer to the following page.

The **lift top** setting is the height above your material surface the tool tip will lift when moving between shapes. It is normally set 5-10mm above the material surface. You must ensure that this setting will also clear any clamps or jig fixtures.

The **lift bottom** position will limit how deep a program can drive the tool head. You should set it so that it is higher than the machine bed.

The lift bottom position has slightly different purposes in two dimensional and three dimensional jobs.

> **Two Dimensional.** Two dimensional files do not include any Z dimensions so the lift bottom position will become the cutting depth. The lift top position is the travel height between cuts.
>
> **Three Dimensional.** Three dimensional files do specify Z dimensions so the lift bottom position becomes a safety level. Regardless of how the file wishes to drive the tool head it will not move lower than the lift bottom position. The lift top position is still used as the travel height between cuts.

To set the material surface, lift top and bottom use Function 4:

[FUNC] 4 [ENTER]

The display will show:

Set Surface: +0.000	Using the [↓] key jog the head down to the material surface and press [ENTER]. Generally this position is the surface of the material to be cut, however in 3D engraving you may wish to set this slightly above the material surface (say 0.5mm) so that the tool fully exits the material when lifting out of the cut.

If you are in NC mode: You can type in a position by pressing [FUNC] then typing in a numeric value and press [ENTER].

Once you have entered the lift top (material surface) position you will be prompted to enter the lift bottom. This is generally the thickness of the material *plus* an allowance for the tool to go through the material (say 0.5mm):

Lift Bottom: -0.000	Type in a value for the material thickness plus a cut through allowance (for 10mm thick material you would enter 10.5) and press the [ENTER] key.

The final prompt will ask you to set the lift top setting;

Lift Top: +0.000	This is the distance above the material that the tool tip will move to when travelling between shapes in the job. It is normally set 5-10mm above the material surface. Type in the desired clearance amount and then press [ENTER]. You must ensure that this setting will also clear any clamps or jig fixtures.

Once the settings have been entered the head will return to the machine top position.

Function 4 and Automatic Tool Change Machines

On machines fitted with Automatic Tool Change units the function 4 setting is performed using TOOL 1. If tool 1 is not loaded in to the spindle when you perform the function 4, the machine will first return the currently loaded tool to the tool changer and then pick up tool 1. It will then carry on as described below. On ATC machines the relative length of tools recorded during the function 352 is referenced from tool 1 in the ATC unit.

It is possible to perform a function 4 on individual tools by selecting the tool number using th +/- key during the function 4.

Let's say that you are in the middle of a job currently cutting with tool 3 and find that you need to change the tool due to wear.

After you have changed the tool perform the function 4 using the method below.

[FUNC] 4 [+/-] 3 [ENTER]

This will allow you to set your depth as described on the previous page using tool 3.

Starting and Running a Job

Once you have understood how the origins work and how to set up the various vertical (Z) positions we can try to run a job. First we must make sure there is a job resident in the controller's memory to run:

Press the [FILE] key. It should show the name of a file such as:

Job TEST1 ready

If there are no jobs in the memory the screen will show:

AMC 3D SERIAL NO. XXXXX

If you keep pressing the [FILE] key the controller will show all the jobs resident in memory in sequence. Once you have run through the list of resident jobs, the list will start again at the beginning.

You can check the dimensional size of the job by pressing [ENTER] while the job name is showing on the screen. For example:

SIZE: Long 420 x High 80

Press [ENTER] to return the display to the job name.

Pressing [START] will begin the process of running the job. Before we do this we generally wish to set the feed and plunge speeds to be used by the job.

Function 1 Feed Speed

The feed speed is altered by using function 1:

[FUNC] 1 [ENTER]

The display will show:

SPEED 10000 mm/min

NOTE: Depending on the program you used to generate the NC file in the first place or the speed controls you may have used in ToolPath you may have feed speed information in the file itself. The controller will always compare the programmed speed in the file to the feed speed setting in Function 1 and use the SLOWER of the two. If there is no feed speed information in the NC file the speed used will be the Function 1 value.

The feed speed can be altered using Function 1 at any time. You can pause a job part way through its run, change the feed speed and resume at the new speed.

This feature is extremely convenient as you may adjust the feed speeds as you go depending on how well the job is progressing.

Function 15 Plunge Speed/Peck Retract

Function 15 allows you to control the maximum plunge speed. This is the maximum speed that the machine will lower the tool bit into the material to be processed. For example:

[FUNC] 15 [ENTER]

The display will show:

PLUNGE SPEED: 2000 MM

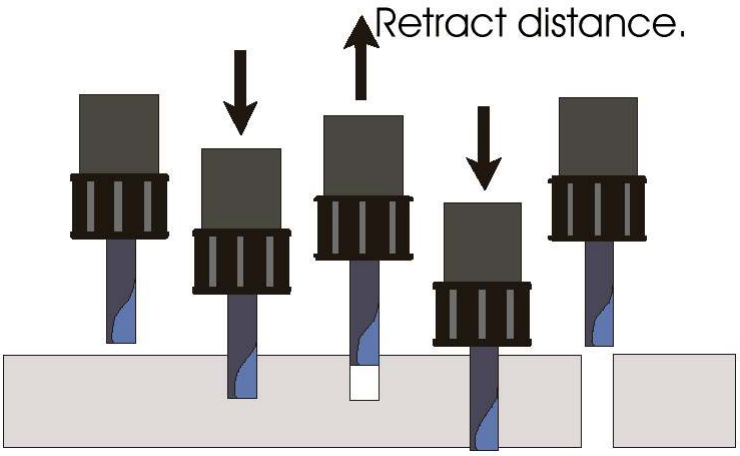
This value can be altered by using the numeric keys exactly the same as function 1. For example to change the value to 1000MM simply enter:

[FUNC] 15 [ENTER]
1000 [ENTER]

This function can also be altered while you are running a job.

Function 15 also allows you to set a peck retract distance.

Peck Retract is only available for jobs that have multi-pass set in them. It allows you to set a value in the Z axis that the head will retract after each plunge before going on to the next depth, hence giving a "pecking" action. It is useful in cases where a tool is required to plunge a long way into the material and so preventing the cutter from clogging up with swarf.



The retract distance must be a positive value. If it is set to zero (default setting) pecking is disabled and the machine will function as if function 29 was not enabled.

If you set retract to a very large number it will always retract to the lift top position after each peck.

The controller will never go higher than the lift top setting.

The settings for Peck retract follow the normal Function 15 settings and are entered simply by typing a value for the retract distance when prompted. To set a 5mm peck retract distance, following the example below.

[FUNC] 15 [ENTER]

PLUNGE SPEED 2000 MM

[ENTER]

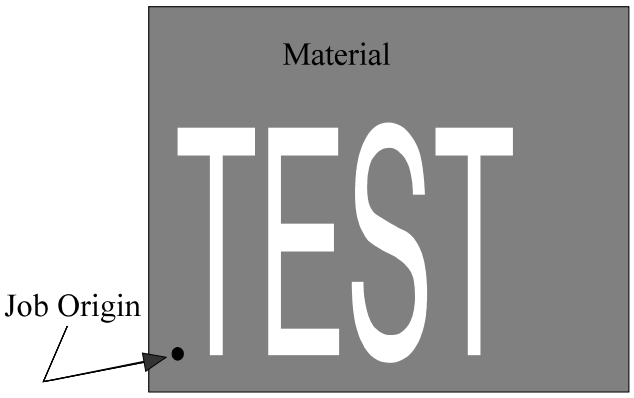
PECK RETRACT: 0.000 MM

5.00 [ENTER]

The display will return to the job ready display.

Positioning the Tool Head and Starting a Job

Once you have set your maximum feed and plunge speeds, and have set up your vertical positions, you can start your job! Use the jog keys to move the tool head to where you wish the job origin to be. Generally this is over the lower left hand corner of your material:



Press [START] and the following will happen:

The Tool head will seek the detector at the machine top position.
The X and Y co-ordinates will be set to (0,0). The present tool head position become the origin.

At this point there is a slight difference whether you are running a two or three dimensional job:

If you are running a two dimensional job:

The tool head will drop down to the Lift Top position.
The display will alternate between showing the size of the job and a message telling you to press [START] to run or [FUNC] 9 [ENTER] to abort.

Press [START] again and the job will begin to run.

At this point the controller takes over. It will run through the entire file. When the job is done, the tool head will return to the origin and the machine top position. The display will show:

Job TEST1 completed

Pausing and Aborting Jobs

While you are running a job it is possible to pause it by pressing the [STOP] button. As soon as you press [STOP] the machine will stop cutting and lift the head to the lift top position. The display will show:

PAUSED ON SHAPE #

At this point you are allowed several "run" features:

You can use the jogging keys to move the head away from the cut. Once you press [START] the head will return to the point it left the cut. Plunge down and continue the job. This feature is useful if you wish to inspect the cut.

You may alter the Feed Speed (Function 1) and/or the Plunge Speed (Function 15). The procedure is identical to that described in preceding sections. Once you press [START] the machine will continue the job at the new speeds.

The lift bottom position can be altered using Function 7. This is described in more detail on page 115. Basically, the operator simply enters a negative distance value to move the lift bottom deeper into the cut, or positive distance value to make it shallower. This feature is useful if you find that the cut is not quite the depth you wish it to be. Again when you press [START] the job will continue where you left off, but using the new lift bottom setting.

Finally you can abort the job you are running. After you have pressed [STOP] simply enter:

[FUNC] 9 [ENTER]

The machine head will move to the machine top position and will return to the home or (0,0) co-ordinate where the job started. The job file will remain resident in the controller so you may start the job over if you wish.

Starting a Job Part Way Through a File

Sometimes you may need to run a job beginning at a point other than the start of the job's file. An example of this would be if you were running a job and the tool bit broke part way through it:

Press [STOP] to pause the machine.

Use Function 9 to abort the job. The machine head will return to the job's origin or start point.

You may use the jog keys to move the head to a convenient position to change the tool bit. Change the tool bit making sure that you do not accidentally move the machine's head.

You should reset the lift top and lift bottom positions whenever you install a new tool bit. It is likely you will not be able to reinstall the new bit to the exact same depth as the original was. Simply use Function 4 to do this the normal way.

You will now have to get the machine's head back to the original start or origin point. Enter:

[FUNC] 13 [ENTER]

Function 13 will cause the controller to return the machine's head to the start point of the last job.

Press [START] the usual way to begin a job. If the machine requires a material surface it will prompt you for one. Drive it to the material surface and press [ENTER] . However before pressing [START] the second time, press the [+/-] key instead. The head will jog over to the plunge point of the second shape in the job file. Press [+/-] again and it will go to the third shape, then the fourth, etc. You can press [+/-] until you have positioned the head at the plunge point of the shape you wish to start with. NOW press [START]. The job will begin cutting at the selected plunge point.

Sometimes you may find that the plunge point you wish to start with is many shapes from the start of the file. Pressing the [+/-] key dozens of times to get the machine's head to the desired plunge point will get tedious. If this is the case press the [ENTER] key instead. The controller will prompt you:

Simply enter in the shape number you wish to start at. The machine will move over to the selected plunge point. You can use the [ENTER] and [+/-] key interchangeably to find your required starting position. Press [ENTER] to commence cutting.

It is also possible to alter the position of the shape you are starting with by using the jog keys to move the head. For example:

Press [START] to begin the start sequence of a job.

[ENTER] 12 [ENTER] will position the head at the plunge point of shape 12 in the job file.
Use the [4] key to jog the head to the left (negative X).

Press [START] and the job will begin with shape 12, plunging at the location the head is presently at.

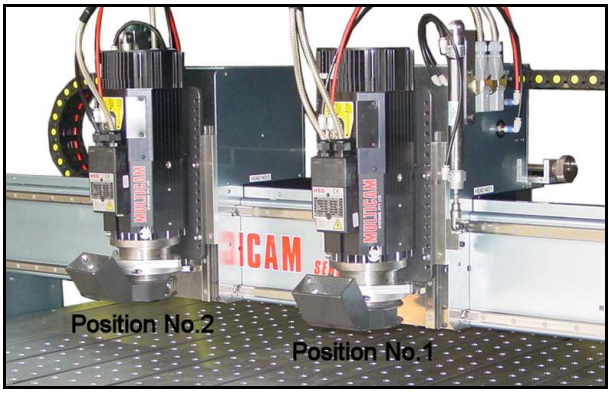
MULTIPLE SPINDLES ON MULTICAM CNC ROUTERS

A popular choice for increased production is a “Dual Carriage” system. Each carriage carries it’s own spindle which can be used either in tandem, for twice the productivity, or as a tool change. When running in tandem the additional head or heads mimic the first head’s operation creating a identical toolpath at a offset to the first head. This offset is chosen by adjusting a lead screw at the back of the head assembly to the distance required for the job. When running as a tool change each head can be assigned as separate tools so you can process a job with up to four different tool sizes or profiles.

Concept of Operation

The basic operation of multiple spindle machines looks a lot more complicated than it really is. All the spindles are mounted on the same gantry and there is only one X and Y axis drive . The only difference is the Z axis, so all the programming parameters with respect to the Z axis have a position 1 to 4 option.

Looking at the front of the gantry the Z axis are numbered from right to left. Axis position 1 is the primary axis (nearest operator). Position 2 is the second from operator, position 3 the third from operator and position 4 is the fourth head from the operator side.



Machine Operators Side

Additional functions at the Sub Console on Multiple Spindle Machines.

Before you can work with multiple spindles you must tell the ToolPath software that your machine is fitted with more than one head. This is described in the section on **Setup|Configure Tooling** on page 95

For a detailed explanation of multiple tools and their setup and use refer to Appendix F commencing on page 161 and Appendix G commencing on page 165.

Function 25

This function is used to set up the Tool Tip Offsets (the difference in length of the cutters in the different heads) for the cutters in the different heads.

For each head you must perform a Function 25. So for head 1 you perform a function 25.1, for head 2 a function 25.2 and so on.

Once all the tool tip offsets have been entered you simply perform your Function 4, 7 and 8 on the tool in head (position) 1. The controller then applies the offsets to the other tools as required to ensure correct machining depths.

Setting up Multiple Tools and Heads

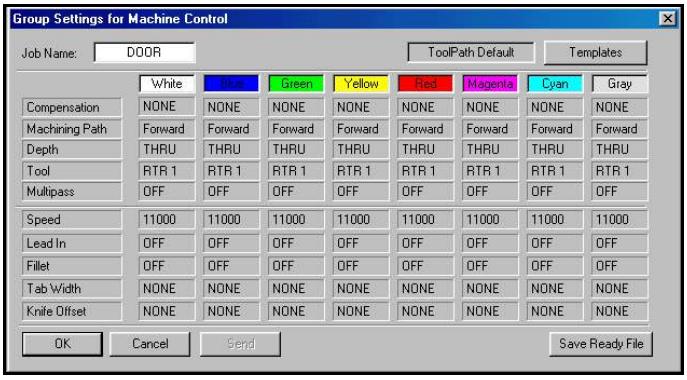
Before you can use multiple Tools and Heads it is necessary to tell ToolPath about certain parameters associated with the tool/head. This is done with the CONFIGURE TOOLING command on the MACHINE INFO tab under the SETUP menu.

Refer to page 95 for more detailed information on how to do this.

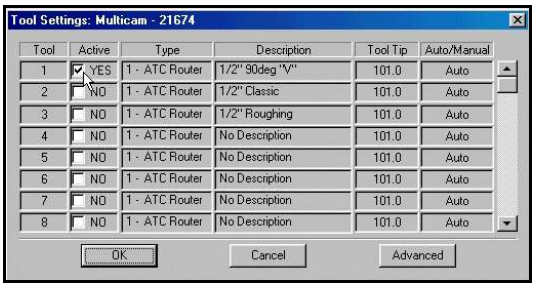
For a detailed explanation of multiple tools and their setup and use refer to Appendix F commencing on page 161 and Appendix G commencing on page 165.

For you to be able to set up tooling and heads your Multicam machine must be turned on and communicating with the host computer.

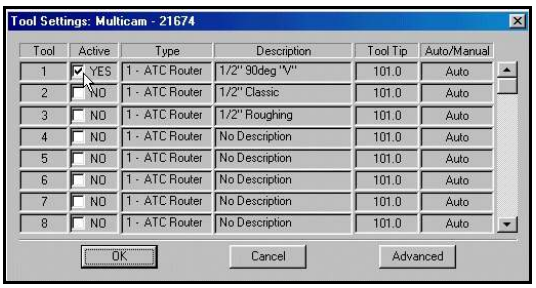
By clicking on the “Tool” field (usually has a default value of RTR1) in the group colour that you wish to assign, a Tool Setup dialogue box will appear.



This is where you tell ToolPath which tool to use for the group you have selected.



To use the second head in a given group simply enable it by clicking the check box in the Tool 2 row to change it from NO to YES.

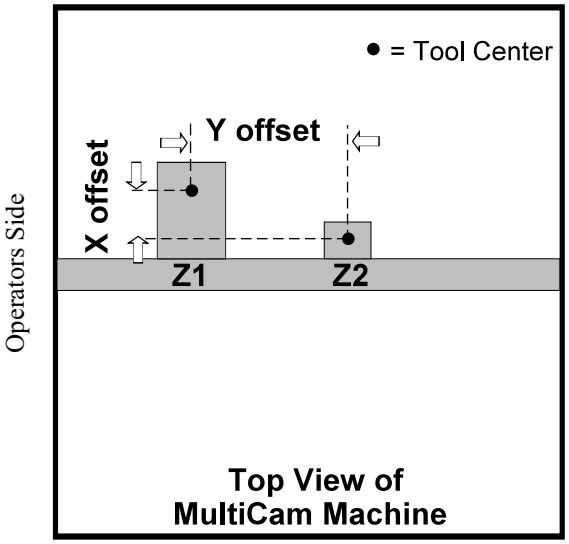


The following sections explain the setup of various tooling combinations and further explains the concept of Tooling Offsets.

Tool Offsets

Many combinations of multiple spindle jobs will require the programming of the offsets between tools. The offset is always measured from the head in position 1 from the centre of the tools. Adjacent is an example of a spindle and drill combination.

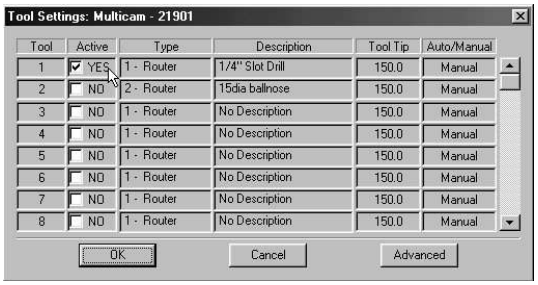
Note: For machines with dual Perske Spindles there is no X offset required.



Dual Spindles Running In Tandem

If you wish to run a job in tandem with both spindles operating together you can basically double your production rate. You will need to tell ToolPath about which tool is located at which head and position. This is done in the **Groups** menu. Refer to Appendix F and Appendix G for more information on setting up multiple heads.

Assume the job you wish to rout is in the "white" group. Click on "RTR1" located in the "White" column. You will see a dialogue box similar to this one. The values of your dialogue box may differ if you have different tools set up with different X and Y offsets, however you must make sure that **Type** of head for both tools 1 and 2 is set to Router and that both these tools are **Active** by clicking on YES/NO to read YES.



Click on OK to confirm the settings. In the White Group in the Groups Menu you will notice that the **Tool** setting now displays RTR1/RTR2 indicating that both heads are active for this group..

Note: the same procedure is used if you want to run 2 drill heads in tandem. All you need do is ensure that the tool type is set to Drill.

At The Machine

- Set the heads the appropriate distance apart by adjusting the lead screw between them.
- Next set the tool tip offsets using function 25
- Set you functions 4, 7 and 8 as required using the tool in position 1. You do not need to perform these functions for any of the other tools in any other position.
- Set your other functions as you normally would, for example, function 1 for feed speed,.
- Now press start. If you have a job with depths in it or 3D you will be prompted to set the material surface.
- Press Start again and the machine will run your job.

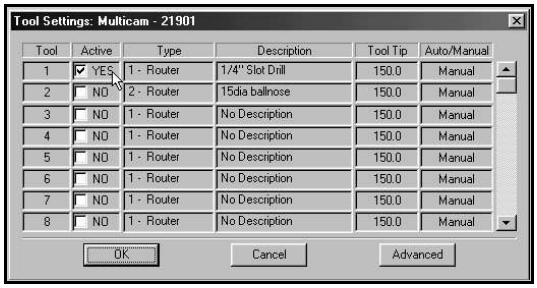
Using The Second Head As A Tool Change

In jobs where multiple tool changes are required it is possible to set up a dual head system so that the second head acts a s the tool changer carrying your second tool.

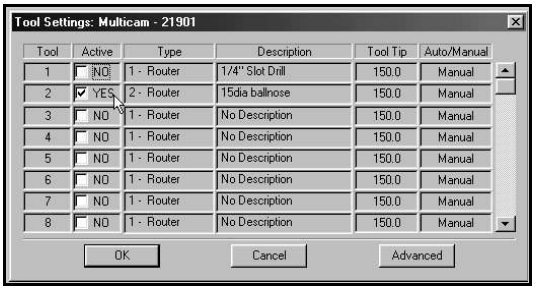
To run a job using the second head as a tool change divide the job into two groups (e.g. white and blue) using the **Groups|Setup** menu.

Leave the white group as it is with RTR1 as your tool.

On the blue group click on Tool and set Tool 1 **Active** to "**NO**" and set Tool 2 **Active** to "**YES**".



Group Settings for Tool 1(White)



Group Settings for Tool 2 (Blue)

Notes:

Your X and Y offset values are entered in the SETUP|CONFIGURE MACHINE dialogue box. It is not possible to change these values here.

The distance of the offsets should be measured from the center of the tools.

These examples are given for two identical heads with no X axis offset. If you are using dissimilar heads, such as a Perske and Drill combination, then you will need to measure your X offset.

At The Machine

- Set the heads the appropriate distance apart by adjusting the lead screw between them.
- Next set the tool tip offsets using function 25
- Set you functions 4, 7 and 8 as required using the tool in position 1. You do not need to perform these functions for any of the other tools in any other position.
- Set your other functions as you normally would, for example, function 1 for feed speed,.
- Now press start. If you have a job with depths in it or 3D you will be prompted to set the material surface.
- Press Start again and the machine will run your job.

APPENDIX A

Multicam RS485 Connectors, LRC Cards and Cables

The Multicam CNC Routing System employs either an RS485 connector or an 8 bit ISA card to establish communications between your design computer and your Multicam machine via a communications cable (called the LRC cable). The card is known as an LRC (Long Range Communications) card.

Without the connector or card you would not be able to transmit jobs to your machine.

RS485 Converters (model 485-1A)

This converter is used instead of LRC cards and are plugged into a serial (COMM) or USB port on the computer with the RS232 side of the converter fitted to the computer. The LRC cable then plugs into the RS485 side of the converter.

On some computers it may be necessary to use a DB25-DB9 adaptor though most new computers only have 9 pin (DB9) outputs on your COMM port.

The RS232-RS485 connector is a PC based device. It will not work on a Mac system.

If you are supplied with a converter that uses USB connection then you will need to load the USB driver. The driver is located on the ToolPath CD. When you first plug in the converter to your computer the Windows' **New Hardware Found Wizard** will start up. Simply follow the on-screen instructions to load the driver off the ToolPath CD.

Caution must be exercised when wiring cables. If you get the orientation wrong you will blow the converter!

LRC (Long Range Communications) Card

The card is designed for use on PC based systems and must be inserted into an 8bit ISA slot or a 16bit VESA slot on your computers' mainboard. The LRC card is not compatible with Macintosh systems.

Generally you will not need to make any adjustments to the Port and IRQ settings on the card, however in some cases hardware conflicts may occur with devices such as sound cards and network cards. In these instances try removing jumper from the IRQ section of J3.

Below is a reference table showing the values that you can set on the LRC card for Port and IRQ addresses.

PORT			***IRQ***	
1	Com 3	3E8	2	Video
2	Com 4	2E8	3	Com 2
3	Special	2E0	4	Com 1
4	Com 2	2F8	5	LPT 2
5	Com 1	3F8	7	LPT 1

Special Notes

The LRC card is "transparent" to Windows '95/98. This means that there will be no reference to the card in your Windows Device Manager.

The LRC card is not compatible with Windows 2000/XT/NT systems.

LRC Cable Configuration for LRC cards

The communications cabling that links your computer to your Multicam machine is made from a 3 pair shielded cable.

The diagram shows the pin configurations for making the LRC cable required to connect the LRC card in the computer to the Multicam controller.

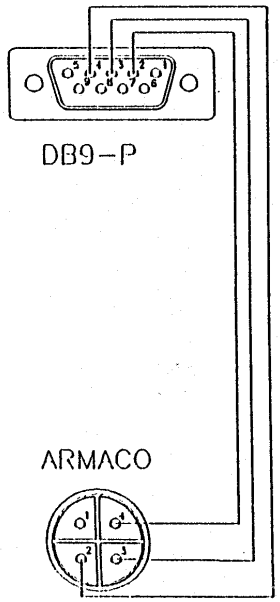
The DB9-P connector (male) plugs into the LRC card at the computer and the ARMCO connector (female) plugs in to the control box at the machine.

Direct wiring to corresponding pins is given in the table below for clarity.

Pin Configurations for LRC Cable when using LRC cards	
DB9-P	ARMCO
2 (White)	4 (White)
3 (Green)	3 (Green)
4 (Black)	2 (Black)

The LRC cable should also be grounded to the control box on the machine by adding a spade connector to the earth (unshielded) wire of the cable.

This will help reduce possible RF interference from external sources and poorly earthed buildings.



Wiring connections as viewed from the rear of the connectors.

LRC Cable Configuration for RS485 converters

RS485-1A Pin configuration for LRC Cable	
DB9-Pin connector	ARMCO 4 pin connector
1 (Red)	1(Red)
2 (Green or Yellow)	2 (White or Orange)
3 (White or Orange)	3 (Green or Yellow)
5 (Black)	4 (Black)

RS485-1A Pin configuration for Control Box connector	
AMC 4 Pin connector	ARMCO 4 pin socket
1 (Black)	1(Red)
2 (Green or Yellow)	2 (White or Orange)
3 (White or Orange)	3 (Green or Yellow)
4 (Red)	4 (Black)

APPENDIX B

Setting a Datum Position using Function 12

Function 12 will home to your (mechanical) machine origin. You may also use Function 12 to set a datum point which will enable you to easily find your starting point for each job.

Setting a datum point means that you avoid manual positioning your machines' start point for each job. It also benefits you by having a reference point in case you encounter any problems, such as a power failure or tool breakage during a job or wish to match two or more jobs accurately.

To set your machine origin follow this procedure;

- Perform a **Function 12** at the machine.
- Manually position the head to the datum point you wish to set as the new machine origin. For instance this may be the corner point of a jig.
- Jog the machine up using the ▲ (up) key. The display will read the X and Y position of the machine. Write these settings down.
- Go to your computer and choose **Setup** from the main menu.
- Choose the Calibration Tab.
- Add the X and Y values to the settings in Home Position if X and Y are set to 0.00 0.00 simply enter the values gathered from the machine.

- Click OK. You will be asked if you wish update the control parameters, choose Yes and the operating instructions will now be sent to the machine.
- Go to the machine and complete a **Function 12** and check the position of the new machine origin. If it is incorrect repeat the procedure until it is correct.

APPENDIX C

Disaster Recovery

Router bits can break due to a number of circumstances. Dullness of edge, mister sprayer unit stops, to much vibration in the sheet etc. Hopefully these problems will be infrequent after a little bit of operator's experience, but they will still happen. As material is expensive we will need to be able to recover jobs if possible.

The bit broke part way through the job.

This is the most common problem. You are cutting several dozen shapes and the bit breaks half through the 20th shape. By the time you notice the problem and run over to pause the job you are missing part of the cut. The following procedure will get you going again quickly:

Note: *On more expensive material it is often a good idea to use Function 12 to set a datum point to register where all your jobs start. Refer to Appendix B for more information.*

- Pause the job by pressing the [STOP] key on the sub console. Take note of the shape number that the pause is on.
- Stop the spindle and change the cutter. If required move the head using the jog keys to a convenient position.
- Once the cutter has been changed you will need to perform a function 4 and function 7 and Function 8 (if required).
- Press the [ENTER] key on the sub console then type in the shape number you wish to start at when prompted. If you select the incorrect shape number, just press [ENTER] again and the machine again will ask you for a shape number. If you need to count the shapes remember to count inside and outside shapes.
- Once at the right shape, ensure that the spindle is on and press [START]. The job will now continue from the point just before the bit broke.

Job Recovery

The Multicam CNC controller incorporates features which enable the operator to recover the exact start position of a job in the event of sudden power loss or if the machine loses position for some other area. In order for this feature to work the operator must do two things;

- When the machine is first powered up a Function 12 (refer to page 115 for more information on Function 12) must be executed, this will enable “soft limits” as described on page 120 as well as enable the job recovery features.

- Before a job is started a Function 3 (refer to page 112 for more information on Function 3) must be performed which records the job start position in relation to the Function 12.

If a job fails and loses position for any reason the original job start position can be easily recovered by following the steps below;

- Perform a Function 12. The machine will seek the soft limits.

- Perform a Function 13. The machine will move to the position stored by the Function 3 setting. Refer to page 116 for more information on Function 13.

The job can now be re-run in its’ entirety or you can start the job from some other point by commencing from a different shape number as described on page 143 or the previous section in this appendix.

APPENDIX D

ToolPath Software Hot Keys

The ToolPath software has a large number of Hot Keys that allows the operator to activate commands by pressing a key on the keyboard rather than using the mouse to select menus then commands.

As a general rule of thumb pressing the letter on the keyboard that corresponds to the *first* letter of the command you want will activate that command. For example pressing **E** on the keyboard will activate the **Edit** menu. Pressing **E** again will activate the **Edit**|**Erase** command. Pressing the **Esc** key will cancel the current command/menu.

Below is a list of hot keys and their actions.

Main Menu

F	Activate File Dialogue
E	Activate Edit Menu
Z	Activate View Menu
A	Activate Analysis Dialogue
L	Activate Layout Dialogue
O	Activate Output Dialogue
S	Activate Status Dialogue
G	Activate Special Menu
Z	Activate Zoom Menu
Q	Edit|Sequence
X	Exit ToolPath

Edit Menu

N	Next Shape
P	Previous Shape
R	Reverse Direction
S	Sequence
M	Move Shape
ESC	Exit

View Menu

S	Shape
A	All
W	Window
L	Left
R	Right
D	Down
U	Up
ESC	Exit

Special Menu

3	Engrave 3D
2	Engrave 2D
ESC	Exit

Edit|Sequence

G	Group
A	Advanced
ESC	Exit

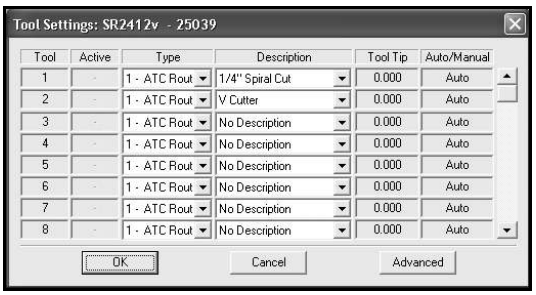
APPENDIX F

Setting Up Multiple Tool Jobs

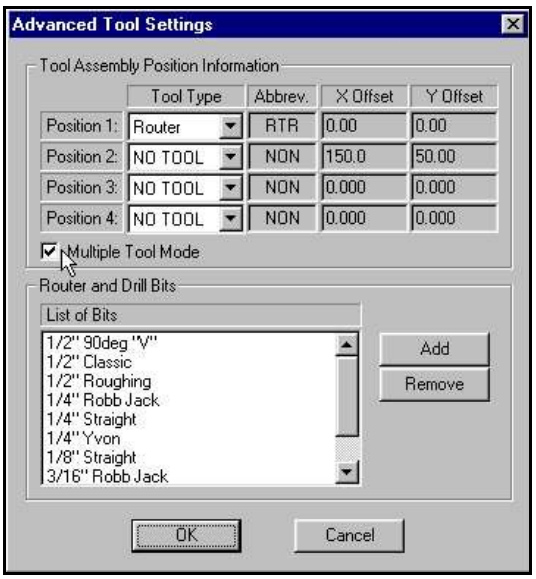
It is possible to set up jobs that require multiple tool changes when using a Quick Change spindle or on machines fitted with an Automatic Tool Change (ATC) unit. Before you can run a multiple tool change job you need to program ToolPath with information about the number and types of tools available. In essence you need to create a Tool Library. This procedure is the same for all machines.

Note: When you are carrying out this procedure your machine must be on and connected to your computer and communicating.

Entering Tooling Information



- Select **SETUP** from the Main Menu, then click on **Configure Tooling** on the Setup dialogue box. Click on the **Advanced** button on the Tool Settings dialogue box. Another dialogue box will appear where you can enter information about the tools that you have.

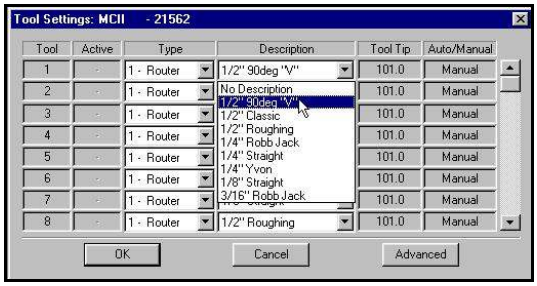


- To activate Multiple Tools check the Multiple Tool Mode check box in the Advanced Tool Settings dialogue box.

- Enter the list of tools you will be using by clicking the **Add** button and typing in a descriptive name for each tool.

- After you have entered your list of tools click **OK**, this will return you to the Tool Settings dialogue box.

- On the Tool Settings dialogue choose the tool from the list of available tools that appear in the description column. The description should match the actual tool that you will be using. You will need to do this for each tool that you will have available. You should take this opportunity to make all the tools available. This will make it easier to select tooling in the Groups Setup.

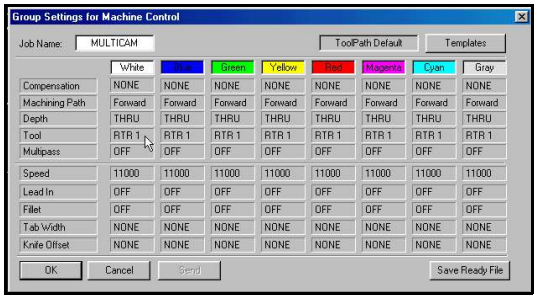


- When all your tools are selected click **OK** to get back to the Setup dialogue box. Click **OK** again and then click **YES** to update the controllers parameters.

Setting Up A Job With Multiple Tools In ToolPath

A job that requires the use of multiple tools must be set up in such a way so that ToolPath knows which tool to use to machine which tool path. This is done in the **GROUPS** menu (refer to the section on **GROUPS** commencing on page 85 for information on assigning tool paths to group colours). Once the job has been set up in this way all that remains to do is to assign the correct tool to the correct group.

In the **GROUPS** menu click on **Setup**. The Group Settings for Machine Control dialogue box will appear.



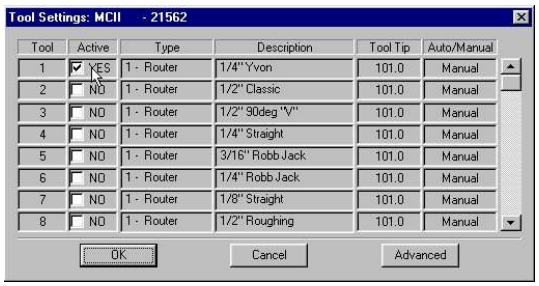
For each of the group colours you can select a tool as assigned previously.

Here you can decide to setup a methodology where a particular tool is always associated with a particular group. The advantage of this is that your design process can be somewhat streamlined by setting standard procedures.

For example a 3mm spiral cutter may be associated with the White group which is always associated in your designs with cutting straight through your material, and a 90° Vee groove cutter may always be associated with the Blue group.

Or you can treat each job on an individual basis as far as tool assignment is concerned.

Click on the Tool field in the group column that you wish to assign a tool to. The Tool Settings dialogue box will appear.



Select the tool you wish to assign to the group by clicking on the check box to enable the tool. Here we are assigning a ¼" Yvon cutter (tool 1) to the white group. Repeat the process for as many tools or groups as you wish.

Finally sequence the job by using the **Sequence by Group** command in the **EDIT** menu.

Setting Up And Running Multiple Tool Jobs At The Machine

Once the job has been correctly set up and transmitted to the machine the job name will be displayed on the machine sub-console with the tool number/s following the job name;

T1, T2,T3, Jobname

This indicates that there are three tools programmed in the design and which tools these are. It is imperative that the tool types assigned to T1, T2 and T3 on the factory floor match the tool assignments in your ToolPath setup.

If you have used a setup jig to set all your tools to the same length in the tool holders you can carry out functions 1, 4, 7, 8 etc as per normal.

Press [START] to commence running the job. You will be prompted when to insert the correct tool. Once the correct tool has been inserted press the [ENTER] key to continue machining.

If your tools are all set at different lengths you must use Function 25.This function allows you to set up the length of your tools in relation to each other. It negates the need to use setting jigs to set all your tool lengths at the same length. You can set up to 16 different Tool Tip Offsets.

Refer to page 121 for more information on using Function 25

Important Safety Note

If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to “wobble” the tool holder it is not correctly engaged.

APPENDIX G

Setting Up Multiple Heads and Tools In ToolPath

Note: When you are carrying out this procedure your machine must be on and connected to your computer.

- Select **SETUP** from the Main Menu, click on the **Configure Tooling** button then click on the **Advanced** button. (refer Fig 1.)
- To activate Multiple Tools check the Multiple Tool Mode check box. (refer Fig 2.)
- Enter the list of tools you will be using by clicking the **Add** button and typing in a descriptive name for each tool.
- To activate the second head change the **Tool Type** from NO TOOL to Router in position 2 on the Advanced Tool Settings dialogue. You will need to enter the X and Y offset for the second head in relation to the first head. Put this value into the relative field in the dialogue box.
- After you have entered your list of tool types and activated the second head (position 2) by choosing a tool in the Tool Type column from the drop down list click **OK**.
- On the Tool Settings dialogue choose the tool from the list of available tools that appear in the description column. The description should match the actual tool that you will be using. You must also choose in which position (which Head it's on) the tool will be put into.
- When all your tools and positions are selected click **OK** to get back to the Setup dialogue box. Click **OK** again and then click **YES** to update the controllers parameters.

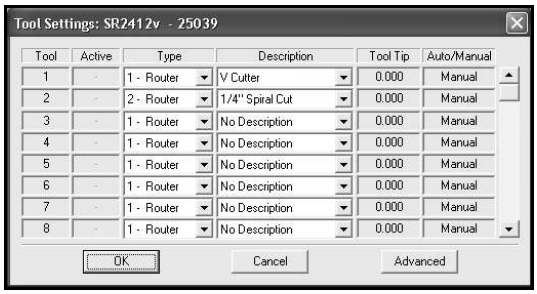


Fig 1.

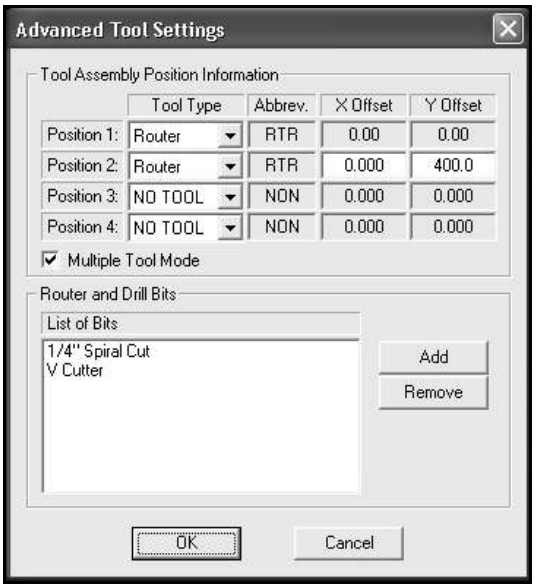


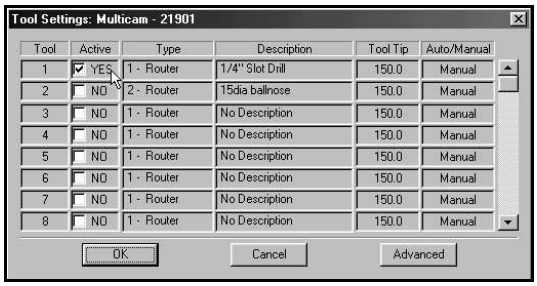
Fig 2.

Running A Job Using Multiple Heads and Tools

A job that requires the use of multiple tools must be set up in such a way so that ToolPath knows which tool to use to machine which tool path. This is done in the **GROUPS** menu. Once the job has been set up in this way all that remains to do is to assign the correct tool to the correct group.

In ToolPath

- In the **GROUPS** menu click on **Setup**.
- Assign each tool to its own group colour.
- In the **EDIT** menu select **Sequence** and then **Sequence by Group**
- In the **Output** screen select the tool for each group by clicking on the tool selection box of each group
- Select the tool you wish to assign to the group by clicking on the check box to enable the tool. Here we are assigning a ¼" slot drill (tool 1,head 1) to the white group. The number before Router applies to which head it's associated with Repeat the process for as many tools or groups as you wish.



Setting Up And Running Multiple Head Jobs At The Machine

Once the job has been correctly set up and transmitted to the machine the job name will be displayed on the machine sub-console with the tool number/s following the job name;

T1, T2, T3 Jobname

This indicates that there are three tools programmed in the design and which tools these are. It is imperative that the tool types assigned to T1, T2 and T3 on the factory floor match the tool assignments in your ToolPath setup.

- If you have used a setup jig to set all your tools to the same length in the tool holders you can carry out functions 1, 4, 7, 8 etc as per normal. Otherwise you will need to use function 25.
- Press [START] to commence running the job. You will be prompted when to insert the correct tool. Once the correct tool has been inserted press the [ENTER] key to continue machining.

If your tools are all set at different lengths you must use Function 25.This function allows you to set up the length of your tools in relation to each other. It negates the need to use setting jigs to set all your tool lengths at the same length. You can set up to 16 different Tool Tip Offsets.

Note: All tools positioned in Head 2 must be put into head 2 to do your Function 25.

Refer to page 121 for more information on using Function 25

APPENDIX H

Setting Up Automatic Tool Changers

The Multicam Automatic Tool Changer (ATC) allows you to program a job so that tool changes can be made automatically without operator intervention.

The Tool Changer is available as either linear style or rotary style. Perfectly suited to applications where a single job may require a number of different types of tools the ATC unit can make complex profiled jobs easy.

Setting up the ATC unit is a two stage process. Firstly you must configure ToolPath for ATC operation then you must setup the ATC unit on the machine itself.

Configuring ToolPath for ATC Operation

Note: When you are carrying out this procedure your machine must be on and connected to your computer.

- Select **SETUP** from the Main Menu, click on the **Configure Tooling** button then click on the **Advanced** button. (refer Fig 1.)
- To activate Multiple Tools check the Multiple Tool Mode check box. (refer Fig 2.)
- Enter the list of tools you will be using by clicking the **Add** button and typing in a descriptive name for each tool.
- To activate the ATC unit change the **Tool Type** from Router to ATC Router in position 1 on the Advanced Tool Settings dialogue (refer Fig 2.).
- After you have entered your list of tools and activated the ATC unit click **OK**.

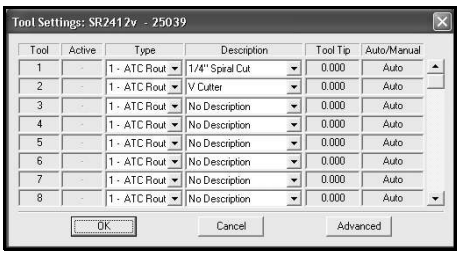


Fig 1.

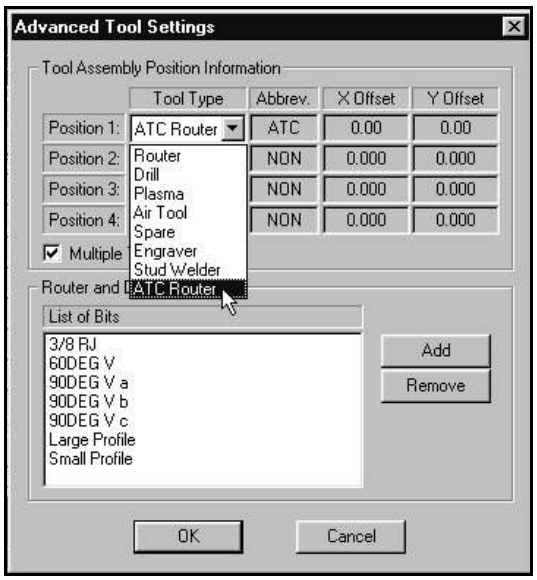


Fig 2.

- On the Tool Settings dialogue choose a tool for each of the ATC positions from the list of available tools that appear in the description column (refer Fig 3.). The description should match the actual tool that you will be using.

- When all your tools and positions are selected click **OK** to get back to the Setup dialogue box. Click **OK** again and then click **YES** to update the controllers parameters.

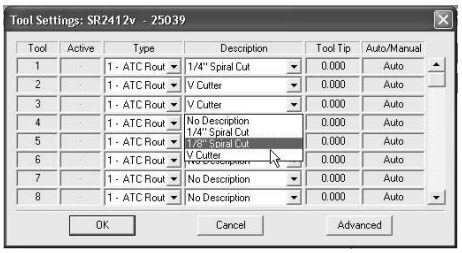
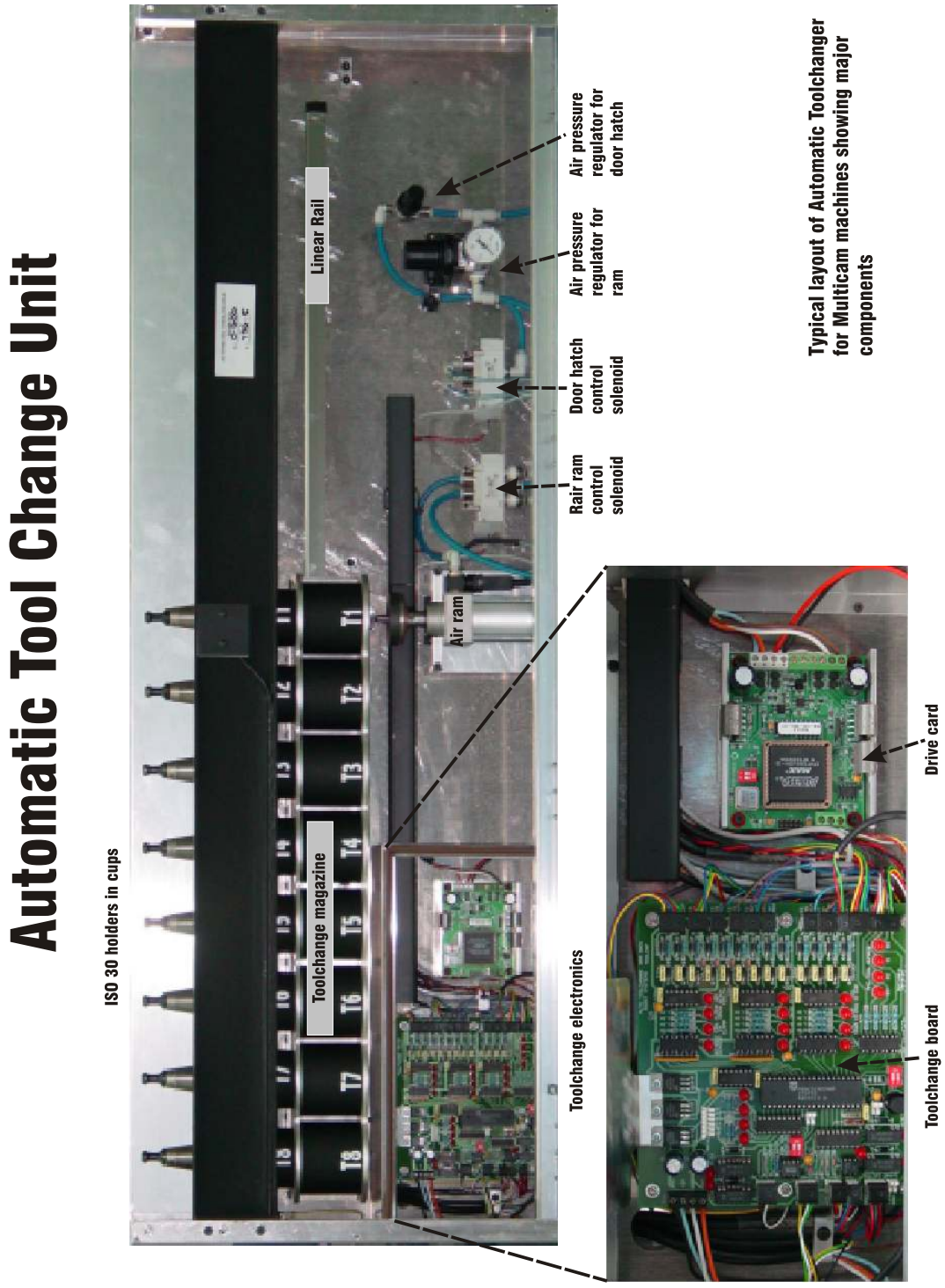


Fig 3.

Once ToolPath has been configured for ATC operation you can select different tools to cut different parts of your job. In order to do this you must assign the parts of your job that requires a different tool to a different GROUP. You can do this in either the GROUPS|SETUP menu or the OUTPUT menu. For details on Groups refer to page 85and page 87 for information on how to assign tools to groups.

The next stage is to set up the ATC unit itself at the machine.

Linear Toolchanger Layout



Setting up the LINEAR Automatic Tool Changer

- Perform a function 12 at the machine to return it to the home position.

- Type in [FUNC] 350 [ENTER] The display will now read *Bypass ATC? NO* Press [ENTER] The display will now read *ATC Type: LINEAR TYPE 1*. Press [ENTER].

- The display will read *Number Of Tools: 8*. Enter the number of tools that your machine has eg 6 and press [ENTER] .

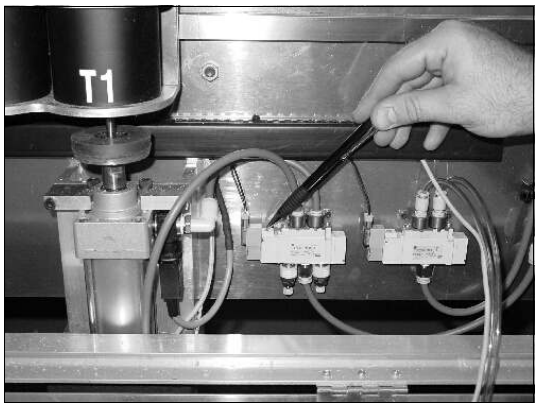
- The display will read *Jog To Locate Taper*, now press the [FUNC] Key. The display will read;
  F6 enter x move: +0.000
  Type in the x co-ordinate for the machine to move and press [ENTER] . This value is the distance that the machine needs to be moved from its current position so that it is in line with the opening in the tool changer mechanism.

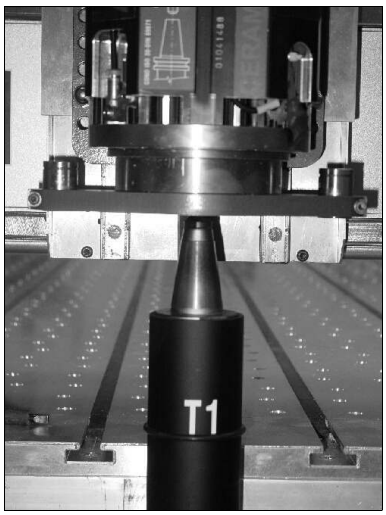
- The display will read ;
  F6 enter y move: +0.000
  Type in the y co-ordinate for the machine to move and press [ENTER]. This value is the distance that the machine needs to be moved from its current position so that it is in line with the opening in the tool changer mechanism.

  Note: If you are resetting these distances look inside the door of the Automatic Tool Changer. The x and y co-ordinates may already be written there or refer to the table later in this section.

- The display will read *Jog To Locate Taper* and the spindle will be located above the Tool Changer. You will now have to test the position. Open the Tool Changer main door with the key supplied. Positioned under the tool cups are 2 air solenoids.

  The left solenoid activates the air to lift the tool cup up and down. Press the button as pictured to raise the cup. The cup will lift up and should place the tool in the spindle. Activate the green ATC button on the side of the spindle to insert the tool. Repeat this and take the tool back out. You should make sure the tool fits correctly into the cup to avoid any problems later.





Note: On 250mm axis machines you will have to jog the head down so the air cylinder will reach the spindle.

When tool and cup fit neatly together press [ENTER].

- The display will now read *No F25 Sensor Installed*, using the **+\-** key, toggle to *Table Mount F25* and press [ENTER]. The screen will display *Jog to F25 sensor*. Jog the head of the machine to the centre of the sensor and press enter. The display will now show *SAVING O/S TO FLASH MEM*, once this is complete the tool changer will be ready for use.

- After the above set up has been completed you must now perform a **Function 352.** The machine will now automatically proceed to pick up each tool and seek the tool tip sensor. It is recommended that this cycle is left to finish with no interruptions.

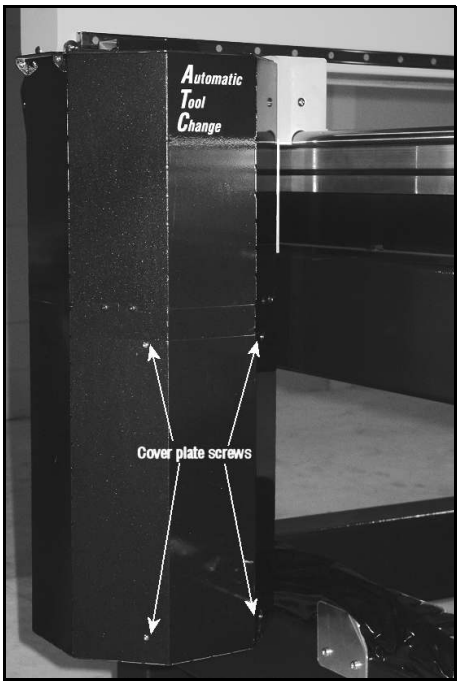
Note; Before performing a **Function 352** you must have all your selected tools loaded in the tool change magazine. Insuring the corresponding tools in the magazine match up with the list of tools in the *Advanced Tooling Setup*.

Finally enter the X an Y positions for the ATC for future reference in the table below.

Automatic Tool Changer X & Y Co-ordinates	
X Position	
Y Position	

Setting up the ROTARY Automatic Tool Changer

The Rotary Tool Change Unit was developed as a means of effecting faster tool changes on large machines. The tool changer is attached to the machine gantry, opposite the operators side. Because it travels with the machine gantry this means that a tool change can be executed at any position along the machine without the machine having to return to a fixed position on the machine. On machines with large process areas this can lead to considerable time savings in job cycles.



The rotary tool change unit is basically the same as the linear unit except that the tool rack is wrapped around a circle. Operation of the unit is the same as that of the linear tool changer but because of space constraints the layout is different in that the mechanical aspects of the unit are physically located above the electronics of the unit.

Setting up the Rotary tool changer in Toolpath is exactly the same as described at the beginning of this chapter, however there are some small differences in setting up the unit at the machine.

In the event that you have to re-set up the Rotary Tool Change unit follow the steps below.

- Perform a function 12 at the machine to return it to the home position. Using the arrow keys on the sub-console drive the machine as far as possible to the lower left corner of the machine as viewed from the operators side. (This will be to the left of the corner locating bracket)

- Type in [FUNC] 350 [ENTER] The display will now read *Bypass ATC? NO* Press [ENTER] The display will now read *ATC Type: LINEAR TYPE 1*. Use the [+/-] key change the display to read *ROTARY TYPE 2*. Press [ENTER].

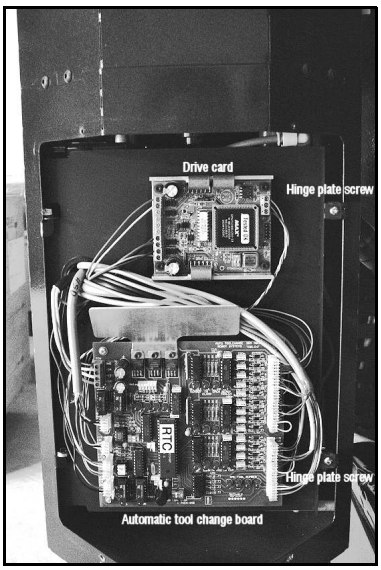
- The display will read *Number Of Tools: 8*. Enter the number of tools that your machine has eg 8 and press [ENTER] . The machine will now return to the home position

  Open the top flap of the tool change unit with the key provided and note the value of the Y axis co-ordinate that is written on the label. For Rotary tool change units the X axis value will always be 0.

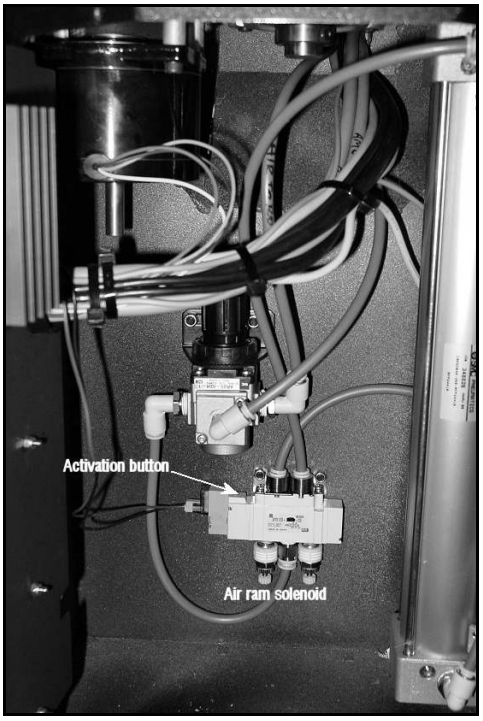
- The display will read *Jog To Locate Taper* Jog the machine using the Y axis keys so that the spindle is located directly above the tool change door and the numbers on the display match those written inside the tool change unit. Press [ENTER]. You can use the [+/-] key to reduce the travel speed of the machine for increased accuracy.

- You will now have to test the position. Remove the cover plate on the Tool Changer by removing the 4 screws as shown in the picture above. This will reveal the tool changer main board and the carousel drive card.

- Remove the hinge plate screws as indicated to reveal the solenoid and air pressure assembly behind.



- The solenoid activates the air to lift the tool cup up and down. Press the button as shown in the adjacent picture to raise the cup. The cup will lift up and should place the tool smoothly in the spindle. Activate the green ATC button on the side of the spindle to insert the tool. You should make sure the tool fits correctly into the cup to avoid any problems later. If it appears that the tool is not being delivered in to the spindle squarely you may need to use the Y axis keys to re-adjust the position of the head.



- When tool and cup fit neatly together press [ENTER] and note the co-ordinate readings on the sub-console display
- The display will now read *Jog to F25 sensor*. Using the direction keys on the sub-console jog the machine so that the head is located centrally above the tool tip sensor then press [ENTER]
- Remove the tool from the spindle using the green button on the side of the spindle to eject the tool and return it to the tool carousel.
- Ensure that your tool change carousel is fully loaded with tools then perform a function 352. The machine will now move over to the tool changer, pick up tool 1 and register its length with the tool tip sensor. This will be repeated until all the tools have been "tipped off".
- Finally enter the X an Y positions for the ATC for future reference in the table below.

Automatic Tool Changer X & Y Co-ordinates	
X Position	0.00
Y Position	
Z Position	

Bypass the Automatic Tool Changer

In the event that the tool changer unit becomes inoperable it is possible to continue using the machine by temporarily bypassing the tool changer using FUNC350.

To temporarily disable the tool change unit follow these steps;

- Type in [FUNC] 350 then press [ENTER]
- The LCD display will read *ATC Bypass? No*.
- Press the [+/-] key to change *No* to *Yes*
- Press the [ENTER] key.
- The LCD display will read *Saving to flash*
- The machine will now operate as if you did not have a tool changer fitted. If your jobs require a tool change the LCD will prompt you to manually insert the new tool when needed.
- It is not necessary to re-send your jobs to the machine if you have temporarily disabled the tool change unit.
- Continue to use your Toolpath software as if the tool changer is still active.

To re-enable the tool change unit follow these steps;

- Type in [FUNC] 350 then press [ENTER]
- The LCD display will read *ATC Bypass? Yes*.
- Press the [+/-] key to change *Yes* to *No*
- Press the [ENTER] key.
- The LCD display will read *Saving to flash*
- The tool changer is now re-enabled.
- It is not necessary to re-setup the tool changer

Note:

If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to "wobble" the tool holder it is not correctly engaged.

Running A Job Using Multiple Tools

A job that requires the use of multiple tools must be set up in such a way so that ToolPath knows which tool to use to machine which tool path. This is done in the **GROUPS** menu. Once the job has been set up in this way all that remains to do is to assign the correct tool to the correct group.

In ToolPath

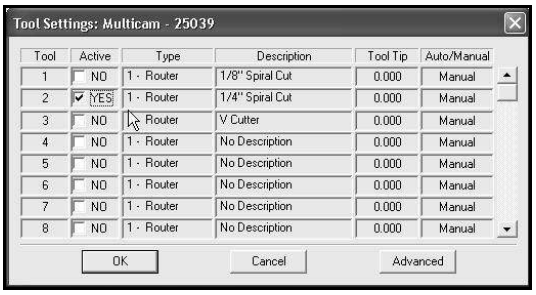
- In the **GROUPS** menu (or alternatively in the **SPECIAL** menu) **MOVE TO GROUP** or **COPY TO GROUP** the particular shapes you wish to machine with different tools.

- If needed perform the individual commands with the corresponding group colours. ie; drill holes , engraving etc.

- It is highly recommended to now go to the **EDIT** menu select **Sequence** and then **Sequence by Group.** This will minimise any unnecessary tool change moves within the job.

- In the **Output** screen assign each tool to its own corresponding group colour, select the tool for each group by clicking on the tool selection box of each group

- Select the tool you wish to assign to the group by clicking on the check box to enable the tool. Here we are assigning a 1/4" dia Spiral Cutter (tool 2) to the blue group. Repeat the process for as many tools or groups as you wish.



Setting Up And Running Multiple Tool Jobs At The Machine

Once the job has been correctly set up and transmitted to the machine the job name will be displayed on the machine sub-console with the tool number/s following the job name;

Job TEST1 T1, 2, 3

The above example indicates that there are three tools programmed in the design and which tool numbers these are. **It is imperative that all the tool types assigned on the factory floor match the tool assignments in your ToolPath setup.**

- Proceed normally with the standard functions Function 12, Function 1 etc. When a Function 4 is applied, the machine will automatically pick up Tool 1. The next prompt will be **Automatic FUNC 25 yes/no** use the +/- key to select **no** as the Tool Tip Settings have automatically been completed with Function 352. Function 8 is also referenced from Tool 1. **(Note: The depth settings for all tools are referenced from tool 1.)**

- Press [START] to commence running the job. Don't be alarmed if Tool 1 is still in the spindle when you push start and your sequence starts with a different Tool, or in fact Tool 1 is not even to be used in a particular job. The machine will automatically seek and collect the correct tool.

*The following may help you understand how **Tool Tip Settings** work (more accurately known as **Tool Tip Off Settings**).*

If your tool change magazine is full of tools of varying lengths, for an example lets use two tools with cut edge lengths of great variance ie; 3mm diameter straight cutter with 8mm cut edge length and a 12mm diameter cutter with a 40mm cut edge length.

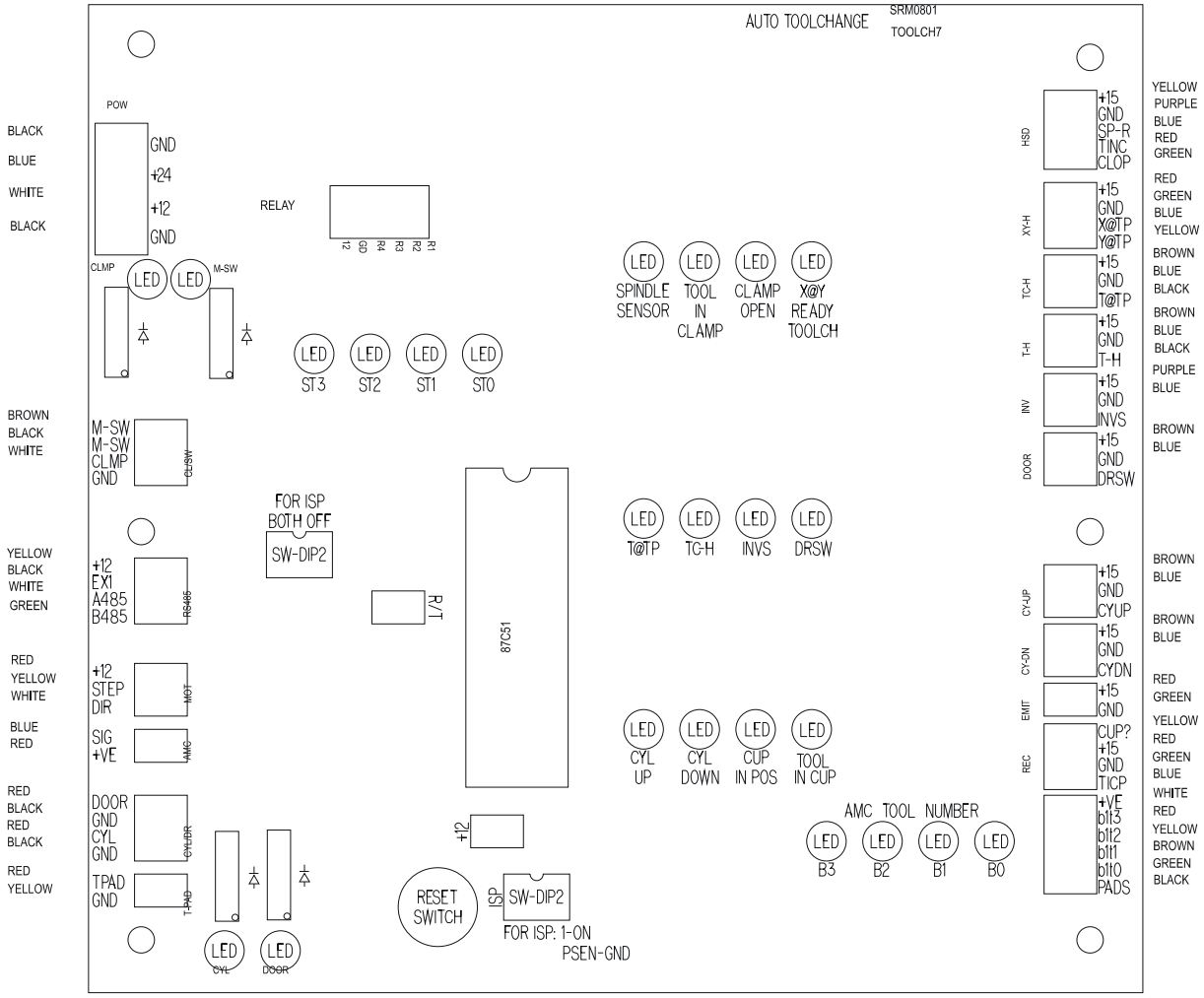
To maintain a constant accurate cutting/engraving depth within each tool opposing to another the Z axis must offset in either a positive or negative value. Hence in our above example to maintain a constant accurate cutting depth between the two tools the Z axis must travel a further distance in a negative value to drive the 3mm tool to the same depth as the 12mm tool.

Note:

If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to "wobble" the tool holder it is not correctly engaged.

Automatic Tool Changer Board (SRM801)

AUTOMATIC TOOL CHANGE BOARD LED DESCRIPTIONS	
ST3	ATC Status LED
ST2	ATC Status LED
ST1	ATC Status LED
ST0	ATC Status LED
SPINDLE SENSOR	LED will be in either the **ON** or **OFF** position when the HSD is not rotating. When the HSD is rotating to LED will **Flicker**.
TOOL IN CLAMP	LED will be **ON** when the ISO30 holder is loaded in the HSD. LED will be **OFF** when the HSD has no ISO30 holder loaded.
CLAMP OPEN	LED will be **ON** only when the HSD clamp is open while loading or unloading a tool or pressing the green tool button.
X @ Y READY TOOLCHANGE	LED will be **ON** when the machine is at the ATC position.
T@TP	LED will be **ON** to indicate the ATC rack is in position.
TC-H	LED will be **ON** when ATC rack is at its home position.
INVS	LED will be **ON** when HSD is running.
DRSW	LED will be **ON** when ATC door is open.
CYL UP	LED will be **ON** when the ATC air ram is in the UP position.
CYL DOWN	LED will be **ON** when the ATC air ram is in the BOTTOM position.
CUP IN POS	LED will be **ON** to indicate that a tool cup is in the ATC rack for the corresponding tool.
TOOL IN CUP	LED will be **ON** to indicate that a ISO30 holder is in the toolcup for the corresponding tool.
B3	LED used for tool number. *Refer to binary chart below*
B2	LED used for tool number. *Refer to binary chart below*
B1	LED used for tool number. *Refer to binary chart below*
B0	LED used for tool number. *Refer to binary chart below*



BINARY CHART TOOL NUMBER ASSIGNMENTS				
TOOL NUMBER	B3	B2	B1	B0
1				ON
2			ON	
3			ON	ON
4		ON		
5		ON		ON
6		ON	ON	
7		ON	ON	ON
8	ON			

APPENDIX I

Multicam Vacuum Tables

A Vacuum Hold-Down table can be a valuable addition to your Multicam machine.

In most cases you can avoid using manual clamping systems and simply “suck” your material onto the machine’s process bed. Vacuum hold down is not ideal for all applications. It is most useful for either high volume repetitive projects or large parts.

Theory Of Operation

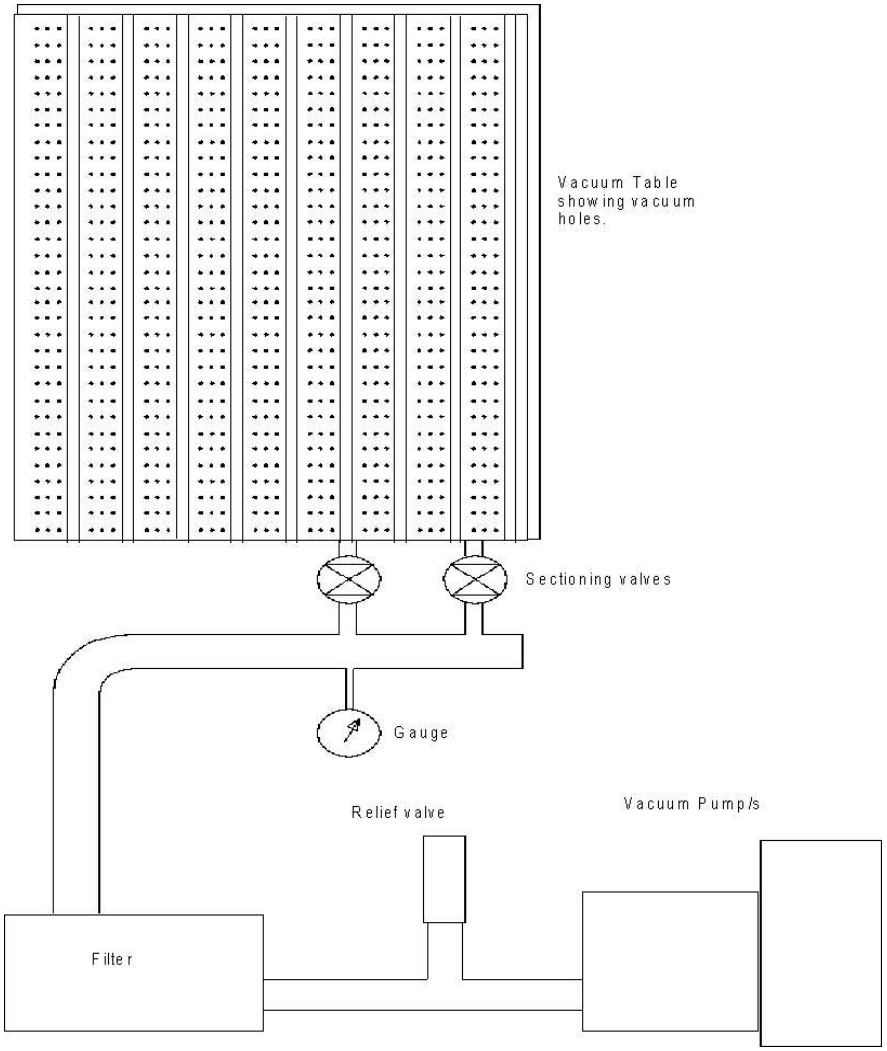
The basic concept of a vacuum table is simple enough: Apply a vacuum to the back of a part and it will stick due to atmospheric pressure. In actual fact it is not the vacuum "sucking" the pieces down to the table top, but rather the atmosphere pushing down on the piece that holds it in place. Having said that there are a few things you must keep in mind when you use a vacuum system:

How much force do I get? This is simply the amount of vacuum multiplied by the area you are applying the vacuum to:

> Most vacuum pumps supply about 10 to 15 millimetres of Mercury commonly shortened to “mmHg”. 1 mmHg is equal to about 40 grams per square centimetre.
>
> This means if we are holding down a part that is 300mm square:
>
> 300mm x 300mm x 0.5 PSI/mmHg x 10 mmHq = **327kg**!
>
> That is quite a lot of power! However this formulae will work against you if your part is very small, say only 25mm by 50mm:
>
> 25mm x 50mm x 0.5 PSI/mmHg x 10 mmHg = **4.5kg**.
>
> A part this small will likely not hold very well if at all.

So far we have assumed that we are getting about 10 mmHg of vacuum. This is true of most pumps regardless of size. As soon as we start introducing leakage to the system the pump must be able to suck away the air flowing into the system as fast as it leaks in, if the pump cannot keep up the vacuum will start to drop off as will the holding force. Generally a good rule of thumb is that a leak 6mm in diameter will require ¼” HP to maintain vacuum. If you have just 20 such leaks (not much over a surface area of 10 or more square metres) you will require 5 HP of capacity just to keep up. You have two choices: plug the leaks or get a bigger pump! It is usually better to control the leakage.

Standard Vacuum Table Layout



Vacuum Table The standard vacuum table has a series of holes machined in the top of the table through which air is drawn. The table is sectioned off according to the number of control valves giving active "zones" on the table.

Sectioning Valves allow the operator to activate or deactivate sections of the tables vacuum grid.

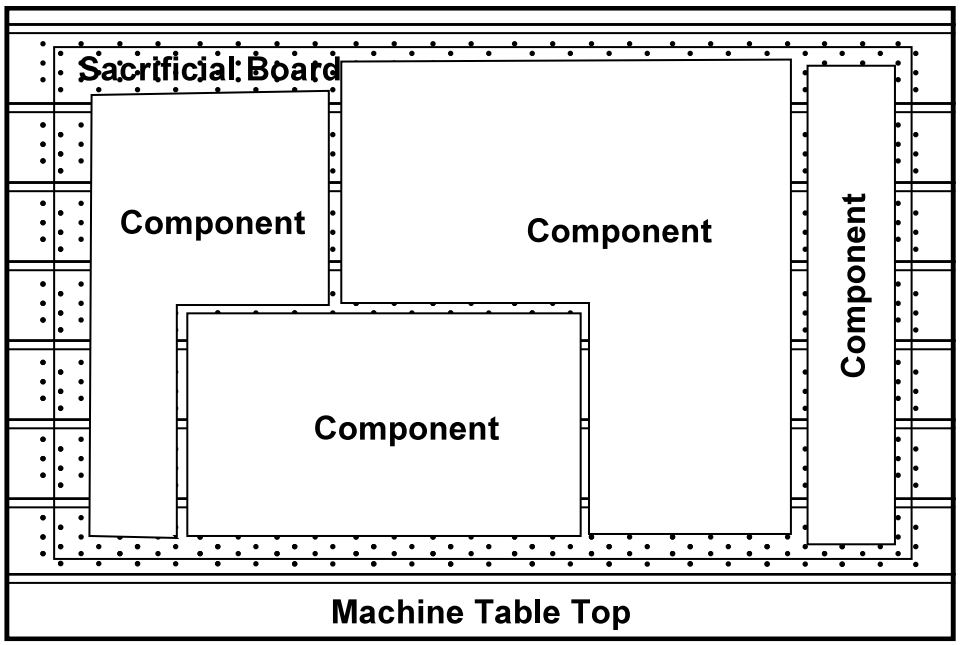
The Filter protects the vacuum pump from dust and particles.

A Pressure Relief Valve ensures that some air will flow through the vacuum pump. Air flow is what cools the pump so if you cut off the air supply with the control valves the relief valve will open and allow a minimum air flow.

The Vacuum Pump supplies the vacuum for the entire system.

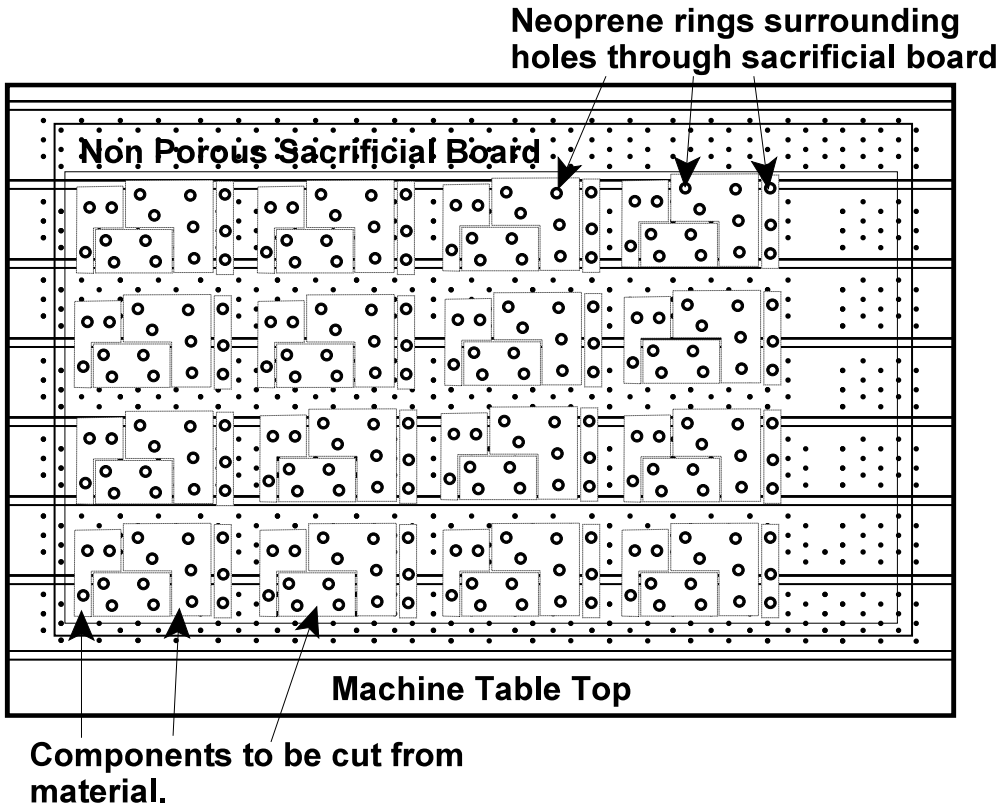
The standard type of vacuum table supplied is known as **Porous Sacrificial Sheet** type. A porous material such as particle board or MDF is used as the sacrificial sheet. This allows the vacuum to pull right through the sacrificial sheet and clamp the material above it down. The advantage of this method is that it is easy, just throw your sacrificial material down, your process material on top, flip on the vacuum and start cutting! The disadvantage is that it can be rather in-efficient. You will not get full vacuum through the porous material so your holding force is reduced, however if you are cutting large parts there will still be plenty of power. This method also tends to be leaky as areas that are not covered with the process material will leak air and increase your vacuum capacity. This method will work fine on large parts, just remember to replace the sacrificial sheet occasionally because all the cut grooves will cause extra leakage.

There are two other basic styles of vacuum table described further on. These are the **Neoprene Gasket** and **Sacrificial Vacuum Jig** styles. Each has its own strengths and weaknesses. Ultimately it is up to the operator to decide which is the best in any given circumstance.



Neoprene Gaskets

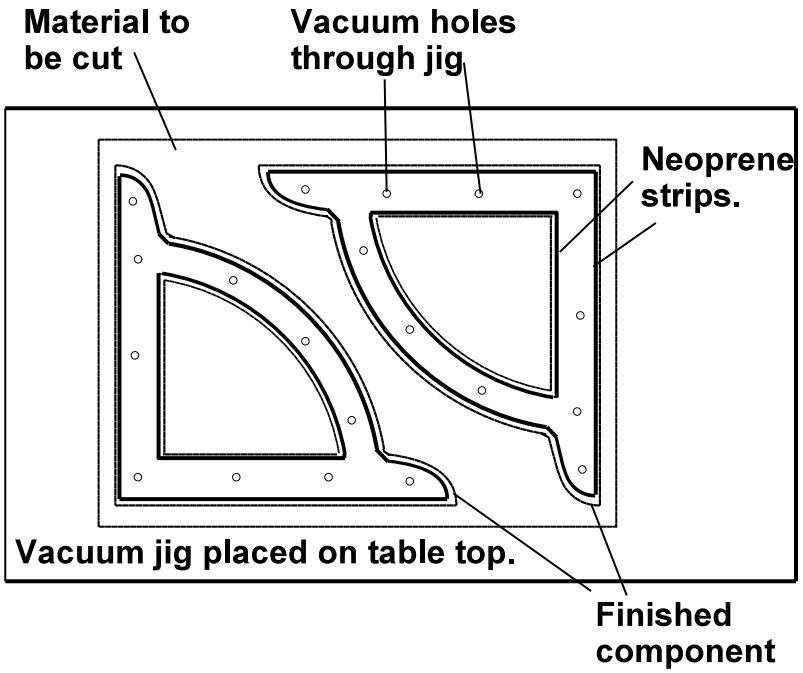
This method is extremely air tight, it will provide maximum holding power with very little pump capacity. It is excellent for small parts. The disadvantage is the set up time and the fact you need the gaskets. It would only make sense for highly repeatable jobs.



Use something non-porous such as plastic as your sacrificial sheet. Drill some holes in it then stick some neoprene tape or donuts around the holes. Place your process material on top of the neoprene tape.

Sacrificial Vacuum Jigs or Call Boards

This is sort a combination of the neoprene gasket and porous sacrificial sheet methods. Basically you use a non-porous sacrificial sheet and cut holes in it such that the holes are beneath your cut out parts. Neoprene or rubber strips are placed around the holes in a shape similar to the component to be cut. These strips must be placed so that they do not get cut when the component is machined out. The diagram below shows how this works. This method is almost as efficient as the neoprene gaskets but requires much greater preparation. You still have to set up the jig so it is only worth the trouble on larger runs of repetitious parts.



Another advantage of this type of arrangement is that it allows you to use small and irregular shaped materials from which to cut your components. This is useful where the routing operation may come after some other machining process such as sawing.

Operating and Maintenance Manual for Perske High Frequency Motors

Operating Instructions

Perske high frequency electric motors are manufactured with precision and thoroughly tested before leaving the factory of Walter Perske GMBH. Due to the high motor speeds as well as demands which are placed on these motors, it is important to observe certain operating procedures to avoid damage and premature failure.

Very Important Upon receipt of your Perske motor, please inspect for damage or missing parts. Report such losses immediately to Multicam Systems.

Spindle Speed Control

To adjust the spindle speed (also known as Revolutions Per Minute, RPM) it is necessary to adjust power frequency getting to the spindle. On machines fitted with an LED display and Spindle Speed Control dial the RPM's are displayed on the LED readout. Adjusting the dial up or down will increase or decrease the speed accordingly within the pre-set range of the spindle inverter.

On machines not fitted with an LED display and Spindle Speed Control dial the RPM's are adjusted by using the up and down arrows on the spindle inverter whilst the spindle is running. On these machines the display is not in RPM's but in Hertz (Hz). The display on the inverter actually indicates the frequency of the power being sent to the spindle. Note that this does not affect the torque or ability of the spindle in any way. It is simply an electronic means of adjusting the RPM of the spindle.

In the latter case it is necessary to convert the Hz reading to a more meaningful RPM figure so that you can calculate the correct speed and feed rates for your cutting job. The Hz and RPM share a linear scale which is to say that the conversion is direct and quite simple.

You can use the table below as a simple guide.

Hz TO RPM CONVERSION FOR PERSKE SPINDLES	
Hertz (Hz) Display	**RPM Conversion**
300	18,000
250	15,000
200	12,000
150	9,000
100	6,000
Based upon a conversion of 60Hz = 300RPM	

Some earlier model machines that had an LED display and Spindle Speed Control dial mounted on the machine actually displayed Hz. In these cases use the above table to convert Hz to RPM and use the dial control to achieve the correct figure.

Tooling

Only use balanced tools and follow the tooling recommendations. Always check the tool to ensure sharpness and balance. Tooling with a vibration effect value greater than 2.8mm/sec (at rated speed) **must not be used**.

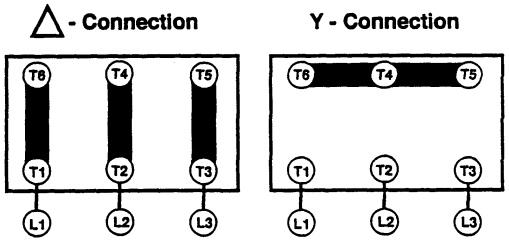
Vibrations

Vibrations dramatically reduce bearing life and will cause premature failure. As such, vibrations must be avoided. Causes of vibration can be:

- unbalanced tooling
- improperly clamped workpiece
- excessive feed rates
- worn or dull tools

Electrical

Before connecting the motor, insure that connecting voltage is the same as stated on the motor's nameplate. Connect wires as shown below making sure that all terminal clamps are tight. Check clamp tightness once a year. All electrical work must be carried out by a licenced electrician.



3.5HP Perske spindles are wired in their terminal block in a *DELTA* (△) pattern.

5.5HP Perske spindles are wired in their terminal block in a *STAR* (γ) pattern.

Inverters

Perske high frequency motors must be driven by high frequency, 3 phase power. Optimum operation and lowest motor temperatures are achieved when using a dynamic frequency inverter delivering a true sinusoidal wave form. If a solid state inverter is utilized, a three phase line reactor must be used to limit voltage spikes and current ripple. This reduces the risk of winding failure, limits operating temperature and reduces motor vibration.

The inverter should never been set to greater than 300Hz

Cleaning

The collet/covernut assembly, spindle shaft taper as well as the fan cover should always be kept clean and free of dust and dirt build-up. If using compressed air to clean tooling and motor of duct or dirt, the motor must be operating at its rated speed to ensure proper sealing protection. Wipe down the collet/covernut and spindle shaft routinely with an oil cleaner.

Conditions

Should the operating conditions of the motor change (temperature rise, noise level, vibration, amperage draw); shut down the motor immediately to avoid further damage and contact the service personnel at Multicam Systems.

Warranty

Walter Perske GMBH warrant the mechanical and electrical components of the spindle free from defects in materials or manufacture for a period of 12 months from the date for purchase, subject to the conditions of use.

Bearings, seals or other components that are subject to normal wear and tear are warranted for a period of six months from the date of purchase, subject to the conditions of use.

Operating and Maintenance Manual for HSD ATC and Collet Motors

Operating Instructions

HSD high frequency electric motors are manufactured with precision and thoroughly tested before leaving the factory of HSD. S.r.l. Due to the high motor speeds as well as demands which are placed on these motors, it is important to observe certain operating procedures to avoid damage and premature failure.

Very Important Upon receipt of your HSD motor, please inspect for damage or missing parts. Report such losses immediately to Multicam Systems.

Spindle Speed Control

To adjust the spindle speed (also known as Revolutions Per Minute, RPM) it is necessary to adjust power frequency getting to the spindle. On machines fitted with an LED display and Spindle Speed Control dial the RPM's are displayed on the LED readout. Adjusting the dial up or down will increase or decrease the speed accordingly within the pre-set range of the spindle inverter.

On machines *not* fitted with an LED display and Spindle Speed Control dial the RPM's are adjusted by using the up and down arrows on the spindle inverter whilst the spindle is running. On these machines the display is not in RPM's but in Hertz (Hz). The display on the inverter actually indicates the frequency of the power being sent to the spindle. Note that this does not affect the torque or ability of the spindle in any way. It is simply an electronic means of adjusting the RPM of the spindle.

In the latter case it is necessary to convert the Hz reading to a more meaningful RPM figure so that you can calculate the correct speed and feed rates for your cutting job. The Hz and RPM share a linear scale which is to say that the conversion is direct and quite simple.

You can use the table below as a simple guide.

Hz TO RPM CONVERSION FOR 3.8KW HSD SPINDLES	
Hertz (Hz) Display	**RPM Conversion**
300	18,000
250	15,000
200	12,000
150	9,000
100	6,000
Based upon a conversion of 60Hz = 300RPM	

Some earlier model machines that had an LED display and Spindle Speed Control dial mounted on the machine actually displayed Hz. In these cases use the above table to convert Hz to RPM and use the dial control to achieve the correct figure.

Maintenance

In order to ensure reliable, trouble free operation the following procedures must be followed.

Pre-heating

When starting the spindle for the first time each day, or after a long period of non-use allow the spindle to undergo a short warm up period. This will allow the bearings to slowly reach a uniform operating temperature. the following cycle speeds are recommended;

50% of maximum speed for 2 minutes
75% of maximum speed for 2 minutes
100% of maximum speed for 1 minute

Air Supply

Ensure that there is a constant and micro filtered air supply delivered to the spindle. Air is used for bearing cooling and also for tool ejection in the case of ATC spindles.

Cleaning

The tapered housing of the spindle shaft must be kept clean and free from dirt, dust, grease, coolant, oil, metal particles, rust or scale. The same is true for the ISO 30 collets and ISO 30 collet holders.

Failure to ensure clean surfaces will cause damage to the spindle motor and cause tools to run off centre.

Ensure that the air supply to the spindle is correctly micro filtered through the micro filter assembly on the Multicam machine. DO NOT allow the filter bowls to over fill with moisture or allow moist air to enter the spindles. Clean out the micro filter bowls at the start of each shift or if there is an excessive build up of moisture.

Regularly check the cooling fan at the top of the spindle to ensure correct operation. Do not allow the cooling fan housing to become obstructed or damaged.

Conditions

Should the operating conditions of the motor change (temperature rise, noise level, vibration, amperage draw); shut down the motor immediately to avoid further damage and contact the service personnel at Multicam Systems.

Important Safety Note

If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to “wobble” the tool holder it is not correctly engaged.

Tooling

Tooling used HSD ATC and collet motors must comply to the following requirements;

- The yellow arrow indicates the direction of rotation toward the right of the spindle (i.e clockwise).
- Collets and Collet Holders and nuts must be compatible with the ER32 standard.
- Avoid the presence of inserts, notches or any other form which may affect the dynamic balance.
- The degree of dynamic balance of the tooling and components must have a value adequate to the speed of the spindle. Do not run large diameter tooling or fly type cutters at high speed.
- Use only sharp tooling and ensure that all tooling is installed correctly in the appropriate collet.
- Do not use damaged, distorted or otherwise incomplete or unbalanced tooling.
- Before inserting the tool into the collet ensure that the shank is clean and even.
- Before installing the ISO 30 collet holder ensure that the taper is free from dust, damage or surface rust. Ensure that the pull stud on the end of the taper is free of damage.
- Use only genuine HSD ISO 30 holders for the HSD ATC motors.
- Do not leave ISO 30 holders in the spindle for long periods of time if the spindle is not in use. This can cause the holder to “sweat” into the spindle shaft making removal difficult.
- Never use tools beyond their rated speed.
- Do not use tools exceeding the following specifications as supplied by HSD.

RPM Range	Max Tool Dia (mm)	Max Weight (kg)
9,000 -12,000	130	4
12,000 - 14,000	100	2.5
14,000 - 18,000	80	2.5
18,000 - 24,000	50	2

Vibrations

Vibrations dramatically reduce bearing life and will cause premature failure. As such, vibrations must be avoided. Causes of vibration can be:

- unbalanced tooling
- improperly clamped workpiece
- excessive feed rates
- worn or dull tools

Inverters

HSD high frequency motors must be driven by high frequency, 3 phase power. Optimum operation and lowest motor temperatures are achieved when using a dynamic frequency inverter delivering a true sinusoidal wave form. If a solid state inverter is utilized, a three phase line reactor must be used to limit voltage spikes and current ripple. This reduces the risk of winding failure, limits operating temperature and reduces motor vibration.

The inverter should never been set to greater than 300Hz

Warranty

HSD S.r.l. warrants that the spindle has been factory tested with positive results. HSD S.r.l. will be responsible for defects in electrical and mechanical components for a period of twelve months from the date of purchase.

Bearings, gaskets, seals or other components that are subject to normal wear and tear are warranted for a period of six months from the date of purchase, subject to the conditions of use. The use of non-genuine tool holders, cover nuts or collets shall void the manufacturers warranty.

Information about Tooling and Cutters

The following section describes the correct use of tools and cutters and will also assist in obtaining maximum life from your cutters.

Tooling Maintenance - Tool Life

- Tools should be changed at the first sign of edge deterioration causing finish degradation or increase in operator effort to maintain feed rates.
- Never allow the tools to dwell in a cut.
- The router bit should be fed in such a manner so that in moving through the work it has a chance to bite or cut its way freely. If the cutter is fed too fast, strain and deflection will occur; if fed too slowly, friction and burning will occur. Both decrease the life of the router bit and are common causes of breakage.
- The router mechanism must be well maintained for any cutting tool to perform properly. Routinely check the collet for wear. Inspect tools being used for collet marks indicating slipping due to wear or dust build up. Collet, run-out and vibration problems cause premature tool failure and associated production difficulties.
- Avoid using adaptor bushings to reduce the size of the collet on a routing or production basis. Tools will not perform properly in bushings over an extended period of time. Bushings are for prototype, experimentation, test and evaluation and not for production.
- When cutting aluminium, brass and other ferrous and non-ferrous metals use a coolant when routing. Heat caused by action between the tool and piece part is enemy #1 to tool life.
- Heat is a function of material density per unit of time, thus, the more dense the material, the faster the feed rate required to minimize heat. A compromise must be reached, however, between finish and heat.
- Tool life is affected dramatically by tool geometry. Rake and clearance angles, as well as cutting edge length, should be examined. Always use the correct tooling geometry for the job at hand.
- Cutter breakage is most often caused by a misapplication of the tool. Do not assume the proper tools are being used. There are many local and regional preferences, which are not good routing practices.

Tooling Maintenance - Tool Breakage

In spite of the structural and metallurgical attributes, which are designed into industrial and professional router bits, breakage occurs. A detailed examination yields the following:

Application related breakage:

- Cutting edge lengths should be as short as possible to accommodate length of cut required.
- If longer cut lengths are required choose tools with the larges possible diameter.
- Larger cutting edge diameters require larger shank diameters. Spiral geometry can direct chip flow and expel chips to reduce heat.
- When tool application is a problem, changing the type of tool is the only solution.
- Excessive heat generated by friction shortens edge life, tends to create burn marks and may cause material being cut to accumulate behind each cutting edge and culminates in tool breakage.
- Shank out of round conditions prohibits effective colleting. This will result in out of balance vibration and will result in poor edge finish and cutter breakage. This will also have a dramatic effect on the life of spindle bearings.

Router Collet integrity:

A cutter is rendered nearly useless if the mechanics of gripping and rotating the tool are not treated with equal importance as the tool itself.

- Operators have snapped five or six tools in succession before inspecting colleting conditions. Overt signals such as breakage and dark markings on the shank of the bit warrant immediate investigation.
- Inspect the collet for out of round or bell mouthed conditions.
- Operators often overlook inspecting new collets based on the assumption that a new collet is correct.
- Always use the correct size collet for the tool. ER series collets are provided to cover a range of tool shanks. An ER collet with a range of 6-7mm is suitable for use with tools with shanks ranging from 6mm to 7mm, however if you are using a tool with a 6mm shank it is preferable to use a collet with a

holding range of 5-6mm because the collet does not need to close down so much to grip the shank of the tool.

- Collet performance will also be affected by dirt dust, bonding agents and scrap, which, occupy space and accelerate wear.

Collet Maintenance:

Collet maintenance is one of the most common causes of inadequate tool life or breakage. There are up to six links in the chain that make up this critical tool holding system called a collet. As a chain is only as strong as the weakest link, a router bit can only be as good as the system that holds it properly. The small amount of time spent to regularly inspect and clean the collet system will be more than offset in productivity increases and reduced overall costs.

1. Internal Collet Clamping Surfaces
The most important link in the tool holder chain is the inside of the collet. Resin migrates up through the slits in the collet and then deposits itself on the inside of the collet. This resin build up, if not removed, causes the collet to grip inconsistently on the tool shank. By not applying equal pressure throughout the entire gripping range of the collet, the tool holder allows the tool to resonate inside, causing slippage inside the collet.

Slippage can cause “fretting”, a condition in which resins are deposited on the shank of the tool. This resin buildup can be easily removed from the inside of the collet with Rust Free and brass tube-type brush. These brass brushes are non destructive and in conjunction with Rust Free can adequately remove the deposits high-pressure air guns cannot. Rust Free should be sprayed on and quickly brushed and wiped completely dry. Do not let the liquid sit and air dry.

2. Internal Spindle & Collet Taper
The inside taper of the spindle and tool holder is a critical surface which accumulates resin build up and should be cleaned at each tool change to maintain best concentricity. Felt brushes are available to fit most taper sizes and provide a quick means of removing short-term buildup.

3. External Collet & Tool Holder Taper
The outside taper of the spindle and tool holder require regular inspection and should be cleaned of all deposits each time the tool is changed. Brass brushes work well for this application, but felt cloths can also be used if the tapers are regularly maintained and the buildup is minor.

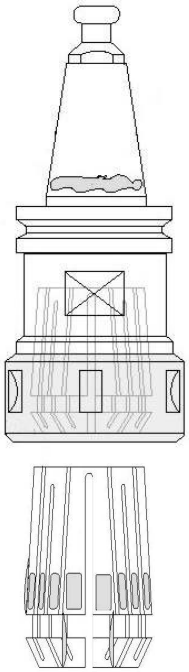
4. Clamping Nut Surfaces
The inside taper of the nut should be clean and free of burrs on the surface. Any surface burrs or contamination will not only skew a collet but can also permanently ruin a new collet. The clamping nut should be cleaned with a brass brush and/ or high-pressure air during every tool change. Special care should be taken to examine the clamping nut threads on a regular basis to ensure they are not damaged.

5. Thrust Bearings
Some collet nuts have an integrated thrust bearing connected to the inside taper. This bearing serves to reduce friction wear between the collet and nut as the nut is tightened. The bearing’s seating surface is the most critical feature and must be kept clean. The bearings should also be kept in smooth operating condition. If there is rough movement, it is a good indication of contamination or abuse. Either instance is indicative of runout and poor operation.

6. Tool Holders
Tool holders such as the ISO 30 have additional matching and mating tolerances beyond those of the older tapers. Because of their unique design, these tool-holding systems can be more prone to runout caused by resin buildup. “Fretting” or “Bronzing” will cause inconsistent gripping in the taper and/ or the flat mating surface and reduce consistency of tool life. If ignored, these conditions can eventually premature spindle failure. The mating surfaces should be cleaned with Rust Free and hand dried immediately afterwards.

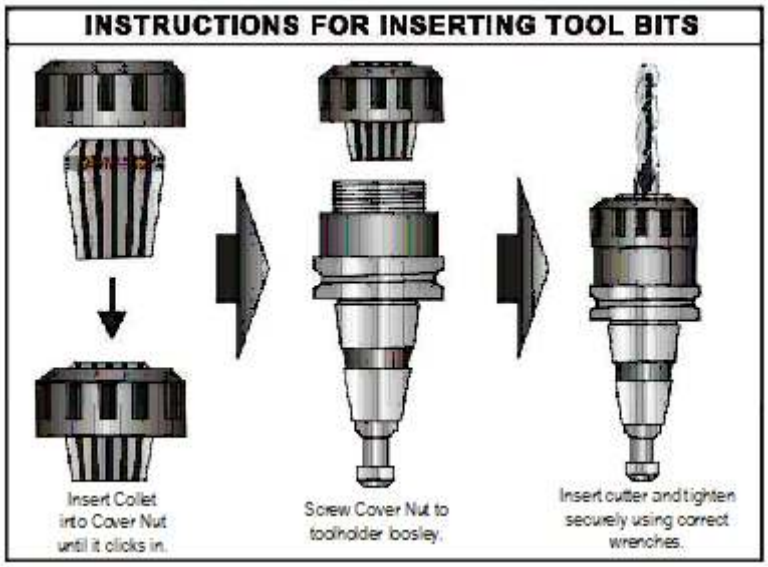
If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to “wobble” the tool holder it is not correctly engaged.

Correct Assembly of Tool Holders and Collets



ISO Tool Holder and ER Collet

Typical Collet Motor Tool Assembly



Note: To ensure trouble free operation, always insert the collet into the collet nut until it clicks in, and tighten loosely onto spindle prior to inserting tool bit. If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to "wobble" the tool holder it is not correctly engaged.

All six of these components are critical and should be regularly maintained. A high performance tool can only perform if the collet is properly maintained

Another item not to be overlooked is the fact that collets should be replaced on a regular basis. This means inspection on each tool change to look for metallic damage such as bell mouthing or burrs inside. If metallic damage is visible, the collet should be discarded and replaced. Even if there is no damage present the collet can be worn out through metal fatigue. Heat is directly transferred from the tool to the collet. These heating/cooling cycles remove the original tempering of the steel. Collets are made from spring steel allowing them to have a certain amount of elasticity to grip the tool. As the heat cycle is repeated this elasticity diminishes. Over time, a collet requires increased tightening to maintain the tool in proper position. As over tightening increases, the collet is distorted, creating eccentricities in the tool holder. Therefore, instead of over tightening older collets and creating a number of other problems, they should be replaced. Often the cost of a new collet can be offset by the cost of needlessly broken tools in one shift alone. Proper positioning of the tool in the collet is critical. The tool should only be gripped on the shank portion of the tool. At no time should any portion of the flute fade out be inside the collet.

Full Grip Collets
Full Grip Collets are identified by their slits that run from both ends, almost cutting the collet in pieces. This type of collet tends to have more flexibility and often comes in what is termed as “Range Collets”, which allow gripping a range of shank sizes. Example: 12-13mm is used for ½ shank tools. This full grip type allows gripping over the entire length of the collet and requires that to be properly used, be filled 75-80% full.

The most important portion of the collet is the mouth. This area is important because all the lateral pressure taken by the tool must be evenly distributed on all ears of the collet for it to cut true or concentric, It is very critical that the 80% rule be followed when using a full grip collet due to the ability of the collet to flare at the back if not full, the collet can actually allow tool movement in very minute amounts often times resulting in tool breakage. Equally as important as filling the collet properly, it should also be understood that it is possible to over-collet as well. This is when the “Flute fadeout” portion of the tool is allowed to extend up inside the collet. This does not allow a firm equal grip by all ears of the collet at the mouth. This allows the tool to have uneven support at the most critical area and the tool material is hard enough to actually scar the inside of the collet, causing permanent damage to the collet. This can also be common cause for tool breakage when it occurs. Breakage often results in permanent damage to the collet due to intense pressure exerted often either “Burring” or “mushrooming the mouth of the collet.

Heat is the biggest enemy of the tool, and the first place the heat goes from the tool is into the collet. It is also important to note that collets are made of spring steel that can, and will over a period of time lose it’s elasticity and harden, making it increasingly tougher to tighten adequately. As this hardening takes place, the steel does not fatigue evenly and often causes the collet to grip tighter on one side than the other, creating runout in the tool. It is important to understand that if they are overrun enough, this over tightening will eventually damage the internal spindle taper resulting in costly repairs. Because it takes place over a period of time, it is very hard to notice but, a safe recommendation for collet life is in the 600 -700 run time hours. This is about 3 months in a two shift operation of normal run times averages, probably much more frequent than one might expect. Just like changing the oil in your car, it is good preventative maintenance that should be done regularly.

Warranty

Machine Model _____ Serial No. _____ Date Of Purchase _____

Warranty Period _____ months from date of purchase, non-transferrable.

Warranty Terms
Multicam Systems Pty Limited guarantees against defects in material and quality of manufacture for the period shown herein, subject to proper application, use and maintenance of the equipment.

A Line Conditioner must be fitted by a Licensed Electrician prior to or at the time of installation at the customers cost. The Line Conditioner is designed to provide constant output voltage when there are fluctuations in the supply voltage. Regular servicing of the System must be carried out at 3 monthly intervals or after 200 working hours whichever occurs first. Four weeks grace may be granted outside of the 3 months by Multicam Systems. You will be contacted prior by the relative State Service Manager. Failure to comply with these conditions of warranty will deem all Warranties Null and Void, without exception. Attempts at servicing any or all of the System, by someone other than Multicam Service technician, will void all warranties immediately.

All attachments and components, including (but not limited to), onboard electronics, spindles, machine aggregates, vacuum pumps, dust collectors, material handling systems and knife cutting attachments supplied by third party manufacturers carry the 12months warranty of their original manufacturer only, with the exception of spindle bearings. Third party items are excluded from any and all provisions of any implied or offered 3year extended warranty. The use of non-recommended fluids in mist sprayer devices will void warranty on these devices. Any warranty expressly stated or impliedly offered by Multicam Systems on these items is not in addition to the warranty offered by the original manufacturer. Spindle motor bearings are warranted for a period of six months from the date shown herein, subject to conditions of use and maintenance as outlined further in this manual. Use of non-genuine tool holders or collets will void the manufacturer's warranty..

Warranty on vacuum pumps is subject proper use and maintenance as detailed in this manual. Inlet temperature is not to exceed 50°C. Pumps are not to be enclosed and must have a minimum of 1m free space on all sides for ventilation.

Warranty Provisions
Multicam Systems Pty Limited will replace or repair, at our own discretion, any part found to be faulty in manufacture or workmanship for the period as stated on this document. All items submitted for warranty repair must be shipped to the offices of Multicam Systems Pty Ltd. All freight costs are to be borne by the customer and do not constitute part of the warranty provisions.

The warranty provided by Multicam Systems Pty Limited does not extend to:-

1. any replacement or repair work carried out by any person other than an employee of Multicam Systems Pty Limited; or
2. any replacement or repair work required to be carried out as a result of damage or fault which arises as a consequence of the customers' inappropriate or incorrect operation of the Machine; or
3. any replacement or repair work required to be carried out as a consequence of the failure on the part of the customer to make available adequate or appropriate services and facilities advised by Multicam Systems Pty Limited as being necessary for the proper operation of the Machine, including but not being limited to adequate or appropriate power supply, or
4. any replacement or repair work required to be carried out as a consequence of use of non-genuine components, including but not being limited to tool holders and collets for spindles, or
5. any labour costs for any replacement, maintenance or repair work of an electrical nature which relates to any electrical component on the Machine or the power supply to the Machine, machine accessories or to the building in which the Machine is located. Engagement of qualified electrical trades persons shall be at the expense of the customer and in accordance with state legislation.
6. Multicam Systems Pty Ltd is not liable for any claim, production loss, damage or expense, without limitations whether direct or indirectly arising out of machine down time caused by warranty claims.

Warranty is non-transferrable. Multicam Systems Pty Limited advises that any replacement or repair work of an electrical nature should always be carried out by persons holding appropriate qualifications to carry out such work including any qualifications required under any State or Federal law.

Signed on behalf of Multicam Systems Pty Ltd;

_____	_____	_____
AUTHORISED SIGNATURE	NAME	DATE

Signed on behalf of (company name): _____

_____	_____	_____
AUTHORISED SIGNATURE	NAME	DATE

Rev:091118

Warranty

Machine Model _____ Serial No. _____ Date Of Purchase _____

Warranty Period _____ months from date of purchase, non-transferrable.

Warranty Terms
Multicam Systems Pty Limited guarantees against defects in material and quality of manufacture for the period shown herein, subject to proper application, use and maintenance of the equipment.

A Line Conditioner must be fitted by a Licensed Electrician prior to or at the time of installation at the customers cost. The Line Conditioner is designed to provide constant output voltage when there are fluctuations in the supply voltage. Regular servicing of the System must be carried out at 3 monthly intervals or after 200 working hours whichever occurs first. Four weeks grace may be granted outside of the 3 months by Multicam Systems. You will be contacted prior by the relative State Service Manager. Failure to comply with these conditions of warranty will deem all Warranties Null and Void, without exception. Attempts at servicing any or all of the System, by someone other than Multicam Service technician, will void all warranties immediately.

All attachments and components, including (but not limited to), onboard electronics, spindles, machine aggregates, vacuum pumps, dust collectors, material handling systems and knife cutting attachments supplied by third party manufacturers carry the 12months warranty of their original manufacturer only, with the exception of spindle bearings. Third party items are excluded from any and all provisions of any implied or offered 3year extended warranty. The use of non-recommended fluids in mist sprayer devices will void warranty on these devices. Any warranty expressly stated or impliedly offered by Multicam Systems on these items is not in addition to the warranty offered by the original manufacturer. Spindle motor bearings are warranted for a period of six months from the date shown herein, subject to conditions of use and maintenance as outlined further in this manual. Use of non-genuine tool holders or collets will void the manufacturer's warranty..

Warranty on vacuum pumps is subject proper use and maintenance as detailed in this manual. Inlet temperature is not to exceed 50°C. Pumps are not to be enclosed and must have a minimum of 1m free space on all sides for ventilation.

Warranty Provisions
Multicam Systems Pty Limited will replace or repair, at our own discretion, any part found to be faulty in manufacture or workmanship for the period as stated on this document. All items submitted for warranty repair must be shipped to the offices of Multicam Systems Pty Ltd. All freight costs are to be borne by the customer and do not constitute part of the warranty provisions.

The warranty provided by Multicam Systems Pty Limited does not extend to:-

1. any replacement or repair work carried out by any person other than an employee of Multicam Systems Pty Limited; or
2. any replacement or repair work required to be carried out as a result of damage or fault which arises as a consequence of the customers' inappropriate or incorrect operation of the Machine; or
3. any replacement or repair work required to be carried out as a consequence of the failure on the part of the customer to make available adequate or appropriate services and facilities advised by Multicam Systems Pty Limited as being necessary for the proper operation of the Machine, including but not being limited to adequate or appropriate power supply, or
4. any replacement or repair work required to be carried out as a consequence of use of non-genuine components, including but not being limited to tool holders and collets for spindles, or
5. any labour costs for any replacement, maintenance or repair work of an electrical nature which relates to any electrical component on the Machine or the power supply to the Machine, machine accessories or to the building in which the Machine is located. Engagement of qualified electrical trades persons shall be at the expense of the customer and in accordance with state legislation.
6. Multicam Systems Pty Ltd is not liable for any claim, production loss, damage or expense, without limitations whether direct or indirectly arising out of machine down time caused by warranty claims.

Warranty is non-transferrable. Multicam Systems Pty Limited advises that any replacement or repair work of an electrical nature should always be carried out by persons holding appropriate qualifications to carry out such work including any qualifications required under any State or Federal law.

Signed on behalf of Multicam Systems Pty Ltd;

_____	_____	_____
AUTHORISED SIGNATURE	NAME	DATE

Signed on behalf of (company name): _____

_____	_____	_____
AUTHORISED SIGNATURE	NAME	DATE

Rev:091118

Machine Configuration Blank

Configuration Parameter		Machine 1	Machine 2	Machine 3
Machine Units		MM	MM	MM
Toolpath Version No.				
Image File Version No.				
Drive Type				
Plot Resolution	Minimum Line Length			
	Arc Error			
Machine Calibration	X Axis			
	Y Axis			
	Z Axis			
Machine Size	X Axis			
	Y Axis			
	Z Axis			
Acceleration	Linear			
	Centripetal			
	Plunge			
	Homing			
Corner Pause				
Maximum Speeds	Feed Rate			
	Travel Rate			
	Plunge Rate			
Home Position	X Axis			
	Y Axis			
ATC Offset Position	X Axis			
	Y Axis			
	Z Axis			

MULTICAM S SERIES

MACHINE TRAINING

Date: __ / __ / ____

Customer Name:_____ ***Machine Serial Number:***_____

Contact Name:_____ ***Technician:***_____

The following persons have been trained in the basic procedures listed below;

_____ _____
NAME NAME

_____ _____
NAME NAME

✔ **check box to confirm**

Basic operation procedures in TOOLPATH software including:

- ☐ **Importing and saving files**
- ☐ **Toolpath Analysis**
- ☐ **Sequencing**
- ☐ **Start point allocation**
- ☐ **Output parameters for Tool allocation**
- ☐ **Tool bit compensation**
- ☐ **Multi Depth Control**
- ☐ **Lead in/Tabs (if applicable)**
- ☐ **Output Templates (if applicable)**
- ☐ **Group allocation**
- ☐ **2D/3D Engrave (if applicable)**
- ☐ **Drill points (if applicable)**
- ☐ **Other** _____

Basic operation procedures on the Multicam machine including:

- ☐ **Machine On/Off procedure**
- ☐ **Using Sub Console**
- ☐ **FUNC 1 Speed**
- ☐ **FUNC 2 Spindle control**
- ☐ **FUNC 4 Depth Settings**
- ☐ **FUNC 5 Device control**
- ☐ **FUNC 9 Aborting a job**
- ☐ **FUNC 10 Cut Direction**
- ☐ **FUNC 12 Machine Homing**
- ☐ **FUNC 15 Plunge Speed**
- ☐ **Automatic Tool Change Functions**
- ☐ **Vacuum Table operation on material without bows**
- ☐ **Use of Clamps for bowed material**
- ☐ **Other** _____

Basic maintenance procedures on the Multicam machine including:

- ☐ **Draining Micro Filter**
- ☐ **Cleaning Air filter (if a vacuum table)**
- ☐ **Dust Extraction bag emptying**
- ☐ **General cleaning procedure**

Technician Signature_____ **Customer Signature**_____

MULTICAM S SERIES

MACHINE TRAINING

Date: __ / __ / ____

Customer Name:_____ ***Machine Serial Number:***_____

Contact Name:_____ ***Technician:***_____

The following persons have been trained in the basic procedures listed below;

_____ _____
NAME NAME

_____ _____
NAME NAME

✔ **check box to confirm**

Basic operation procedures in TOOLPATH software including:

- ☐ **Importing and saving files**
- ☐ **Toolpath Analysis**
- ☐ **Sequencing**
- ☐ **Start point allocation**
- ☐ **Output parameters for Tool allocation**
- ☐ **Tool bit compensation**
- ☐ **Multi Depth Control**
- ☐ **Lead in/Tabs (if applicable)**
- ☐ **Output Templates (if applicable)**
- ☐ **Group allocation**
- ☐ **2D/3D Engrave (if applicable)**
- ☐ **Drill points (if applicable)**
- ☐ **Other** _____

Basic operation procedures on the Multicam machine including:

- ☐ **Machine On/Off procedure**
- ☐ **Using Sub Console**
- ☐ **FUNC 1 Speed**
- ☐ **FUNC 2 Spindle control**
- ☐ **FUNC 4 Depth Settings**
- ☐ **FUNC 5 Device control**
- ☐ **FUNC 9 Aborting a job**
- ☐ **FUNC 10 Cut Direction**
- ☐ **FUNC 12 Machine Homing**
- ☐ **FUNC 15 Plunge Speed**
- ☐ **Automatic Tool Change Functions**
- ☐ **Vacuum Table operation on material without bows**
- ☐ **Use of Clamps for bowed material**
- ☐ **Other** _____

Basic maintenance procedures on the Multicam machine including:

- ☐ **Draining Micro Filter**
- ☐ **Cleaning Air filter (if a vacuum table)**
- ☐ **Dust Extraction bag emptying**
- ☐ **General cleaning procedure**

Technician Signature_____ **Customer Signature**_____

MULTICAM S SERIES

MACHINE COMMISSIONING CHECKLIST

Date: __ / __ / ____

Customer Name:_____ ***Machine Serial Number:***_____

Contact Name:_____ ***Technician:***_____

- ☐ **Check that all items have arrived undamaged**
- ☐ **Ensure availability of correct power for machine - 3ph 32a with neutral**
- ☐ **Ensure availability of correct power for additional appliances (if req'd)**
- ☐ **Machine is installed with adequate clearances all round.**
- ☐ **Is machine frame earthed? YES / NO**
- ☐ **Ensure availability of correct air supply for machine - 110-120psi**
- ☐ **Check & tighten all screws incl spindle mounts & electrical connections**
- ☐ **Level Machine**
- ☐ **Install appliances (spindle, sub console, vac pumps, dust collector etc)**
- ☐ **Square up spindle to machine table.**
- ☐ **Check table vacuum zoning is correct**
- ☐ **Install labels, warning stickers & vacuum zone stickers**
- ☐ **Have customer install tool mounting jig if supplied**
- ☐ **Install FlashRAM & check machine startup procedure**
- ☐ **Test X,Y & Z homing (Func 12)**
- ☐ **Test torque on X,Y & Z axes**
- ☐ **Test Machine movement over entire process area.**
- ☐ **Check inverter acceleration/deceleration settings. ACC_____ DEC_____**
- ☐ **Check motor direction for vac pump & dust collector correct (if supplied)**
- ☐ **Set vacuum table relief valve to -32 to -33kPa when all valves closed**
- ☐ **Vacuum reading with all zones closed _____kPa, with material _____kPa**
- ☐ **Setup Automatic Tool Change position.**
- ☐ **Setup Tool Tip Sensor position (FUNC 350 for ATC), (FUNC 18 for QTC)**
- ☐ **Fit "Tool Change Setup Card" to inside of tool changer door.**
- ☐ **Install latest Toolpath software & Image File into C:\TPWIN folder**
- ☐ **Install RS485 communications device and check compatibility**
- ☐ **Test Toolpath operation and communication is 100%**
- ☐ **Check machine loading.**
- ☐ **Square machine over maximum area**
- ☐ **Calibrate Machine X, Y and Z.**
- ☐ **Set up homing Position**
- ☐ **Check configuration settings in Toolpath**
- ☐ **Set Tool descriptions and allocations in Advanced Tool**
- ☐ **Download several test jobs**
- ☐ **Perform multiple tool tip offset (FUNC 352)**
- ☐ **Run test jobs (not in material)**
- ☐ **Install design software (if supplied)**
- ☐ **Check compatibility with design software to Toolpath**
- ☐ **Warranty sighted**
- ☐ **Fill out Customer Information Sheet**
- ☐ **Fill out Machine configuration in manual**
- ☐ **Machine is in full working order**

Notes:_____

_____

_____

Technician Signature_____ **Customer Signature**_____

MULTICAM S SERIES

MACHINE COMMISSIONING CHECKLIST

Date: __ / __ / ____

Customer Name:_____ ***Machine Serial Number:***_____

Contact Name:_____ ***Technician:***_____

- ☐ **Check that all items have arrived undamaged**
- ☐ **Ensure availability of correct power for machine - 3ph 32a with neutral**
- ☐ **Ensure availability of correct power for additional appliances (if req'd)**
- ☐ **Machine is installed with adequate clearances all round.**
- ☐ **Is machine frame earthed? YES / NO**
- ☐ **Ensure availability of correct air supply for machine - 110-120psi**
- ☐ **Check & tighten all screws incl spindle mounts & electrical connections**
- ☐ **Level Machine**
- ☐ **Install appliances (spindle, sub console, vac pumps, dust collector etc)**
- ☐ **Square up spindle to machine table.**
- ☐ **Check table vacuum zoning is correct**
- ☐ **Install labels, warning stickers & vacuum zone stickers**
- ☐ **Have customer install tool mounting jig if supplied**
- ☐ **Install FlashRAM & check machine startup procedure**
- ☐ **Test X,Y & Z homing (Func 12)**
- ☐ **Test torque on X,Y & Z axes**
- ☐ **Test Machine movement over entire process area.**
- ☐ **Check inverter acceleration/deceleration settings. ACC_____ DEC_____**
- ☐ **Check motor direction for vac pump & dust collector correct (if supplied)**
- ☐ **Set vacuum table relief valve to -32 to -33kPa when all valves closed**
- ☐ **Vacuum reading with all zones closed _____kPa, with material _____kPa**
- ☐ **Setup Automatic Tool Change position.**
- ☐ **Setup Tool Tip Sensor position (FUNC 350 for ATC), (FUNC 18 for QTC)**
- ☐ **Fit "Tool Change Setup Card" to inside of tool changer door.**
- ☐ **Install latest Toolpath software & Image File into C:\TPWIN folder**
- ☐ **Install RS485 communications device and check compatibility**
- ☐ **Test Toolpath operation and communication is 100%**
- ☐ **Check machine loading.**
- ☐ **Square machine over maximum area**
- ☐ **Calibrate Machine X, Y and Z.**
- ☐ **Set up homing Position**
- ☐ **Check configuration settings in Toolpath**
- ☐ **Set Tool descriptions and allocations in Advanced Tool**
- ☐ **Download several test jobs**
- ☐ **Perform multiple tool tip offset (FUNC 352)**
- ☐ **Run test jobs (not in material)**
- ☐ **Install design software (if supplied)**
- ☐ **Check compatibility with design software to Toolpath**
- ☐ **Warranty sighted**
- ☐ **Fill out Customer Information Sheet**
- ☐ **Fill out Machine configuration in manual**
- ☐ **Machine is in full working order**

Notes:_____

_____

Technician Signature_____ **Customer Signature**_____

MULTICAM SERIES

GUIDE TO SAFE OPERATION

To ensure the proper and safe operation of the machine it is important that the following points be observed.

- Ensure that there are no tools or materials left on or near the rack and bearing rails of the gantry and the axis of movement.
- Ensure all guarding is in place including dust extraction devices. The machine cutting heads including spindles and oscillating knife heads are supplied with safety guarding. Do not operate the machine with safety guards removed.
- Ensure that all clamping systems are free of obstructions and that no other personnel are likely to be injured or engaged prior to activating the clamps.
- Ensure that all tools have been removed from the cutting area.
- Ensure that the guards are in place and correctly positioned.
- Ensure that the material being cut and the sacrificial board is no wider or thicker at any point than the width and height under the gantry and does not impede the movement of the gantry.
- Ensure that only recommended cutting fluids are used in mist dispensing systems.
- Ensure that all cutting tools to be used are sharp and in correct working order.
- Ensure that the filter for the vacuum pump is clean and that the correct vacuum valves are On/Off and are clear of any obstructions.
- Ensure that there is at least 500mm of clear open space around the machine.
- Ensure that all electrical and air supply components are in correct working order and have not been damaged in any way.
- Ensure that the vacuum pump inlet/outlet ports and pressure release valve are not covered.
- Ensure that the area around the machine is clean and free from obstructions which could lead to slips, trips or falls.
- Ensure that tools are correctly fitted in to collets and that collets are free from wear or damage.
- Ensure that ISO tool holders are in good condition and are free from wear or damage.
- If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to "wobble" the tool holder it is not correctly engaged.

To prevent damage to the machine or injury to machine operator it is imperative that all safety directions be fully observed when operating the Multicam machine.

Declaration by Safety Office/Owner

I acknowledge that safety aspects of the machine and its operation have been explained to me and persons trained to operate the machine. I acknowledge that future operators will receive appropriate training prior to being permitted to operate the machine and that all safety procedures associated with the operation of machinery will be strictly adhered to in accordance with our obligations under Occupational Health and Safety legislation.

_____ _____

PRINT NAME SIGNED & DATE

MULTICAM SERIES

GUIDE TO SAFE OPERATION

To ensure the proper and safe operation of the machine it is important that the following points be observed.

- Ensure that there are no tools or materials left on or near the rack and bearing rails of the gantry and the axis of movement.
- Ensure all guarding is in place including dust extraction devices. The machine cutting heads including spindles and oscillating knife heads are supplied with safety guarding. Do not operate the machine with safety guards removed.
- Ensure that all clamping systems are free of obstructions and that no other personnel are likely to be injured or engaged prior to activating the clamps.
- Ensure that all tools have been removed from the cutting area.
- Ensure that the guards are in place and correctly positioned.
- Ensure that the material being cut and the sacrificial board is no wider or thicker at any point than the width and height under the gantry and does not impede the movement of the gantry.
- Ensure that only recommended cutting fluids are used in mist dispensing systems.
- Ensure that all cutting tools to be used are sharp and in correct working order.
- Ensure that the filter for the vacuum pump is clean and that the correct vacuum valves are On/Off and are clear of any obstructions.
- Ensure that there is at least 500mm of clear open space around the machine.
- Ensure that all electrical and air supply components are in correct working order and have not been damaged in any way.
- Ensure that the vacuum pump inlet/outlet ports and pressure release valve are not covered.
- Ensure that the area around the machine is clean and free from obstructions which could lead to slips, trips or falls.
- Ensure that tools are correctly fitted in to collets and that collets are free from wear or damage.
- Ensure that ISO tool holders are in good condition and are free from wear or damage.
- If manually changing tools in a quick tool change motor ensure that the tool holder is correctly engaged in the motor. Do this by firmly grasping the nut on the tool holder and pulling in a downward and sideways movement and checking for looseness. If you are able to "wobble" the tool holder it is not correctly engaged.

To prevent damage to the machine or injury to machine operator it is imperative that all safety directions be fully observed when operating the Multicam machine.

Declaration by Safety Office/Owner

I acknowledge that safety aspects of the machine and its operation have been explained to me and persons trained to operate the machine. I acknowledge that future operators will receive appropriate training prior to being permitted to operate the machine and that all safety procedures associated with the operation of machinery will be strictly adhered to in accordance with our obligations under Occupational Health and Safety legislation.

PRINT NAME

SIGNED & DATE

Machine Safety Hazard Identification, Assessment and Control

Note: Machine commissioning and training is not complete until this document has been signed and presented to the machine operator or chief safety officer. This document to be used in conjunction with your in-house Safe Work Method Statement (SWMS) and displayed at the machine in plain site of the machine operator.

Machine Description: Multicam CNC Flat Bed Routing Machine Model: _____ **Serial No:** _____

Item/Device	Hazard Identification	Hazard Assessment	Risk Control Strategies (recommended for Purchaser/Buyer/User)
GANTRY	STRIKING/CRUSHING	LOW	Maintain safe working distance from machine when in operation. Do not lean on machine when in operation. Do not place loose objects in the path of the gantry. Do not lean objects against machine. Remove all loose objects around moving parts. Keep well clear of moving parts.
CLAMPING SYSTEMS	STRIKING/CRUSHING	MEDIUM	Ensure that all clamping systems are free of obstructions and that no other personnel are likely to be injured or engaged prior to activating the clamps.
VACUUM BED	STRIKING	LOW	In the event of vacuum pump failure during cutting, cut parts may move or be ejected from the machine bed. Maintain a safe distance from the machine when in operation. Wear suitable PPE. Do not operate machine if machine table vacuum is low or vacuum pump/s are faulty.
SPINDLE KNIFE HEADS	CUTTING ENTANGLEMENT STRIKING	MEDIUM	Isolate main power before changing/inspecting cutters or knife blades. Ensure tool is not rotating or moving before changing/inspecting cutters or knife blades. Hands must always be kept well clear from spindle and cutters when machine is in operation. Long or loose hair must be covered. Loose clothing/jewellery must not be worn when operating the machine or changing/inspecting cutters. All materials must be suitably clamped/held/secured before cutting. Use of suitable PPE (cut resistant gloves) is recommended when handling router cutters or knife blades.
AIR ASSIST DEVICE	CUTTING/CRUSHING SHEARING	MEDIUM	Air assist is used to control the weight of the spindle. A sudden release in air pressure or disruption of controlling circuitry will cause the spindle to drop rapidly. Even if power is off or air supply is not present. Keep hands and fingers clear or all moving parts and crush points.
CONTROLLER AND POWER BOX	ELECTRICAL	MEDIUM	All electrical enclosures should only be opened with the correct tool that is not to be kept with the machine. Electrical work should only be carried out a licensed electrician.
AUTOMATIC TOOL CHANGER	ENTANGLEMENT STRIKING ELECTRICAL	LOW	Ensure all covers are secured before starting machine. Do not operate machine with tool change cabinet open. All electrical enclosures should only be opened with the correct tool that is not to be kept with the machine. Electrical work should only be carried out a licensed electrician.
MIST SPRAYER	CONTAMINATION	LOW	Use only approved products in the Mist Sprayer Unit. Do not use flammable products such as methylated spirit in the Mist Sprayer Unit. Area must be well ventilated to prevent build up of mist vapour.
DUST COLLECTOR	CONTAMINATION/NOISE FATIGUE	MEDIUM	A dust extraction system must be used if cutting porous materials or materials that create dust or fibres during the cutting process. Hearing and eye protection must be worn when operating the machine.
MACHINE BED	ERGONOMIC	MEDIUM	Heavy or large pieces must me carried/loaded by more than one person or loaded using mechanical or automated methods.
MATERIAL LOADER	STRIKING/CRUSHING/ SHEARING	LOW	Do not stand behind the gantry when executing automated material loading. Do not place hands or fingers between the machine bed/frame and the on-loading table
MATERIAL OFFLOADING DEVICE	STRIKING/CRUSHING	LOW	Do not stand in front of the gantry when executing automated material offloading. Do not place hands or fingers between the machine unloading arm and the product being removed from the machine.
OFFLOADING TABLE OR CONVEYOR	STRIKING/CRUSHING	LOW	The machine unloading arm moves the finished product on to the offloading table or conveyor. Ensure that the offloading table or conveyor bed height is below the machine bed height. Ensure that the offloading table or conveyor bed is placed within 10mm of the machine bed. Do not place hands or fingers between the product being removed from the machine and the offloading table or conveyor
OTHER HAZARDS	NOISE, FATIGUE	LOW	Appropriate PPE must be worn when operating the machine. In enclosed areas or small premises noise from the dust extraction system, vacuum pumps and spindle can elevate ambient noise levels. This must be taken in to consideration when developing your SWMS.

References: Australian Worksafe Standard for Plant (NOHSC:1010-1994) and AS4024 Revised January, 2017

Authorised and signed by Operator/Safety Officer: _____ Installer:_____

Machine Safety Hazard Identification, Assessment and Control

Note: Machine commissioning and training is not complete until this document has been signed and presented to the machine operator or chief safety officer. This document to be used in conjunction with your in-house Safe Work Method Statement (SWMS) and displayed at the machine in plain site of the machine operator.

Machine Description: Multicam CNC Flat Bed Routing Machine Model: _____ **Serial No:** _____

Item/Device	Hazard Identification	Hazard Assessment	Risk Control Strategies (recommended for Purchaser/Buyer/User)
GANTRY	STRIKING/CRUSHING	LOW	Maintain safe working distance from machine when in operation. Do not lean on machine when in operation. Do not place loose objects in the path of the gantry. Do not lean objects against machine. Remove all loose objects around moving parts. Keep well clear of moving parts.
CLAMPING SYSTEMS	STRIKING/CRUSHING	MEDIUM	Ensure that all clamping systems are free of obstructions and that no other personnel are likely to be injured or engaged prior to activating the clamps.
VACUUM BED	STRIKING	LOW	In the event of vacuum pump failure during cutting, cut parts may move or be ejected from the machine bed. Maintain a safe distance from the machine when in operation. Wear suitable PPE. Do not operate machine if machine table vacuum is low or vacuum pump/s are faulty.
SPINDLE KNIFE HEADS	CUTTING ENTANGLEMENT STRIKING	MEDIUM	Isolate main power before changing/inspecting cutters or knife blades. Ensure tool is not rotating or moving before changing/inspecting cutters or knife blades. Hands must always be kept well clear from spindle and cutters when machine is in operation. Long or loose hair must be covered. Loose clothing/jewellery must not be worn when operating the machine or changing/inspecting cutters. All materials must be suitably clamped/held/secured before cutting. Use of suitable PPE (cut resistant gloves) is recommended when handling router cutters or knife blades.
AIR ASSIST DEVICE	CUTTING/CRUSHING SHEARING	MEDIUM	Air assist is used to control the weight of the spindle. A sudden release in air pressure or disruption of controlling circuitry will cause the spindle to drop rapidly. Even if power is off or air supply is not present. Keep hands and fingers clear or all moving parts and crush points.
CONTROLLER AND POWER BOX	ELECTRICAL	MEDIUM	All electrical enclosures should only be opened with the correct tool that is not to be kept with the machine. Electrical work should only be carried out a licensed electrician.
AUTOMATIC TOOL CHANGER	ENTANGLEMENT STRIKING ELECTRICAL	LOW	Ensure all covers are secured before starting machine. Do not operate machine with tool change cabinet open. All electrical enclosures should only be opened with the correct tool that is not to be kept with the machine. Electrical work should only be carried out a licensed electrician.
MIST SPRAYER	CONTAMINATION	LOW	Use only approved products in the Mist Sprayer Unit. Do not use flammable products such as methylated spirit in the Mist Sprayer Unit. Area must be well ventilated to prevent build up of mist vapour.
DUST COLLECTOR	CONTAMINATION/NOISE FATIGUE	MEDIUM	A dust extraction system must be used if cutting porous materials or materials that create dust or fibres during the cutting process. Hearing and eye protection must be worn when operating the machine.
MACHINE BED	ERGONOMIC	MEDIUM	Heavy or large pieces must me carried/loaded by more than one person or loaded using mechanical or automated methods.
MATERIAL LOADER	STRIKING/CRUSHING/ SHEARING	LOW	Do not stand behind the gantry when executing automated material loading. Do not place hands or fingers between the machine bed/frame and the on-loading table
MATERIAL OFFLOADING DEVICE	STRIKING/CRUSHING	LOW	Do not stand in front of the gantry when executing automated material offloading. Do not place hands or fingers between the machine unloading arm and the product being removed from the machine.
OFFLOADING TABLE OR CONVEYOR	STRIKING/CRUSHING	LOW	The machine unloading arm moves the finished product on to the offloading table or conveyor. Ensure that the offloading table or conveyor bed height is below the machine bed height. Ensure that the offloading table or conveyor bed is placed within 10mm of the machine bed. Do not place hands or fingers between the product being removed from the machine and the offloading table or conveyor
OTHER HAZARDS	NOISE, FATIGUE	LOW	Appropriate PPE must be worn when operating the machine. In enclosed areas or small premises noise from the dust extraction system, vacuum pumps and spindle can elevate ambient noise levels. This must be taken in to consideration when developing your SWMS.

References: Australian Worksafe Standard for Plant (NOHSC:1010-1994) and AS4024 Revised January, 2017

Authorised and signed by Operator/Safety Officer: _____ Installer:_____