Pa

Patran 2021

Interface to DYTRAN Preference Guide

HEXAGON | MSC Software

Corporate
MSC Software Corporation
4675 MacArthur Court, Suite 900
Newport Beach, CA 92660
Telephone: (714) 540-8900
Toll Free Number: 1 855 672 7638
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa
MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: europe@mscsoftware.com

Japan
MSC Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific
MSC Software (S) Pte. Ltd.
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@mscsoftware.com

Worldwide Web
www.mscsoftware.com

Support
http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx

P3:V2021:Z:INT-DT:Z:DC-USR-PDF

Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, please write to us at documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on.)
- Your suggestions for correcting/improving documentation

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at:

http://msc-documentation.questionpro.com.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

Contents

Patran Interface to Dytran Preference Guide

3 Running an Analysis

4 Read Results

5 Read Input File

6 Files

1 Overview

Purpose

Patran is an analysis software system developed and maintained by MSC Software Corporation. The core of the system is a finite element analysis pre and postprocessor. Several optional products are available including; advanced post processing programs, tightly coupled solvers, and interfaces to third party solvers. This document describes one of these interfaces.

The Patran Dytran interface provides a communication link between Patran and Dytran. It also provides for the customization of certain features in Patran.

Selecting Dytran as the analysis code preference in Patran activates the customization process. These customizations ensure that sufficient and appropriate data is generated for the Patran Dytran interface. Specifically, the Patran forms in *five* main areas are modified:

1. Materials
2. Element Properties
3. Finite Elements/MPCs and Meshing
4. Loads and Boundary Conditions
5. Analysis forms

The interface is a fully integrated part of the Patran system. The casual user will never need to be aware that separate programs are being used. For the expert user, there are *four* main components of the preference: a *PCL* function, `load_mscdytran()`, which will load all Dytran specific definitions, like element types and material models, into a Patran database, *mscdytran_template.db* which contains all Dytran and MSC Nastran specific definitions, *p3dytran* to convert model data from a database into the analysis code input file and vice-versa, and *dytranp3* to translate results and/or model data from the analysis code results file into a database.

The *PCL* function `load_mscdytran()` can be invoked by simply typing its name into the Patran command line. This will load Dytran specific definitions into the database currently opened. These specific definitions can be added to any database (which does not already contain Dytran specific definitions) at any time. Obviously, a database must be open for `load_mscdytran()` to operate correctly. See Dytran Preference Integration with Patran for complete information and a description of how to create a new template database.

p3dytran translates model data between the Patran database and the analysis code-specific input file format. This translation must have direct access to the originating database when an Dytran input file is being created. p3dytran also translates basic topology information from the code specific input files and imports that data into Patran.

dytranp3 translates results and/or model data from the analysis, code-specific results file into the Patran database. This program can be run so the data is loaded directly into the database, or if incompatible computer platforms are being used, an intermediate file can be created.

Reading Dytran Input Files

This release of the Patran Dytran interface provides support for reading Dytran input files. Nodes, elements, coordinate systems, materials, properties and lbcs are read from an input file.

Data read from an Dytran input file can also be read from LS-DYNA3D and PAMCRASH input files in the keyword format. In all cases the data should be imported into an empty database.

Dytran Product Information

Dytran is a general-purpose explicit finite element computer program for nonlinear dynamic analysis of structures in three dimensions. It is developed, supported, and maintained by MSC Software Corporation. See the Dytran User's Manual for a general description of capabilities as well as detailed input instructions.

What is Included with this Product?

The Dytran Preference product includes the following items:

1. A PCL function load_mscdytran() contained in `p3patran.plb` which adds Dytran specific definitions to any Patran database (not already containing such definitions) at any time.
2. A PCL library called `mscdytran.plb` that is included in the `<installation_directory>` directory. This library is used by the analysis forms to produce analysis code specific translation parameter, solution parameter, etc., forms.
3. Two executable programs called `dytranp3` and `p3dytran`, both contained in the `<installation_directory>/bin/exe` directory. These programs translate information from Dytran state and time history files into a Patran database and translate information from a database into a Dytran input file. These programs can be run independent of Patran but typically run underneath, and are transparent to the user.
4. A script file called `MscDytranExecute`, contained in the `<installation_directory>/bin/exe` directory. This script can be run independently of Patran but typically run underneath Patran, transparent to the user.
5. This Analysis Preference guide is included as part of the product. An on-line version is also provided to allow the direct access to this information from within Patran.
6. ATB hybridII and hybrid III dummy files are included in the `<installation_directory>/mscdytran_files` directory.

Dytran Preference Integration with Patran

Creation of an Dytran Template Database

Three versions of the Patran database are delivered in the `<installation_directory>` directory, "`base.db`", "`template.db`" and "`mscdytran_template.db`". The `template.db` database contains the analysis code specific definitions for all of the MSC supported interfaces. The `mscdytran_template.db` database contains analysis code specific definitions for the mscdytran and mscnastran interfaces. "`base.db`" is basically devoid of analysis code specific definitions but does contain some basic definitions. In order to create a template database which contains only Dytran specific definitions, the user should follow these steps:

1. Within Patran open a new database using `base.db` as the template.
2. Enter `load_mscdytran()` into the command line.
3. Save this database under a name like `mscdytran.db` to be your new "Dytran only" template database.

4. From then on, when opening a new database, choose `mscdytran.db` as your template database.

An existing database which has been derived from `base.db` may not contain the specific definitions needed to run the Dytran Preference. But, these definitions can be added at any time by simply typing `load_mscdytran()` into the Patran command line.

Due to the excessive size of “`template.db`” it is highly recommended that the user either select “`mscdytran_template.db`” or create his own unique template database which contains only the analysis code specific definitions pertaining to the analysis codes of immediate interest to him. This will produce considerably smaller and simpler databases than would the use of “`template.db`”. For more details about adding analysis code specific definitions to a database and/or creating unique template databases, refer to Modifying the Database Using PCL (Ch. 6) in the *PCL and Customization* or to the Patran *Installation and Operations Guide*.

Patran Dytran Preference Components

The diagrams shown below indicate how the functions, scripts, programs and files which constitute the Dytran Preference affect the Patran environment. Site customization, in some cases, is indicated.

Figure 1-1 shows the process of running an analysis. The `mscdytran.plb` library defines the Translation Parameter, Solution Type, Solution Parameter, and Output Request forms called by the Analysis form. When the *Apply button* is pushed on the Analyze form `p3dytran` is executed. `p3dytran` reads data from the database and creates the Dytran input file. If `p3dytran` finishes successfully, and the user requests it, the script will then start Dytran.

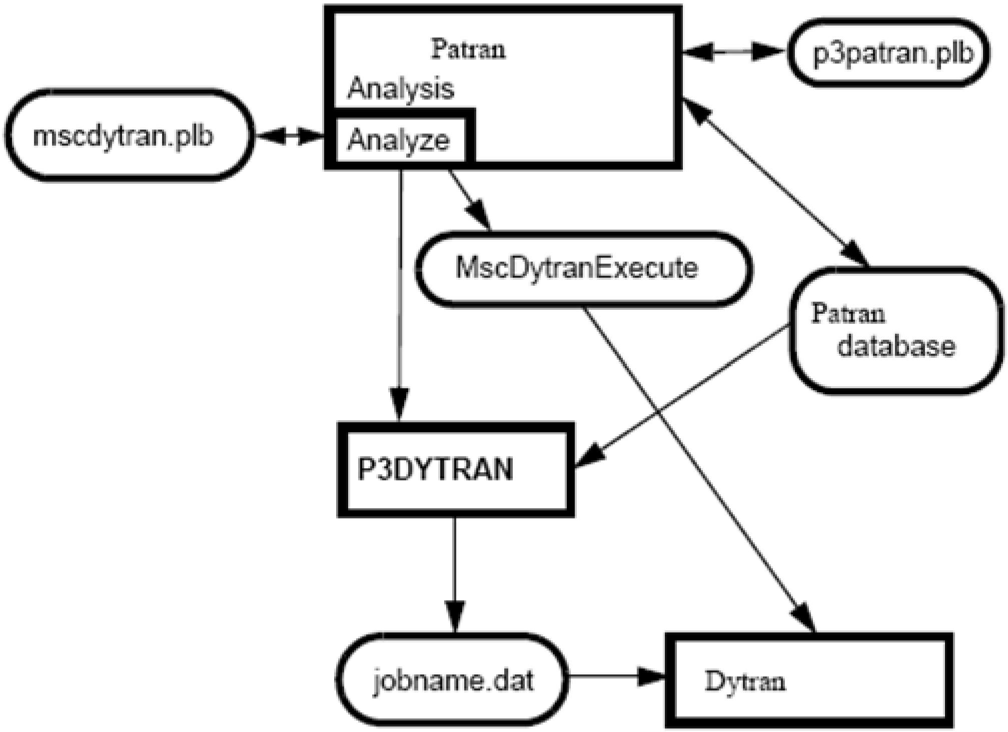


Figure 1-1 Forward Translation

Figure 1-2 shows the process of reading information from Dytran Archive or Time History files. When the *Apply button* is selected on the Read Results form the `dytranp3` results translation is started. The Patran database is closed while this translation occurs. `dytranp3` reads the data from the Dytran State and Time History Files. If `dytranp3` can find the desired database, the results will be loaded directly into it. However, if it cannot find the database (e.g., you are running on incompatible platforms), `dytranp3` will write all the data into a flat file. This flat file can be taken to wherever the database is, and read by using the read file selections.

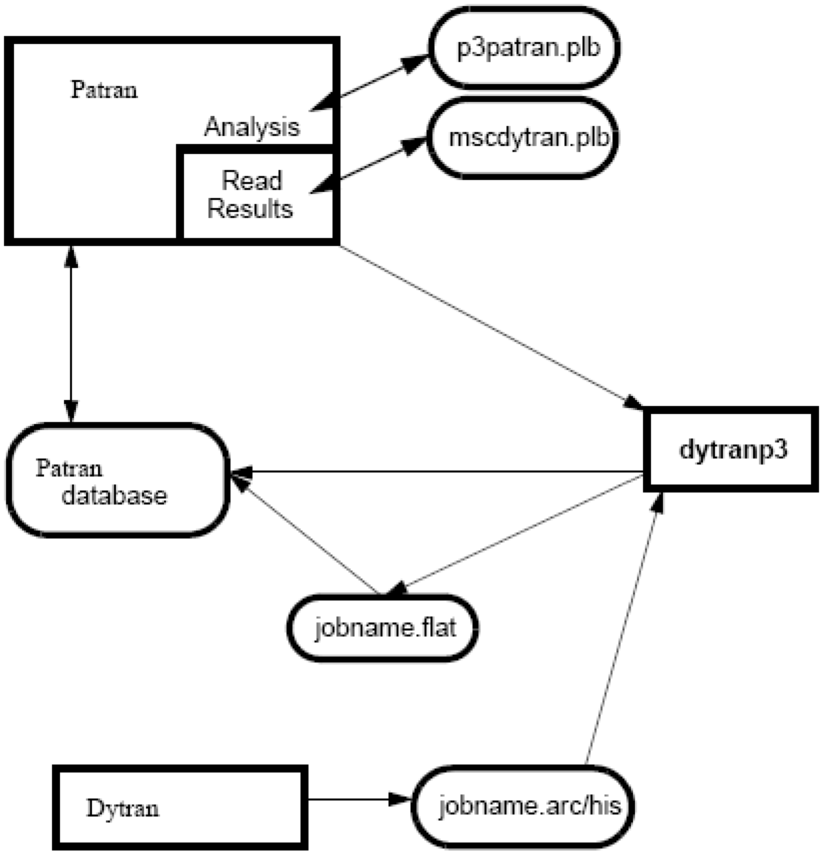


Figure 1-2 Results File Translation

Figure 1-3 shows the process of translating information from an Dytran input file into a Patran database. The behavior of the main Analysis/Read Input File form and the subordinate file select form is dictated by the `mscdytran.plb` PCL library. The apply button on the main form activates the `p3dytran` program which reads the specified Dytran input file into the Patran database.

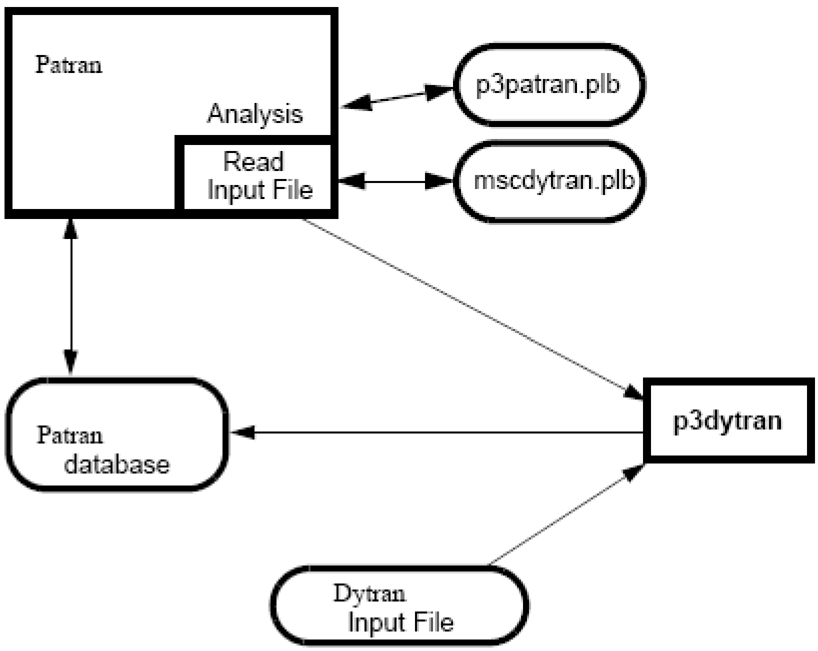


Figure 1-3 Dytran Input File Translation

Configuring the Patran Dytran Execute File

The `MscDytranExecute` script file controls the execution of the Dytran analysis code.

This script file contains information specific to the local installation of Dytran. In order for the “Full Run” option to correctly spawn Dytran, this data must be edited into the script file after installation of Patran and must be updated any time the Dytran installation changes. The site specific parameters are:

```
Host=”
Scratchdir=”
Acommand=”
```

The Host parameter identifies the machine on which Dytran is installed. If Dytran is locally installed, set this parameter to LOCAL. If Dytran is installed on a remote machine, set the parameter to the name of the remote machine.

If the Hosts parameter is set to LOCAL, the Scratchdir parameter is not meaningful and should be left blank (“). If Dytran is installed on a remote machine, the Scratchdir parameter should be set to a directory on the remote machine that can be used to store the analysis files during analysis. If the Dytran host machine is remote, the submit script will copy the Dytran input file to the Scratchdir directory, run Dytran in the Scratchdir directory, copy the output files to the user's working directory and then delete the files in the Scratchdir directory.

The Acommand parameter must be set to the executable command for Dytran.

If Dytran is locally installed on your machine, your submit script might be as follows:

```
Host='LOCAL'
Scratchdir='/tmp'
Acommand='/msc/bin/mscdytran'
```

If Dytran is installed on a remote host, your submit script would look more like the following:

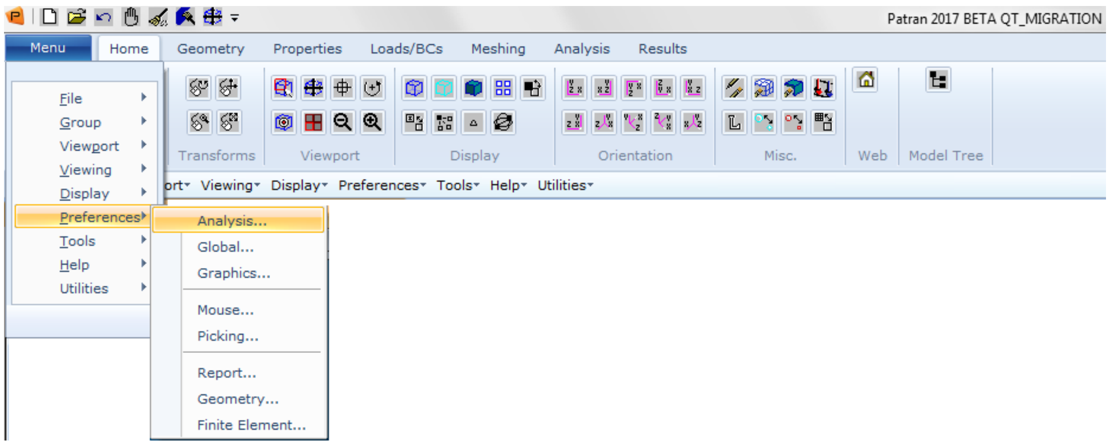
```
Host='lansing'
Scratchdir='/tmp'
Acommand='/msc/bin/mscdytran'
```

2 Building A Model

Introduction to Building a Model

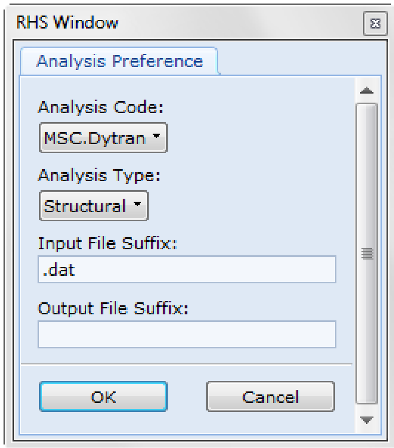
There are many aspects to building a finite element analysis model. In several cases, the forms used to create the finite element data are dependent on the selected analysis code and analysis type. Other parts of the model are created using standard forms.

The Analysis option on the Preferences menu brings up a form where the user can select the analysis code (e.g., Dytran).



The analysis code may be changed at any time during model creation. This is especially useful if the model is to be used for different analyses, in different analysis codes. As much data as possible will be converted if the analysis code is changed after the modeling process has begun. The analysis option defines what will be presented to the user in several areas during the subsequent modeling steps.

These areas include the material and element libraries, including multi-point constraints, the applicable loads and boundary conditions, and the analysis forms. The selected Analysis Type may also affect the allowable selections in these same areas. For more details, see The Analysis Form (p. 6) in the *Reference Manual - Part V*.



Parameter	Description
Analysis Code	To use the Patran Dytran Analysis Preference, this should be set to MSC.Dytran.
Analysis Type	The only Analysis Type for Dytran is Structural.
Input File Suffix **Output File Suffix**	Indicates the file suffixes used in creating file names for Dytran input and output files.

Table 2-1 summarize the various Dytran commands supported by the Patran Dytran Analysis Preference.

Table 2-1 Supported Dytran Commands

File Section	Subsection	Data Entry	Method
File Management Section	Prestress Analysis	BULKOUT, NASTDISP, PRESTRESS, SOLUOUT	Analysis/Initiating Calculation
	New Analysis	NASTINP, NASTOUT, SOLINIT, START	Analysis/Initiating Calculation
	Restart Control	RESTART, RSTFILE, RSTBEGIN	Analysis/Restart
	User Code	USERCODE	Analysis/Initiating Calculation
	File Selection	SAVE, TYPE	Analysis/Output Requests
		IMMFILE	Analysis/Initiating Calculation

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Executive Control	Execution Control	CEND, MEMORY-SIZE, TIME	Analysis/Execution Controls

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Case Control	Analysis Control	ENDSTEP, ENDTIME	Analysis/Execution Controls
		CHECK	Analysis/Initiating Calculation
	Data Selection	SPC, TIC, TLOAD,	Analysis/Select Load Cases
	Output Control	SET, SETC	Analysis/Output Requests
		TITLE	Analysis
	Output Selection - Entity Specification	ACC, COG, CONTS, PLSURFS, CSECS, EBDS, LEMENTS, BAGS, GRIDS, HIC, MATS, RELS, RIGIDS, UBSURFS, URFACES	Analysis/Output Requests
		PLANES, USASURFS	Not Supported
	Output Selection - Variable Specification	CONTOUT, CPLOUT, SOUT, BDOUT, LOUT, BAGOUT, POUT, ATOUT, BOUT, ELOUT, UBSOUT, SURFOUT	Analysis/Output Requests

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Case Control (continued)	Output Frequency	STEPS, TIMES	Analysis/Output Requests
	User Defined Output	ELEXOUT, GPEXOUT	Analysis/Output Requests
	Input File Control	INCLUDE	Analysis/Translation Parameter
	Miscellaneous	PARAM	Analysis
Bulk Data	Grid Points	GRID	Finite Elements/Nodes
		GRDSET	Analysis/Default Gridpoint
		GROFFS	Analysis/Gridpoint Offset
	Coordinate Systems	CORD2C, CORD2R, CORD2S	Geometry/Coordinates
		CORD1C, CORD1R, CORD1S	Not Supported
		CORD3R, CORD4R	Not Supported
		CORDROT	Not Supported
	Hourglass	HGSUPPR	Element Property
	Lagrangian, 0-D	CONM2	Element Property
		PDAMP	Element Property
		PELAS, PELAS1, PELASEX	Element Property
	Lagrangian, Solid Elements	CHEXA	Elements/3D
		CPENTA	Elements/3D
		CTETRA	Elements/3D
		PSOLID	Element Property

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Lagrangian, Surface Elements	CQUAD4	Elements/2D
		CTRIA3	Elements/2D
		PSHELL	Element Property
		PSHELL1	Element Property
		PCOMP, PCOMPA	Element Property
	Lagrangian, 1-D Elements	CBAR	Elements/1D
		CBEAM	Elements/1D
		CROD	Elements/1D
		CDAMP1	Elements/1D
		CDAMP2	Not Supported
		CELAS1	Elements/1D
		CELAS2	Not Supported
		CSPR	Elements/1D
		CVISC	Elements/1D
		PBAR	Element Property
		PBCOMP	Element Property
		PBEAM, PBEAM1, PBEAML	Element Property
		PBELT	Element Property
		PDAMP	Element Property
		PELAS, PELAS1, PELASEX	Element Property
		PROD	Element Property
		PSPR, PSPR1, PSPREX	Element Property
		PVISC, PVISC1, PVISCEX	Element Property
		PWELD, PWELD1, PWELD2	Element Property

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Eulerian, Solid Elements	CHEXA	Elements/3D
		CPENTA	Elements/3D
		CTETRA	Elements/3D
		PEULER, PEULER1	Element Property
	Mesh Generator	MESH	LBC/Mesh Generator
	Constitutive Models	DMAT, DMATEL, DMATEP, DMATOR	Material
		DYMAT14, DYMAT24, DYMAT25, DYMAT26	Material
		FOAM1, FOAM2	Material
		MAT1, MAT2, MAT8, MAT8A	Material
		FABRIC	Material
		RUBBER1	Material
		SHEETMAT	Material
	Yield Models	YLDEX, YLDJC, YLDMC, YLDMSS, YLDPOL, YLDRPL, YLDTM, YLDVM, YLDZA	Material
		YLDHY	Not Supported
	Shear Models	SHREL, SHREX, SHRLVE, SHRPOL	Material

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Equations of State	EOSEX, EOSGAM, EOSIG, EOSJWL, EOSPOL, EOSTAIT	Material
	Failure Models	FAILEST, FAILEX, FAILEX1, FAILMES, FAILMPS, FAILPRS, FAILSDT	Material
	Spallation Models	PMINC	Material
	Rigid Bodies	MATRIG	Material
		RBE2	Finite Elements/MPC's
		RBE2-FULLRIG	LBC
		RBHINGE	LBC/Rigid Body Hinge
		RELEX	Not Supported
		RELLIPS	LBC/Rigid Ellipsoid
		RIGID	LBC/Rigid Surface
		RPLEX	Not Supported
	ATB Interface	ATBACC, ATBJNT, ATBSEG	Not Supported
	Lagrangian, Single Point Constraints	SPC, SPC1, SPC2, SPC3	LBC

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Lagrangian, Contact Surfaces	CONTACT, CONTFORC	LBC/Contact
		SURFACE, SUBSURF, CFACE, SET1	LBC/Contact
		CONTINI	Not Supported
		CONTREL	LBC/Rigid Ellipsoid
		CSEG, CFACE1	Not Supported
	Lagrangian, Connections	JOIN	Not Supported
		BJOIN	LBC/Bjoin
		KJOIN	LBC/Kjoin
		RCONN	LBC/Rigid Connection
		RCONREL	Not Supported
		RJCYL, RJPLA, RJREV, RJSPH, RJUNI, RJTRA	LBC/Rigid Joint Constraint
	Lagrangian, Rigid Wall	WALL	LBC/Planar Rigid Wall
	Lagrangian, Rigid Body Constraint	RBC3	Not Supported
	Lagrangian, Transient Loading	TLOAD1	LBC (Field)
		DAREA	Not Supported
		FORCE, FORCE1, FORCE2	LBC
		MOMENT, MOMENT1, MOMENT2	LBC
		PLOAD	LBC
		PLOAD4	Not Supported
		RFORCE	LBC/Follower Force
		GRAV	Analysis/General Parameters

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Lagrangian, Enforced Motions	TLOAD1	LBC (Field)
		TLOAD2	Not Supported
		DAREA	Not Supported
		FORCE	LBC
		FORCE3, FORCEEX	Not Supported
		MOMENT	LBC
	Lagrangian, Initial Conditions	TIC3	LBC/Init. Rotation Field
		TICGP	LBC/Init. Velocity
		TICEL	LBC/Init. Cond. Euler
		TIC, TIC1, TIC2, TICEEX, TICGEX	Not Supported
	Eulerian, Single Point Constraints	ALEGRID, ALEGRID1	LBC/Coupling
		SPC, SPC1, SPC2, SPC3	LBC
	Eulerian, Flow Boundary	TLOAD1	LBC
		FLOW	LBC/Flow
		FLOWEX, FLOWDEF	Not Supported
		PORFLOW	LBC/Coupling
		CFACE	LBC
	Eulerian, Wall	WALLET	LBC/Barrier
	Eulerian, Gravity	GRAV	Analysis/Gen. Parameters

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Eulerian, Initial Conditions	TIC3	LBC/Init. Rotation Field
		TICGP	LBC/Init. Velocity
		TICEUL, TICVAL	LBC/Init. Cond. Euler
		CYLINDER, SPHERE	LBC/Init. Cond. Euler
		MATINI	LBC/Init. Cond. Euler
		BOX	Not Supported
	Eulerian, Container	FFCONTR	LBC/Fluid Filled Containers
	Lagrangian Loading and Constraints	PLOADEX	Not Supported
	Detonation Wave	DETSPH	LBC/Detonation Wave
	Body Force	BODYFOR	LBC/Body Force
	Euler/Lagrange Coupling	COUPLE, COUOPT, COUPOR, COUHTR, COUINFL, COUPLE1, COUP1FL, COUP1INT	LBC/Coupling
		HTRCONV, HTRRAD	LBC/Coupling
		PORFLOW	LBC/Coupling
		SURFACE, SUBSURF, CFACE, SET1	LBC/Coupling
		ALE	LBC/Coupling

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Euler/Lagrange Coupling (continued)	GBAG, GBAGCOU, GBAGPOR, GBAGHTR, GBAGINFL	LBC/Airbag
		COUPLE, COUOPT, COUPOR, COUHTR, COUINFL	LBC/Airbag
		GBAGC	Not Supported
		HTRCONV, HTRRAD	LBC/Airbag
		INFLATR, INFLATR1, INFLHYB, INFLHYB1, INFLFRAC, INFLGAS, INFLTANK, INITGAS	LBC/Airbag
		PERMEAB, PERMGBG, PORHOLE, PORLHOLE, PORFCPL, PORFGBG, PORFLCPL, PORFLGBG	LBC/Airbag
		POREX	Not Supported
		SURFACE, SUBSURF, CFACE, SET1	LBC/Airbag
	Parameters	PARAM	Analysis
	Tabular Input	TABLED1	Fields
		TABLEEX	Not Supported

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Bulk Data (continued)	Miscellaneous	IGNORE, MADGRP, SECTION, SGAUGE, USA	Not Supported
	Sets	SET1, SETC	Analysis/Special Features
	Solution Control	ACTIVE	Analysis/Entity Activation
		VISCDMP	Analysis/Dynamic Relaxation
	Output	SECTION	Not Supported
	Prestress Analysis	NASINIT	Analysis/Initiating Calculations
	Include File Control	INCLUDE	Automatic
	Bulk Data Control	BEGIN BULK	Automatic
		END DATA	Automatic
Parameter Options	Coupling Subcycling	COSUBCYC, COSUBMAX	Analysis/Sub Cycling Params
	Blending Control	DELCLUMP, FBLEND	Analysis/Coupling Parameter
	Time Step Control	INISTEP, MAXSTEP, MINSTEP, STEPFCT, STEPFCTL	Analysis/Execution Control
	License Control	AUTHQUEUE	Analysis/Execution Control
	Mass Scaling	SCALEMAS	Analysis/Execution Control
	Limits	FMULTI, LIMITER, MICRO, RHOCUT, RKSCHEME, ROHYDRO, ROMULTI, ROSTR	Analysis/Eulerian Parameter
		SNDLIM	Analysis/General Parameters
		VELCUT, VELMAX	Analysis/Eulerian Parameter

Table 2-1 Supported Dytran Commands (continued)

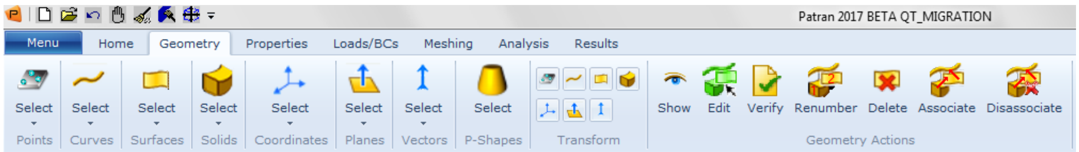
File Section	Subsection	Data Entry	Method
Parameter Options (continued)	Restart Control	RSTDROP	Analysis/Restart Control
	Ale Motion Control	ALEITR, ALETOL, ALEVER	Analysis/ALE Parameters
	Coupling Control	FASTCOUP	Analysis/Coupling Parameters
	Contact Control	CONTACT, LIMCUB	Analysis/Contact Parameters
	Miscellaneous	CFULLRIG	Analysis/General Parameters
		EULTRAN	Analysis/Eulerian Parameter
	Miscellaneous (continued)	EXTRAS	Analysis/User Subroutine Par
		GEOCHECK	Analysis/General Parameters
		IMM	Analysis/Initiating Calculations
		MATRMRG1	Analysis/Rigid Body Merging
		MIXGAS	Analysis/Eulerian Parameter
		NZEROVEL	Analysis/Default Gridpoint
		RJSTIFF	Analysis/General Parameters
		UGASC	Analysis/Eulerian Parameter
		VARACTIV	Analysis/Variable Activation
		CLUMPENER, ENTROPY-FIX, FAILDT , FLOW - METHOD, HYDROBOD, IGNFRCER, MATRMERC, OLDLAGTET, PARALLEL, PLCOVCUT, TOLCHK, USA_CAV	Not Supported

Table 2-1 Supported Dytran Commands (continued)

File Section	Subsection	Data Entry	Method
Parameter Options (continued)	Material Parameter Control	BULKL, BULKQ, BULKTYP	Analysis/Bulk Viscosity Params
		HGCMEM, HGCSOL, HGCTWS, HGCWRP, HGSHELL, HGSOLID	Analysis/Hourglass Params
		HVLFAIL, PMINFAIL	Analysis/General Parameters
		HGCOEFF, HGTYPE	Not Supported
	Shell Options	SHELLFORM, SHELMSYS, SHPLAST, SHTHICK, SLELM	Analysis/General Parameters
	Dynamic Relaxation	VDAMP	Analysis/Dynamic Relaxation
	ATB Positioning	ATBSEGCREAT	Not Supported
	Output Control	ATBAOUT, ATBHOUTPUT, ATBTOUT, AUTHOINFO, CONM2OUT, ELDLTH, FAILOUT, IEEE, INFO-BJOIN, MESHELL, MESHPLN, NASIGN, RBE2INFO, SHSTRDEF, STRNOUT	Analysis/Output Controls
		ERRUSR, HICGRAV	Not Supported
Parameter Options (continued)	Pre-Stressing Analysis	INITFILE, INITNAS	Analysis/Initiating Calculation
	Initial Metric Method	IMM	Analysis/Initiating Calculation

Coordinate Frames

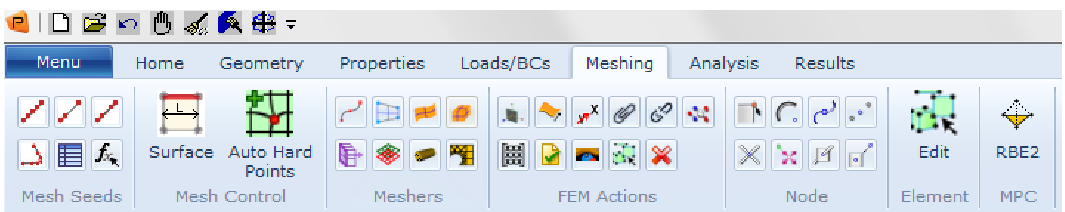
Coordinate frames will generate unique CORD1R or CORD2R entries.



Only Coordinate Frames which are referenced by nodes, element properties, or loads and boundary conditions can be translated. For more information on creating coordinate frames see Creating Coordinate Frames (p. 447) in the *Geometry Modeling - Reference Manual Part 2*.

Finite Elements

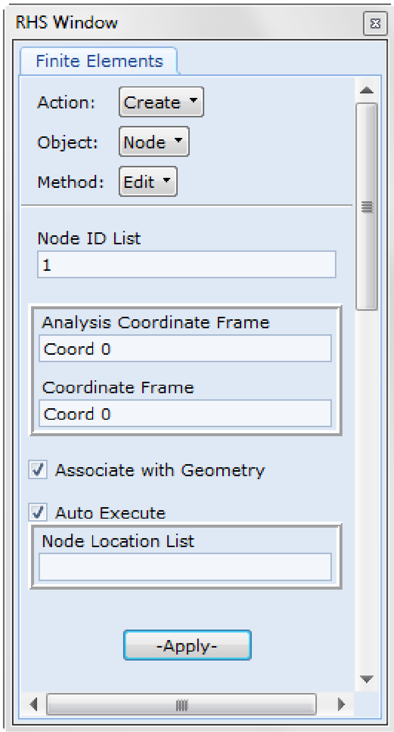
Finite Elements in Patran allows the definition of basic finite element construction. Created under Finite Elements are the nodes, element topology and multi-point constraints.



For more information on how to create finite element meshes, see Mesh Seed and Mesh Forms (p. 22) in the *Reference Manual - Part III*.

Nodes

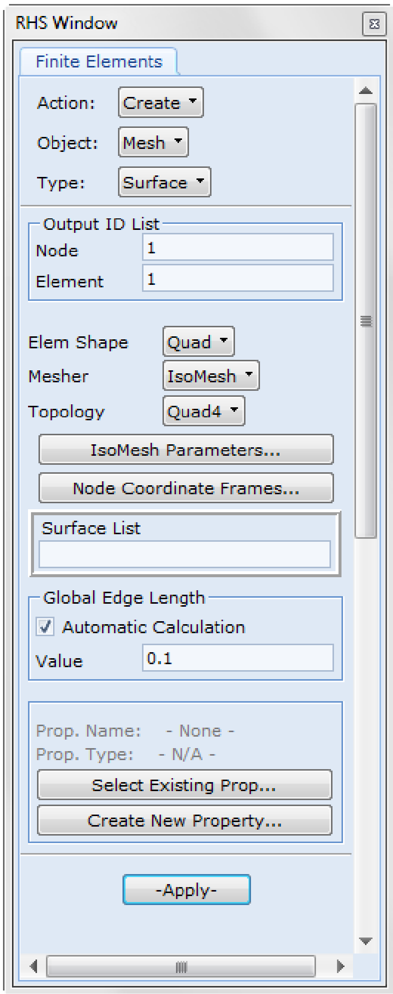
Nodes in Patran will generate unique GRID Bulk Data entries. Nodes can be created either directly using the Node object, or indirectly using the Mesh object.



Parameter	Description
Analysis Coordinate Frame **Coordinate Frame**	The analysis frame is not used anywhere by Dytran. The Reference Coordinate system is used during node generation only.

Elements

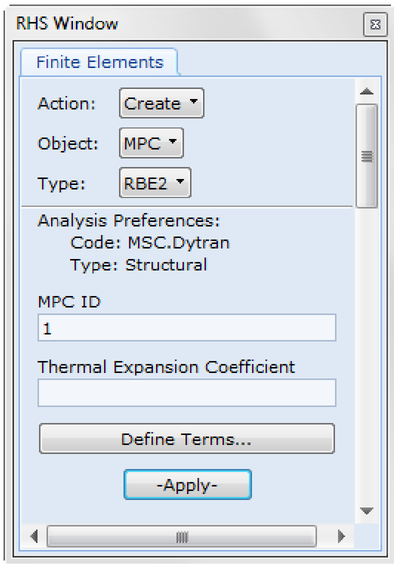
Finite Elements in Patran assigns element connectivity, such as Quad/4, for standard finite elements. The type of Dytran element created is not determined until the element properties are assigned. See the Element Properties Form for details concerning the Dytran element types. Elements can be created either discretely using the Element object or indirectly using the Mesh object.



Parameter	Description
Output ID list	Elements which are not referenced by an element property region which is understood by the Patran Dytran forward translator will not be translated.

Multi-Point Constraints

Multi-point constraints (MPCs) can also be created from the Finite Elements menu. These elements define a rigorous behavior between several specified nodes. The forms for creating MPCs are found by selecting MPC as the Object on the Finite Elements form. The full functionality of the MPC forms are defined in Create Action (Mesh) (p. 9) in the *Reference Manual - Part III*.



Parameter	Description
MPC ID	Used to specify the ID to associate to the MPC when it is created.

MPC Types

To create an MPC, first select the type of MPC to be created from the option menu. The MPC types that appear in the option menu are dependent on the current settings of the Analysis Code and Analysis Type preferences. The following table describes the MPC types which are supported for Dytran.

MPC Type	Analysis Type	Description
RBE2	Structural	Creates a constraint equation between one degree of freedom of one node and selected degrees of freedom of other nodes.

Degrees-of-Freedom

Whenever a list of degrees-of-freedom is expected for an MPC term, a listbox containing the valid degrees-of-freedom is displayed on the form. A degree-of-freedom is valid if:

1. It is valid for the current Analysis Code Preference.

2. It is valid for the current Analysis Type Preference.
3. It is valid for the selected MPC type.

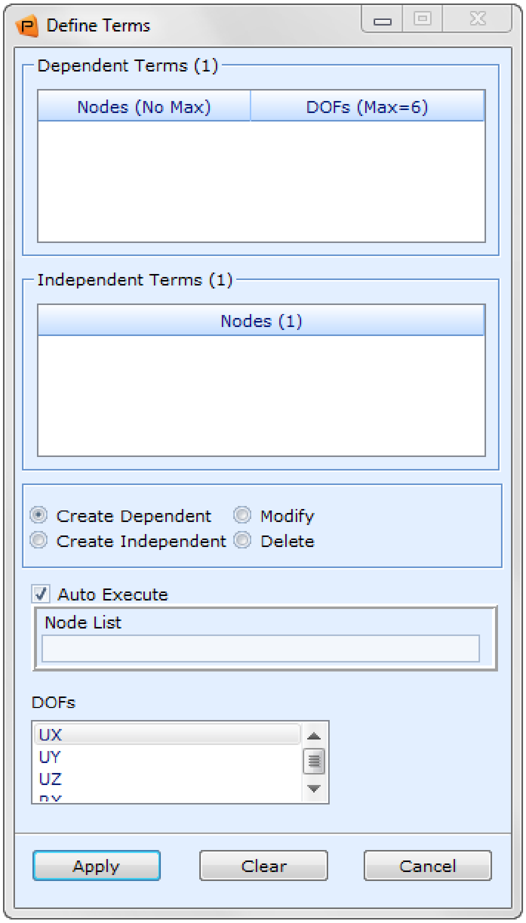
In most cases, all degrees-of-freedom, which are valid for the current Analysis Code and Analysis Type preferences, are valid for the MPC type.The following degrees-of-freedom are supported for the various analysis types:

Degree-of-freedom	Analysis Type
UX	Structural
UY	Structural
UZ	Structural
RX	Structural
RY	Structural
RZ	Structural

Note: Care must be taken to make sure that a degree-of-freedom that is selected for an MPC actually exists at the nodes. For example, a node that is attached only to solid structural elements will not have any rotational degrees-of-freedom. However, Patran will allow you to select rotational degrees-of-freedom at this node when defining an MPC.

RBE2 MPCs

This subordinate MPC form appears when the Define Terms button is selected on the Finite Elements form. This form is used to create a RBE2 Bulk Data entry.



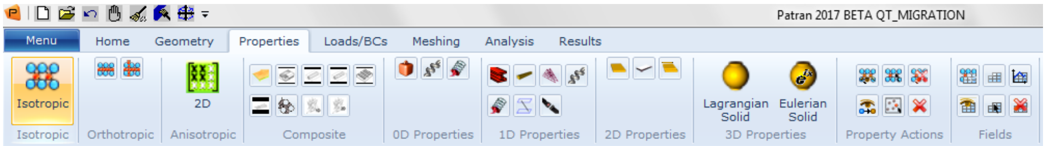
Parameter	Description
Dependent Terms	Holds the dependent term information.
Independent Terms	Holds the independent term information.

Material Library

The Materials form will appear when the Material toggle, located on the Patran application selections, is chosen. The selections made on the Materials menu will determine which material form appears, and ultimately, which Dytran material will be created.

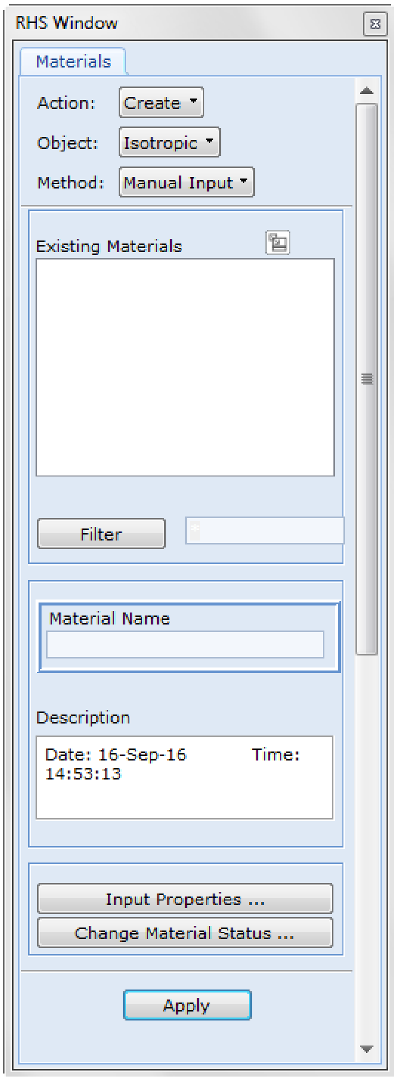
The following pages give an introduction to the Materials form, and details of all the material property definitions supported by the Patran Dytran Preference.

Only material records which are referenced by an element property region will be translated. References to externally defined materials will result in special comments in the Dytran input file, with material data copied from user identified files. This reference allows a user not only to insert material types that are not supported directly by the Dytran preference, but also to make use of a standard library of materials.



Materials Form

This form appears when Materials is selected on the main form. The Materials form is used to provide options to create the various Dytran materials



Parameter	Description
Object	This toggle defines the basic material orthotropy, and can be set to Isotropic, 2D Orthotropic, 3D Orthotropic, 2D Anistropic or Composite.
Method	The method may be Manual Input or Externally Defined. If it is set to Externally Defined, this form will have an “Apply” button which is used to ensure that the needed material is added to the set of available materials.
Existing Materials	Lists the created materials whose names pass the filter.
Material Name	Defines the material name. A unique material ID will be assigned during translation.
Description	Describes the material that is being created.
Input Properties	Generates a form that is used to define the material properties.
Change Material Status	Generates a form that is used to indicate the active portions of the material mode. By default, all portions of a created material model are active.

The following table outlines the options when Create is the selected Action.

Table 2-2 Materials

Object	Option 1
Isotropic	■ LinElas (DMATEL) ■ LinElas (MAT1) ■ LinElas (DMATEP) ■ LinElas (DMAT) ■ LinFluid (DMAT) ■ Ideal Gas (DMAT) ■ Tait Cavitation Model (DMAT) ■ JWL Explosive (DMAT) ■ Ignition and Growth (DMAT) ■ NonLinElas (DMAT) ■ NonLinPlas (DMAT) ■ NonLinFluid (DMAT) ■ User Equation of State (DMAT) ■ LinViscoElas (DMAT) ■ Rigid (MATRIG) ■ Soil (DYMAT14) ■ Soil (DYMAT25) ■ Foam (DYMAT14) ■ Foam (FOAM1) ■ Foam with Hysteresis (FOAM2) ■ Concrete (DYMAT25) ■ Rock (DYMAT25) ■ Cowper-Symonds (DYMAT24) ■ ElasPlas (DYMAT24) ■ ElasPlas (DMATEP) ■ Johnson-Cook (DMAT) ■ Snow (DMAT) ■ ElasPlas (DMAT) ■ Rubber (RUBBER1) ■ Linear Elastic
2D Orthotropic	■ LinElas (MAT8) ■ Woven Fabric (FABRIC) ■ Linear Elastic

Table 2-2 Materials

Object	Option 1
3D Orthotropic	▪ Sheetmaterial (SHEETMAT) ▪ ElasFail (DMATOR) ▪ Honeycomb (DYMAT26)
2D Anisotropic	▪ LinElas (MAT2) ▪ Linear Elastic
Composite	▪ Laminate

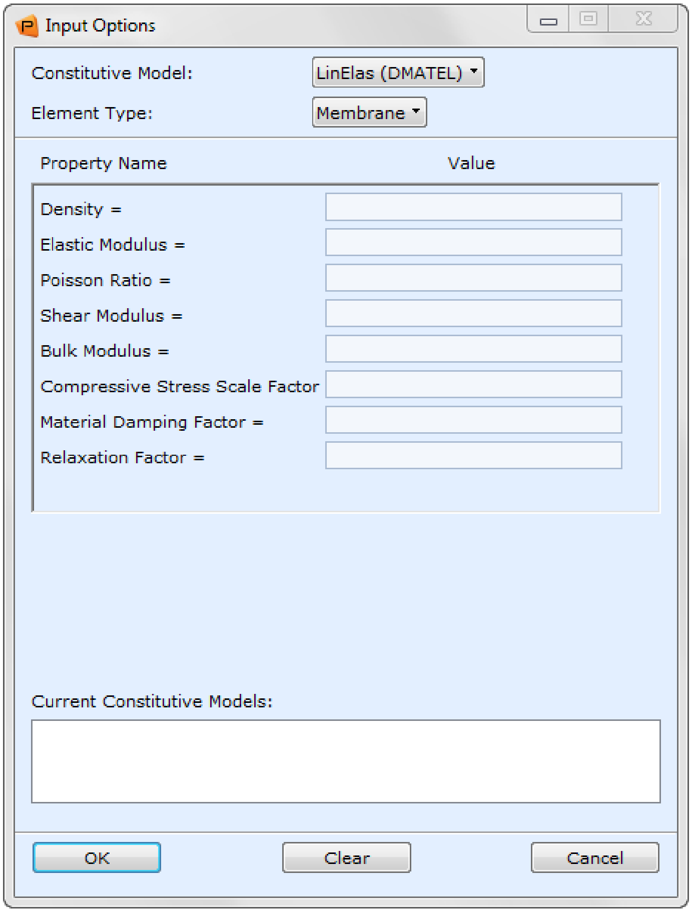
Isotropic

Linear Elastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and when one of the LinElast options is the selected Constitutive Model on the Input Options form.

Option 1	Option 2
Linear Elastic	MAT1 DMATEL DMATEP DMAT

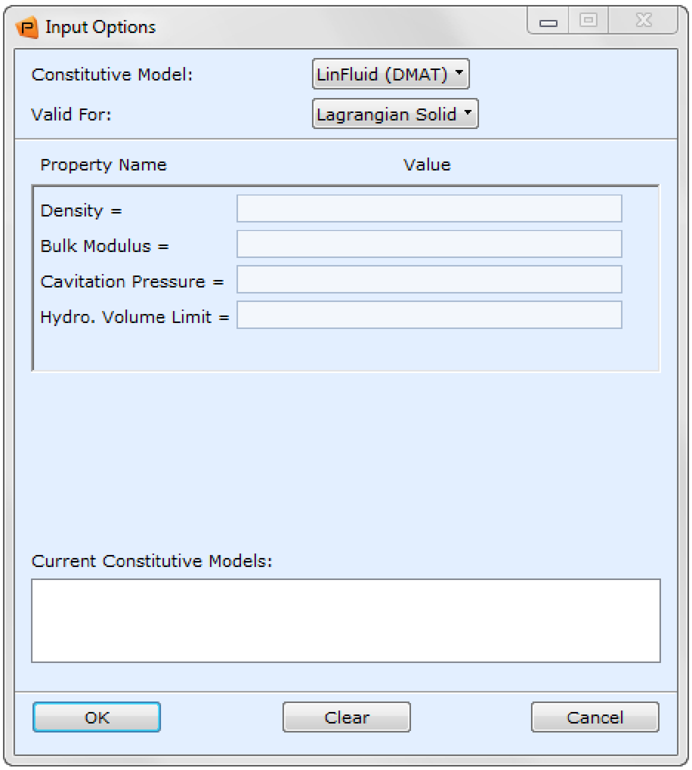
Use this form to define a linear elastic material using one of the 4 available Dytran descriptions.



Parameter	Description
Constitutive Model	Choose here among the 4 available LinElas implementations.
Element Type	The choices available here will depend upon the model selected above. During translation the element type will be checked against the type set here.
Relaxation Factor	The entry here, if present, will depend upon the selection made above. Consult the dytran User's manual for the Bulk Data entry identified in the constitutive model name.

Linear Fluid

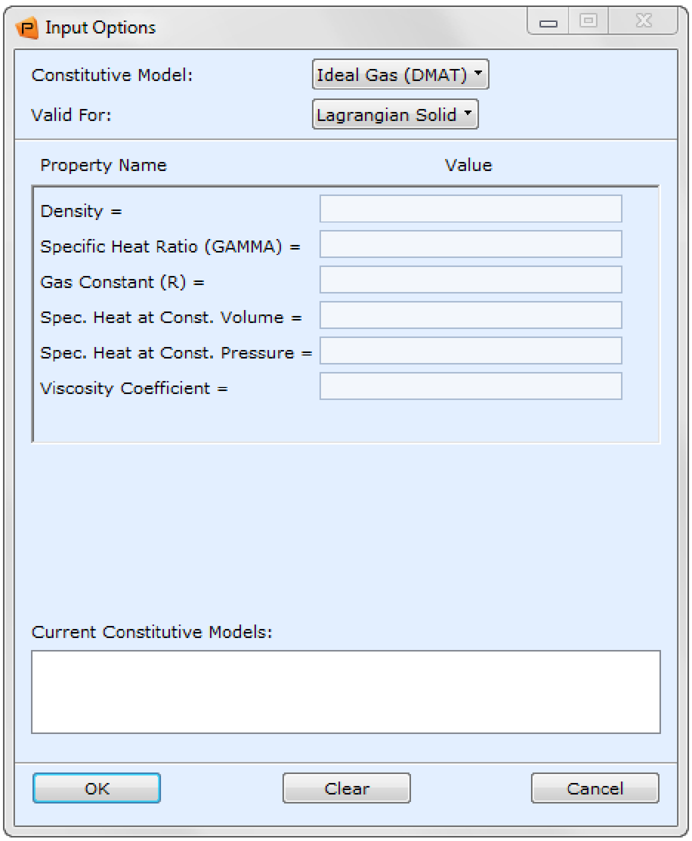
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and when the LinFluid (DMAT) option is selected as the Constitutive Model on the Input Options form. Use this form to define a linear fluid material using the DMAT description.



Parameter	Description
Valid For	Choose between Lagrangian Solid and Eulerian Solid (Hydro).

Ideal Gas

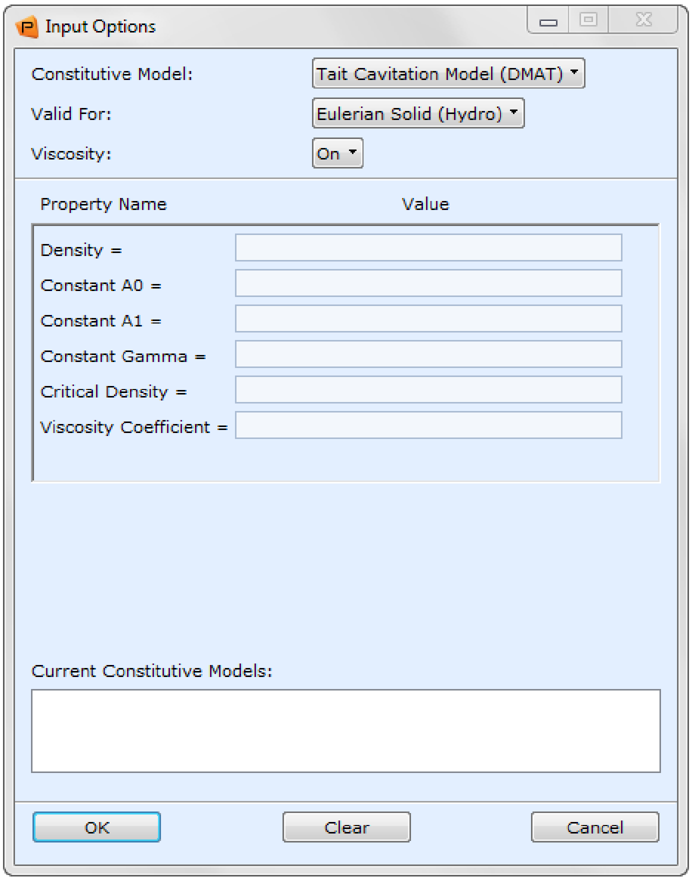
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and when the Ideal Gas (DMAT) option is selected as the Constitutive Model on the Input Options form. Use this form to define an ideal gas using the DMAT description.



Parameter	Description
Valid For	Choose between Lagrangian Solid and Eulerian Solid (Hydro).

Tait Cavitation Model

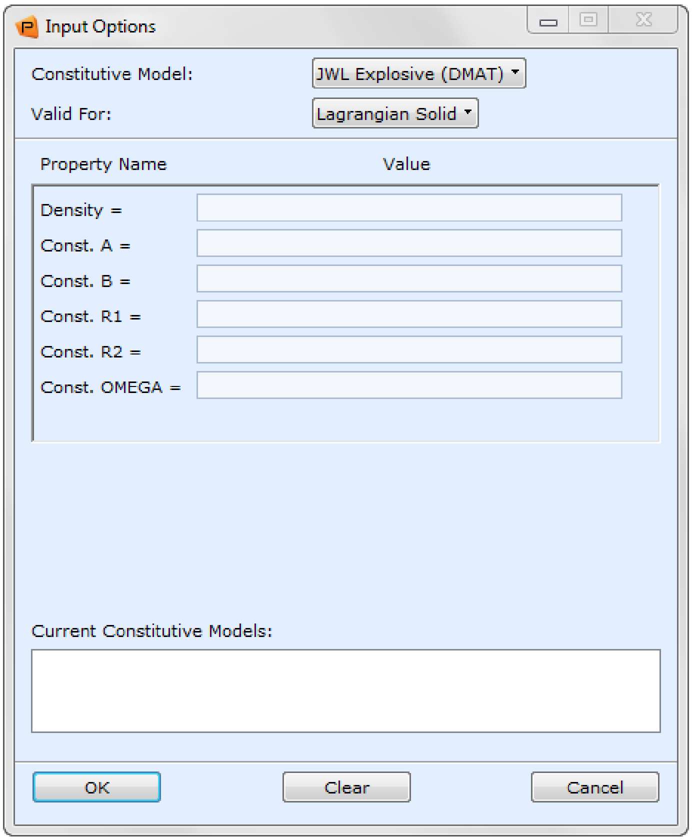
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and when the Tait Cavitation Model (DMAT) option is selected as the Constitutive Model on the Input Options form. Use this form to define a Tait Cavitation Model using the DMAT description.



Parameter	Description
Valid For	Eulerian Solid (hydro) only.
Viscosity	Viscosity Options: On Off
Viscosity Coefficient	Only appears when Viscosity is set to On.

JWL Explosive (DMAT)

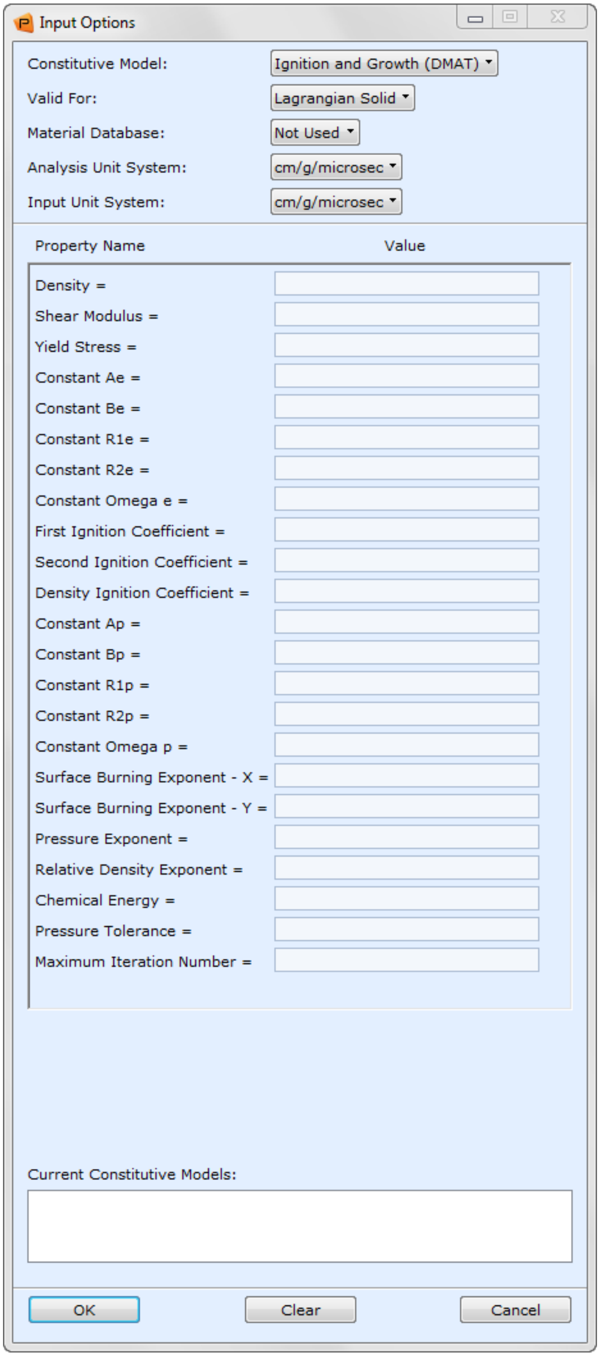
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the JWL Explosive (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define a JWL explosive material using the DMAT description.



Parameter	Description
Valid For	Choose between Lagrangian Solid and Eulerian Solid (Hydro).

Ignition and Growth

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Ignition and Growth (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define an Ignition and Growth DMAT description.


Input Options
Constitutive Model:
Ignition and Growth (DMAT)
Valid For:
Lagrangian Solid
Material Database:
Not Used
Analysis Unit System:
cm/g/microsec
Input Unit System:
cm/g/microsec
Property Name
Value
Density =
Shear Modulus =
Yield Stress =
Constant Ae =
Constant Be =
Constant R1e =
Constant R2e =
Constant Omega e =
First Ignition Coefficient =
Second Ignition Coefficient =
Density Ignition Coefficient =
Constant Ap =
Constant Bp =
Constant R1p =
Constant R2p =
Constant Omega p =
Surface Burning Exponent - X =
Surface Burning Exponent - Y =
Pressure Exponent =
Relative Density Exponent =
Chemical Energy =
Pressure Tolerance =
Maximum Iteration Number =
Current Constitutive Models:
OK
Clear
Cancel

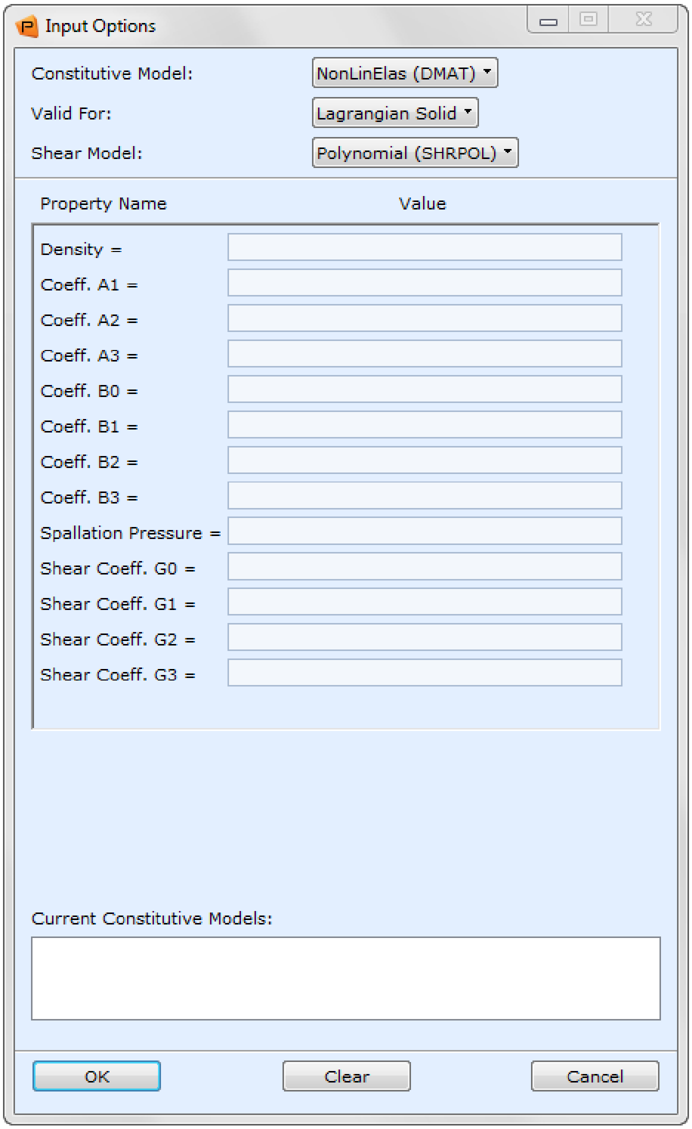
Parameter	Description
Valid For	Applicable for Lagrangian Solid and Eulerian Solid (Hydro) Elements.
Material Database	Choose between: Not Used, PBX-9404 (a), TATB, PETN, Cast TNT, LANL COMP B, Military COMP B, PBX-9404 (b), LX17.
Analysis Unit System	Choose between: cm/g/microsec, SI, Metric, Imperial, mm/mg/microsec.
Input Unit System	If the Material Database is set to Not Used choose between: cm/g/microsec, SI, Metric, Imperial, mm/mg/microsec Otherwise, the only option is N/A.

Note:	If Material Database is not set to Not Used, the only available parameters are Shear Modulus, Yield Stress, Pressure Tolerance and Maximum Iteration Number.

Non Linear Elastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the NonLinElas (DMAT) option is the selected. Constitutive Model on the Input Options form. Use this form to define a non linear elastic using a DMAT description

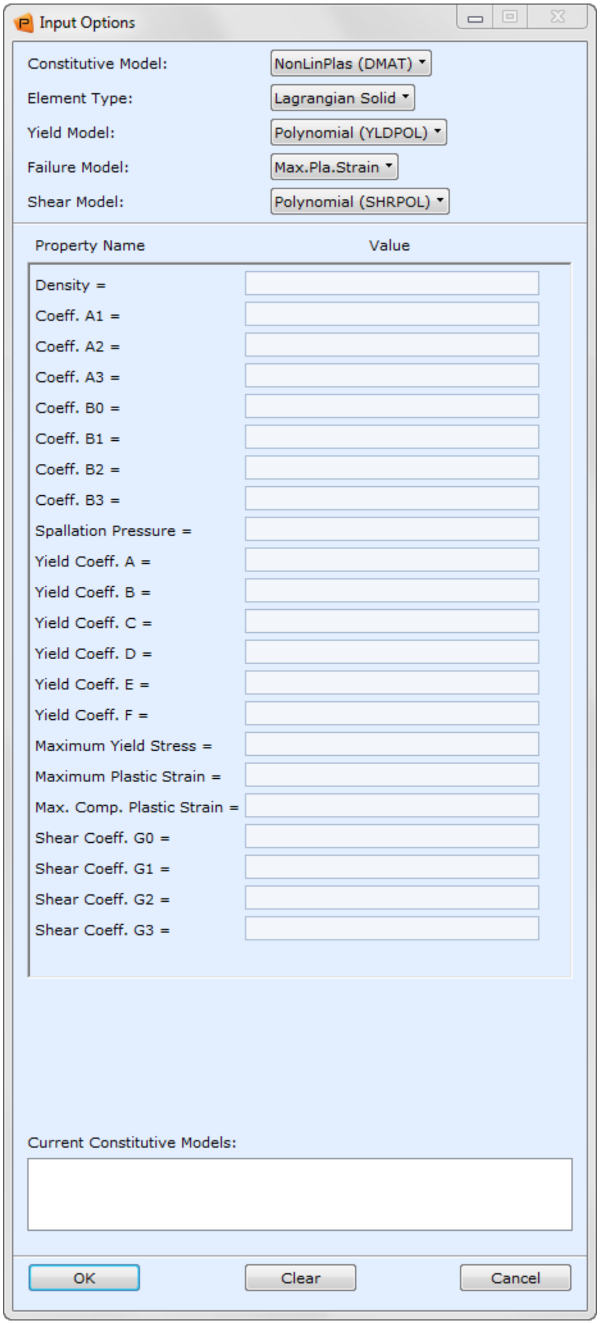
.



Parameter	Description
Valid For	Choose between: -Lagrangian Solid -Eulerian Solid (Strength).
Shear Model	Choose between: -Polynomial (SHRPOL) -User Subroutine (SHREX).
Shear Coeff G0 **Shear Coeff G1** **Shear Coeff G2** **Shear Coeff G3**	Only for Polynomial Shear Model.

Non Linear Plastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the NonLinPlas (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define a non linear plastic using a DMAT description


Input Options
Constitutive Model:
NonLinPlas (DMAT)
Element Type:
Lagrangian Solid
Yield Model:
Polynomial (YLDPOL)
Failure Model:
Max.Pla.Strain
Shear Model:
Polynomial (SHRPOL)
Property Name
Value
Density =
Coeff. A1 =
Coeff. A2 =
Coeff. A3 =
Coeff. B0 =
Coeff. B1 =
Coeff. B2 =
Coeff. B3 =
Spallation Pressure =
Yield Coeff. A =
Yield Coeff. B =
Yield Coeff. C =
Yield Coeff. D =
Yield Coeff. E =
Yield Coeff. F =
Maximum Yield Stress =
Maximum Plastic Strain =
Max. Comp. Plastic Strain =
Shear Coeff. G0 =
Shear Coeff. G1 =
Shear Coeff. G2 =
Shear Coeff. G3 =
Current Constitutive Models:
OK
Clear
Cancel

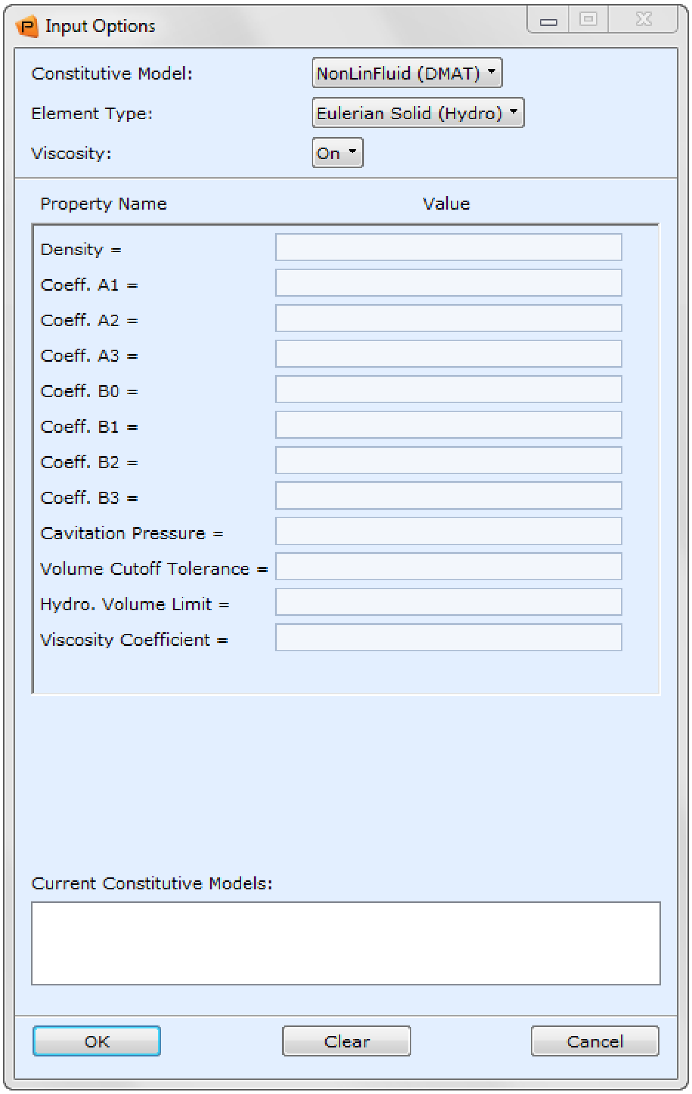
Parameter	Description
Element Type	Choose between: -Lagrangian Solid -Eulerian Solid (Strength)
Yield Model	Choose between: -Polynomial (YLDPOL) -User Subroutine (YLDEX)
Failure Model	Choose between: -Max. Pla. Strain -Max. Equ. Stress -Max. Pla. Strain Timestep -User Subroutine -None
Shear Model	Choose between: -Polynomial (SHRPOL) -User Subroutine (SHREX)

Note:

1. When Element Type is Eulerian Solid, the only Failure options are None, Max. Pla. Strain and User Subroutine
2. Only show Maximum Plastic Strain for Failure options Max. Pla. Strain and Max. Pla. Strain Timestep
3. Only show Maximum Equivalent Stress for Failure option Max. Equ. Stress
4. Only show Minimum Time Step for Failure option Max. Pla. Strain Timestep
5. Only show Volume Cutoff Tolerance for the Element Type option Eulerian Solid
6. Only show Yield Coefficients for the Yield Model option Polynomial
7. Only show Shear Coefficients for the Shear Model option Polynomial

Non Linear Fluid

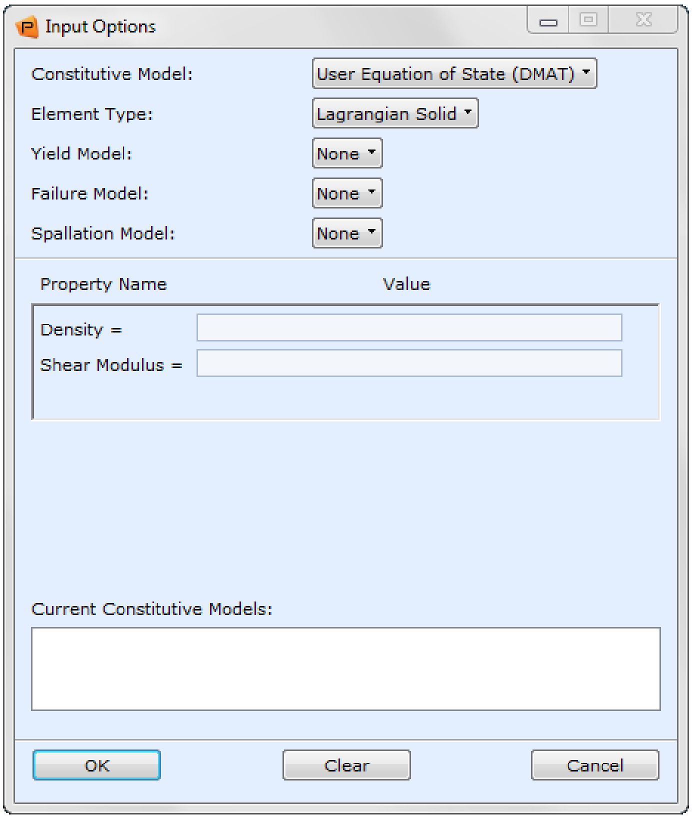
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Non Linear Fluid (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define a non linear fluid using a DMAT description.



Parameter	Description
Element Type	Choose between Lagrangian Solid and Eulerian Solid (Hydro).
Viscosity	Viscosity Options: Off On (only for Eulerian Solid)

User Equation of State

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the User Equation of State (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define a User Equation of State using a DMAT description.

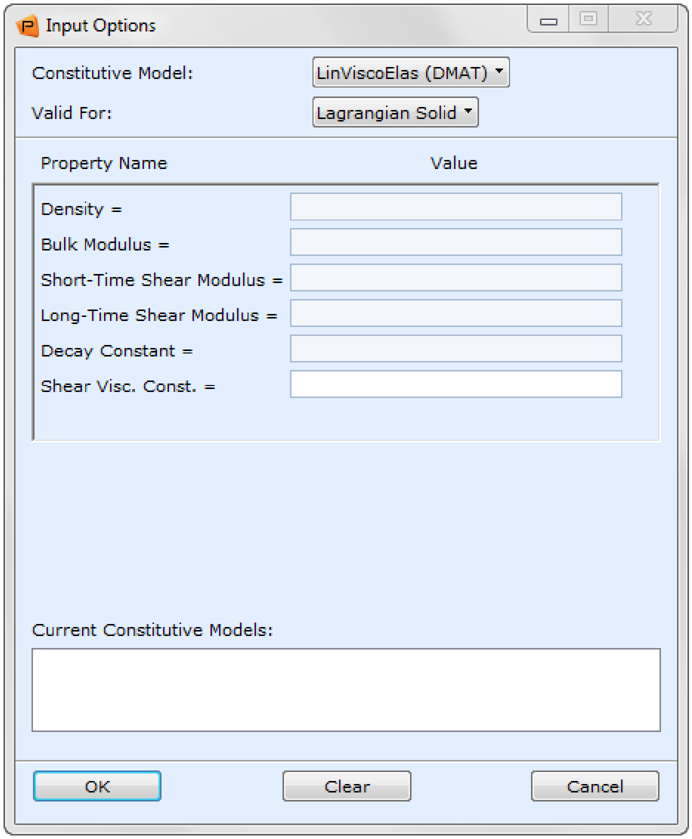


Parameter	Description
Element Type	Choose between: -Lagrangian Solid -Eulerian Solid (Hydro) -Eulerian Solid (Strength)
Yield Model	Choose between: -None -Von Mises
Failure Model	Choose between: -None -Max. Pla. Strain -Max. Equ. Stress -Max. Pla. Strain Timestep -User Subroutine
Spallation Model	Choose between: -None -Spallation Pressure
Property Name: Additional Parameters	Additional parameters are: -Yield Stress -Maximum Plastic Strain -Max. Comp. Plastic Strain -Maximum Equiv. Stress -Minimum Time Step -Spallation Pressure -Volume Cutoff Tolerance

Note:	
	1. When Element Type is Eulerian Solid (both), the only Failure options are None, Max. Pla. Strain and User Subroutine 2. Only show Yield Stress for option Von Mises 3. Only show Maximum Plastic Strain for options Max. Pla. Strain and Max. Pla. Strain Timestep 4. Only show Maximum Equivalent Stress for option Max. Equ. Stress 5. Only show Minimum Time Step for option Max. Pla. Strain Timestep 6. Only show Spallation Pressure for option Spallation Pressure 7. Only show Volume Cuttoff Tolerance for Eulerian Solid (both)

Linear Viscoelastic

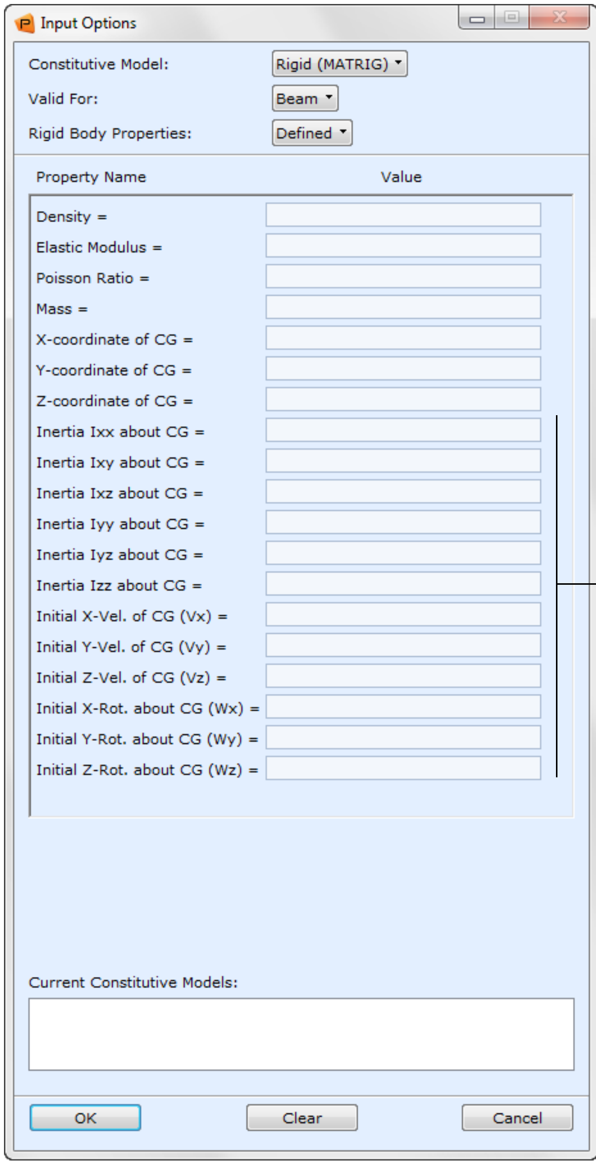
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Linear ViscoElastic (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define a linear viscoelastic material using a DMAT description.



Parameter	Description
Valid For	This model is only applicable for Lagrangian solid elements.

Rigid

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Rigid (MATRIG) option is the selected Constitutive Model on the Input Options form. Use this form to define a rigid material using the MATRIG description.



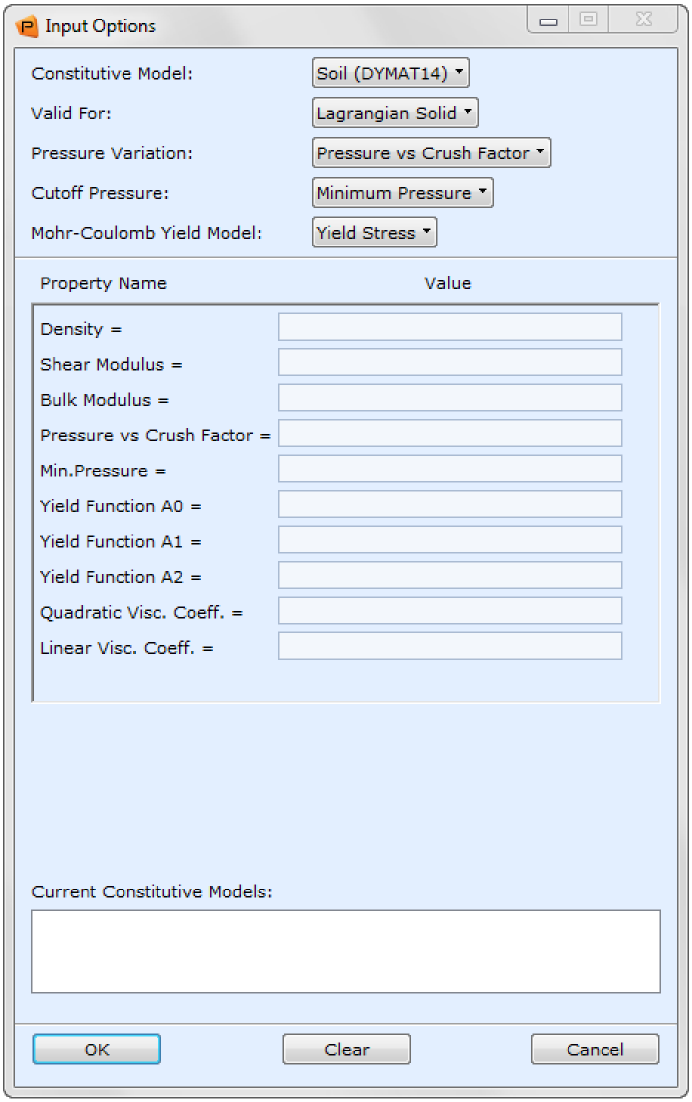
None of this data is required if properties are calculated from the geometry.

Parameter	Description
Valid For	This material is valid for beams, shells, and Lagrangian solids.
Rigid Body Properties	Choose between defining the body properties (center of gravity and inertia) and calculating data within Dytran, from the geometry.

Note:	Local coordinate system (CID) can be defined in the form under Analysis -> Execution Control -> Add CID to MATRIG.

Soil (DMAT14)

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Soil (DYMAT14) option is the selected Constitutive Model on the Input Options form. Use this form to define a soil using the DYMAT14 model.

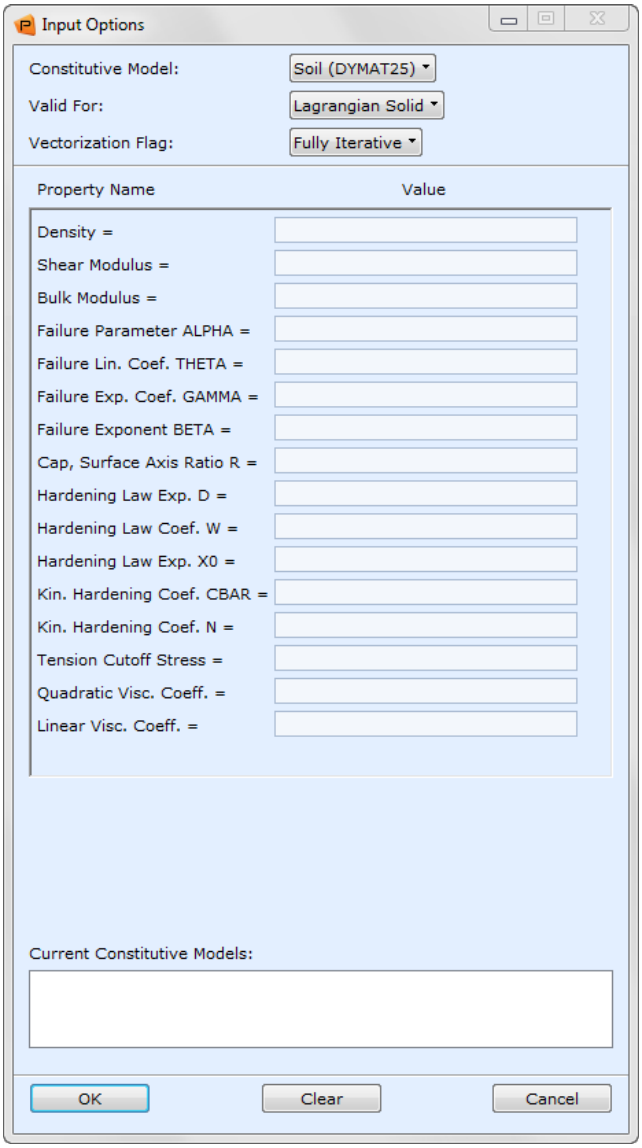


Parameter	Description
Valid For	This material is valid for Lagrangian solids only.
Pressure Variation	Choose between Pressure vs. Crush Factor and Pressure vs. Volumetric Strain.
Cut-Off Pressure	Choose between defining Minimum Pressure, Failure Pressure, and Calculated Cutoff Pressure.

Parameter	Description
Mohr-Coulomb Yield Model	Choose between Yield Stress, Dytran Yield Surface, and Patran LS-DYNA3D Yield Surface.
Pressure vs. Crush factor	This entry depends upon the choice of Pressure Variation definition. This is defined as a strain dependent field, in which Crush or Strain are both treated as strains.
Min Pressure	This entry, if present, depends upon the choice of the Cutoff Pressure method.

Soil (DYMAT25)

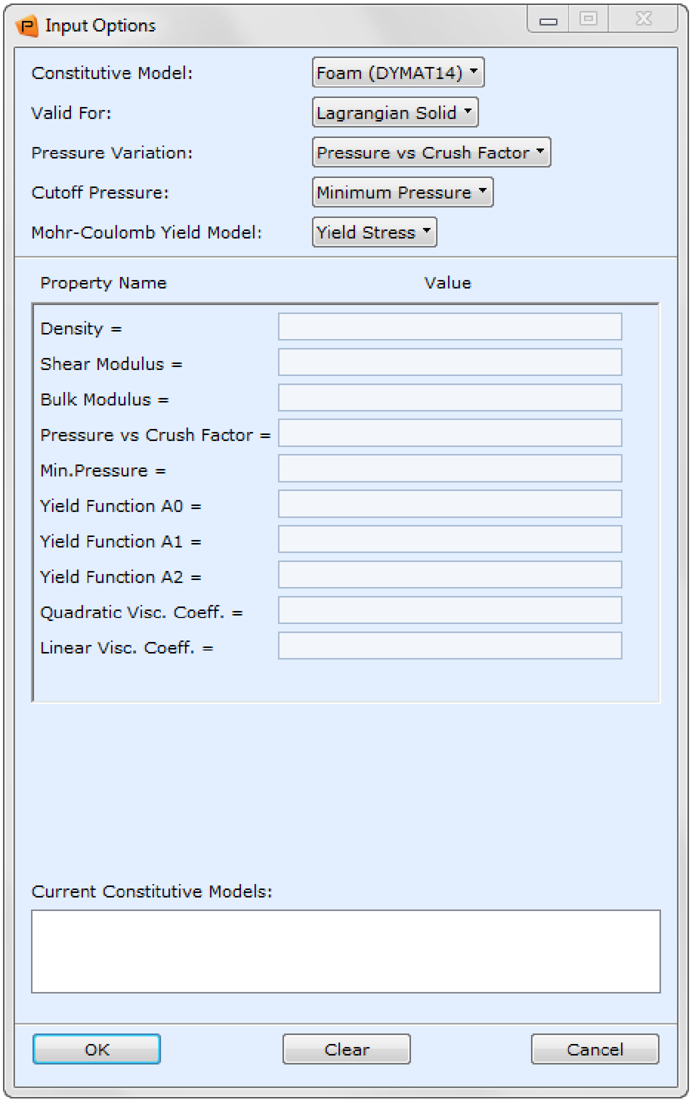
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Soil (DYMAT25) option is the selected Constitutive Model on the Input Options form. Use this form to define a soil using the DYMAT25 mode.



Parameter	Description
Valid For	Only Lagrangian Elements.
Vectorization Flag	Vector Options: Fully Iterative Vectorized

Foam (DMAT14)

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Foam (DYMAT14) option is the selected Constitutive Model on the Input Options form. Use this form to define a foam using the DYMAT14 model.

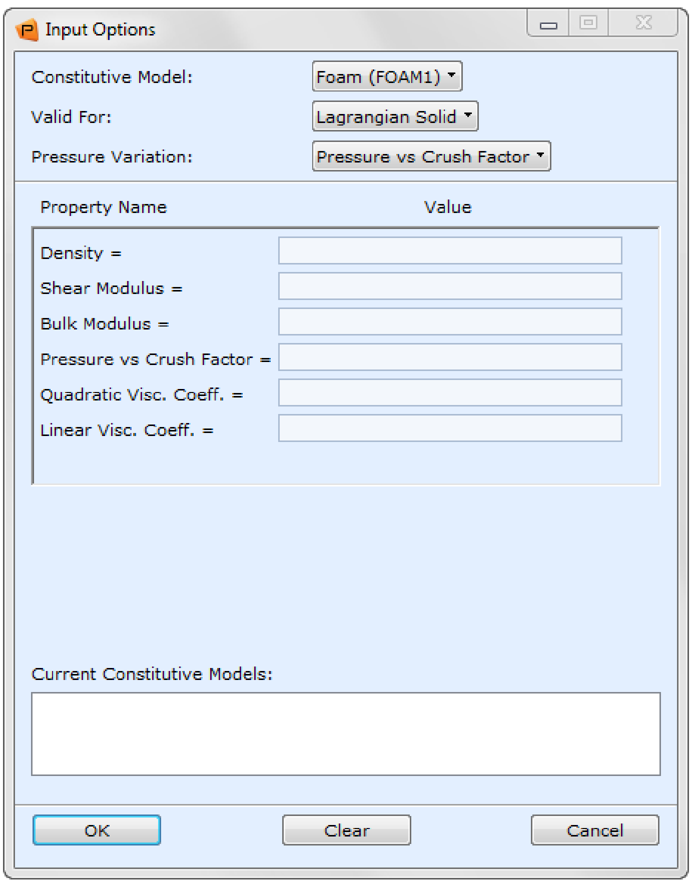


Parameter	Description
Valid For	This material is valid for Lagrangian solids only.
Pressure Variation	Choose between Pressure vs Crush Factor and Pressure vs Volumetric Strain.
Cutoff Pressure	Choose between defining Minimum Pressure, Failure Pressure, and Calculated Cutoff Pressure.
Mohr Coulomb Yield Model	Choose between Yield Stress, Dytran Yield Surface, and Patran LS-DYNA3D Yield Surface.

Parameter	Description
Pressure Vs Crush Factor	This entry depends upon the choice of Pressure Variation definition. This is defined as a strain dependent field, in which Crush or Strain are both treated as strains.
Min. Pressure	This entry, if present, depends upon the choice of Cutoff Pressure method.

Foam (FOAM1)

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Foam (FOAM1) option is the selected Constitutive Model on the Input Options form. Use this form to define a foam using the FOAM1 model.

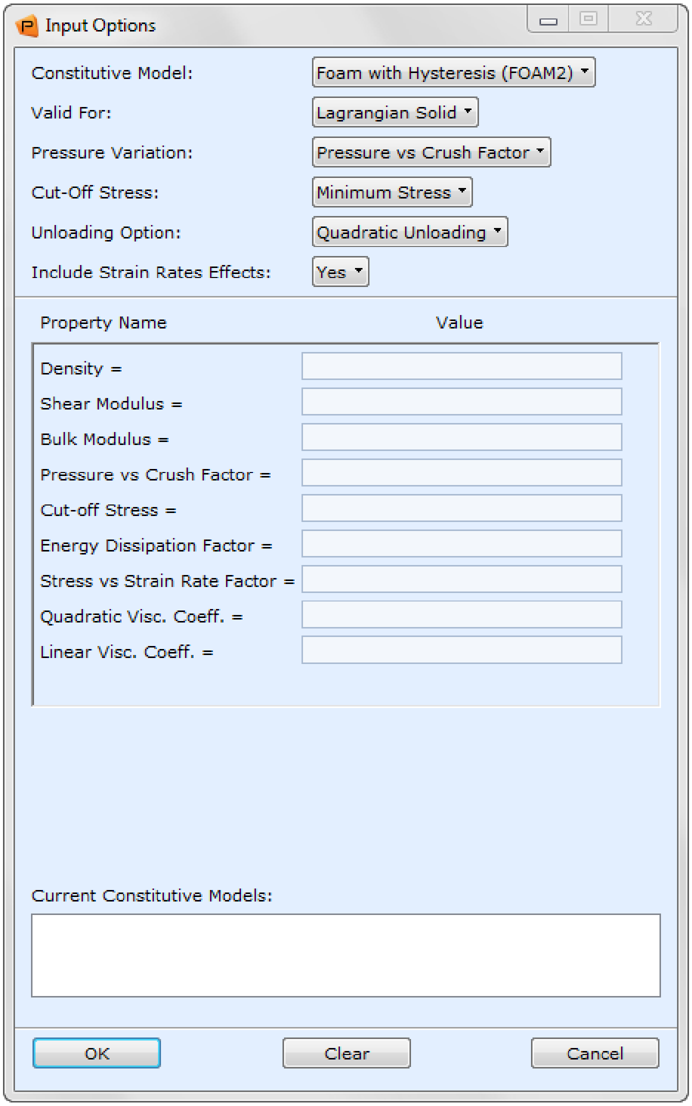


Parameter	Description
Valid For	This model is only applicable for Lagrangian solid elements.

Parameter	Description
Pressure Variation	Choose between Pressure vs Crush Factor and Pressure vs Volumetric Strain.
Pressure Vs Crush Factor	This entry depends upon the choice of Pressure Variation definition. This is defined as a strain dependent field, in which Crush or Strain are both treated as strains.

Foam with Hysteresis (FOAM2)

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Foam with Hysteresis (FOAM2) option is the selected Constitutive Model on the Input Options form. Use this form to define a foam using the FOAM2 model.

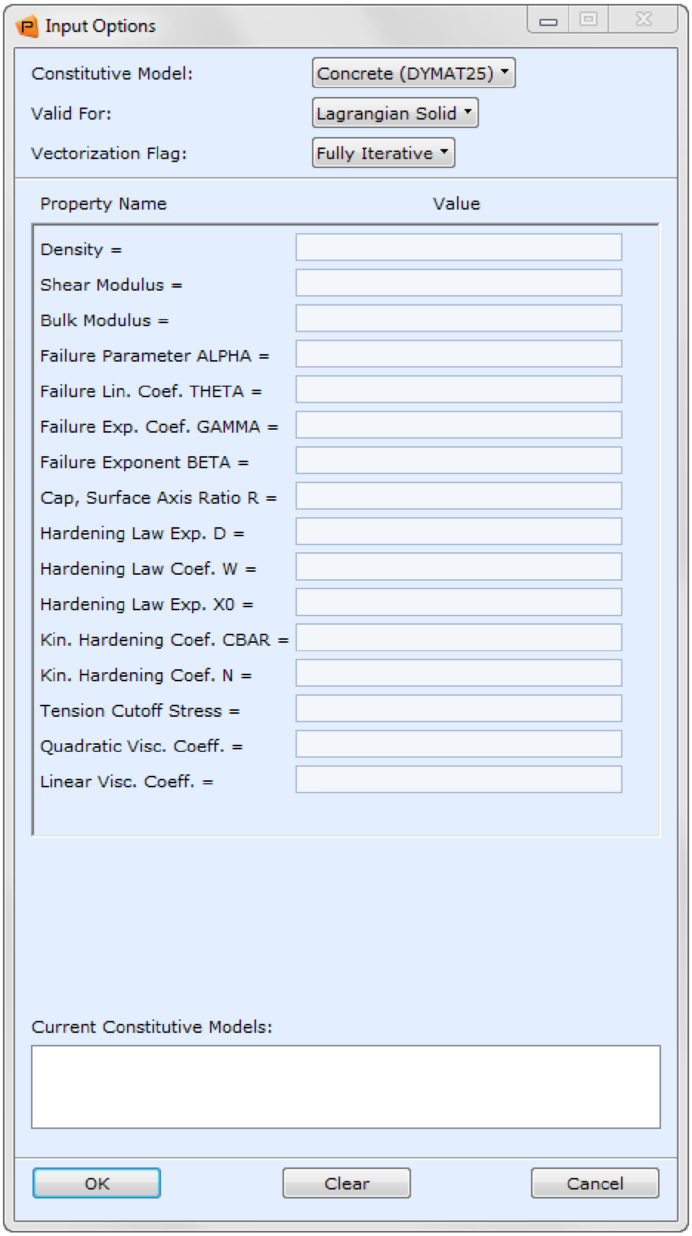


Parameter	Description
Valid For	This material is valid for Lagrangian solids only.
Pressure Variation	Choose between: Pressure vs. Crush Factor Pressure vs. Vol. Strain

Parameter	Description
Cut-Off Stress	Choose between: Minimum Stress or Stress for Tensile Failure.
Unloading Option	Unloading options: Quadratic Linear Exponential
Include Stress Strain Rate Effects	Stress Strain Effects: Yes No
Pressure vs. Crush Factor	Name of this field equals the value of option Pressure Variation.
Stress vs. Strain Rate Factor	Only appears when Stress Strain Effects is set to Yes.

Concrete

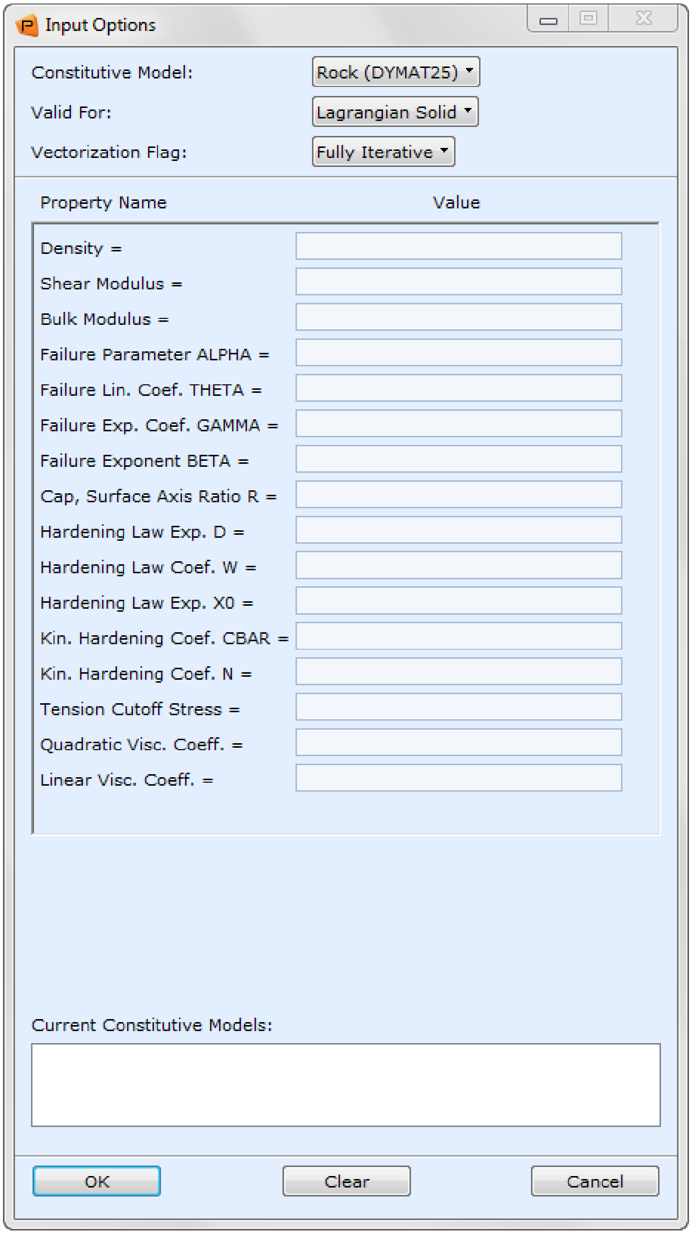
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Concrete option is the selected Constitutive Model on the Input Options form. Use this form to define a concrete using the DYMAT25 model.



Parameter	Description
Valid For	Only Lagrangian Elements.
Vectorization Flag	Vector Options: Fully Iterative Vectorized

Rock

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Rock option is the selected Constitutive Model on the Input Options form. Use this form to define a rock using the DYMAT25 model.

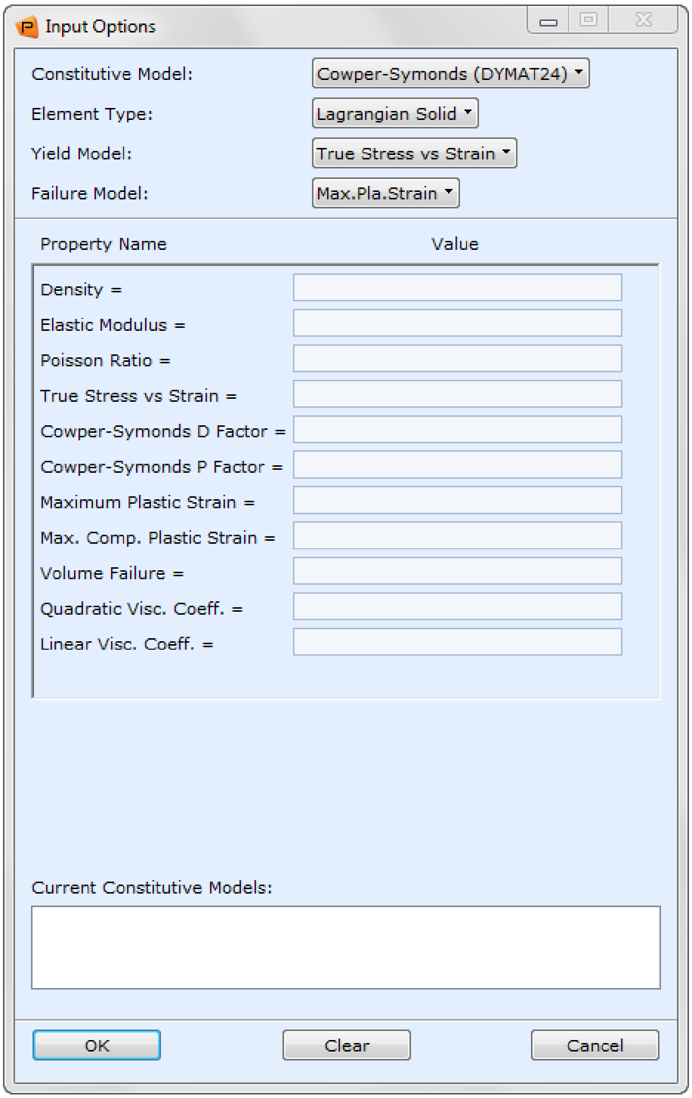


Parameter	Description
Valid For	Only Lagrangian Elements.
Vectorization Flag	Vector Options: Fully Iterative Vectorized

Cowper-Symonds

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Cowper-Symonds (DYMAT24) option is the selected Constitutive Model on the Input Options form. Use this form to define an elastoplastic material using the Cowper-Symonds model using the DYMAT24 description.

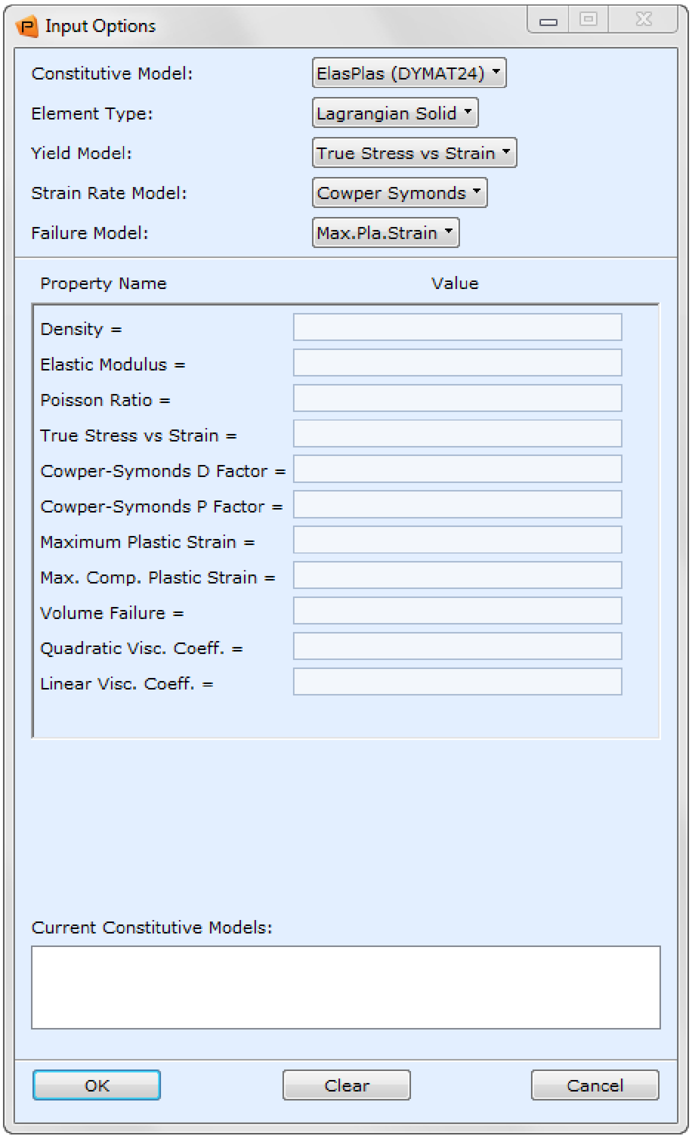
.



Parameter	Description
Element Type	This material is valid for beams, shells, and Lagrangian solids.
Yield Model	Choose between Von Mises, Bilinear, True Stress vs Strain, Engineering Stress vs Strain, True Stress vs Plastic Strain and Plastic Modulus vs Plastic Strain. The appearance of the rest of the form will vary depending on the selection made.
Failure Mode	Choose between Maximum Plastic Strain and None.
True Stress vs. Strain	This entry depends upon the choice of yield model. The Mises model requires definition of a Yield Stress whilst the Bilinear model requires definition of the Yield Stress and Hardening Modulus.

ElastoPlastic

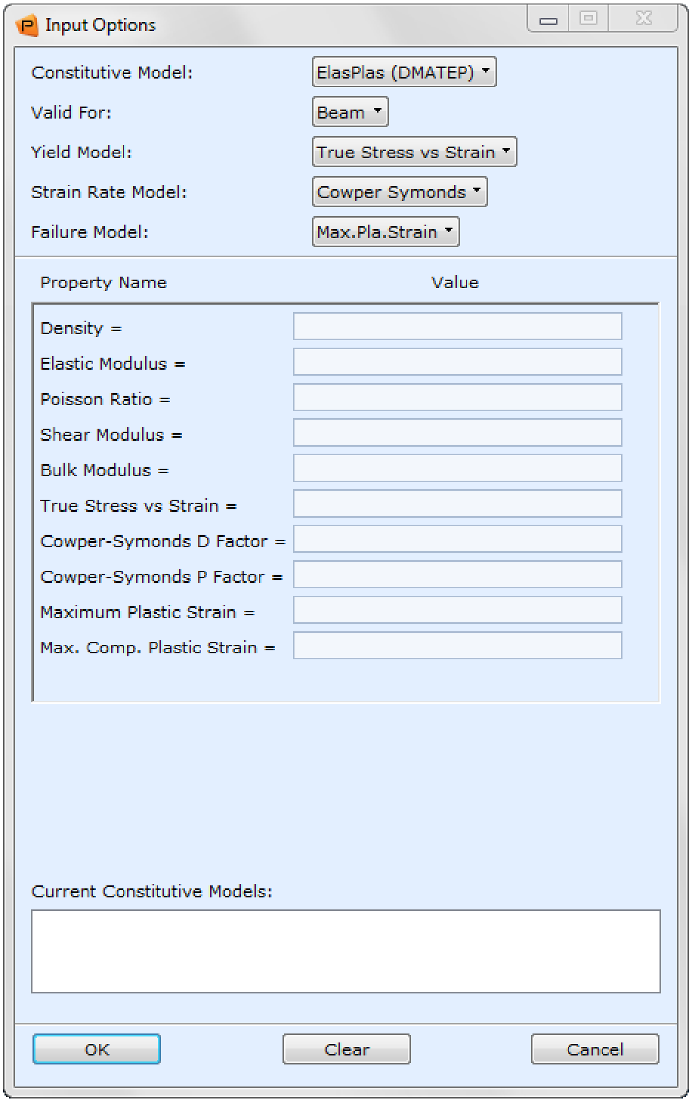
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the ElastoPlastic (DYMAT24) option is the selected Constitutive Model on the Input Options form. Use this form to define the DYMAT24 model.



Parameter	Description
Element Type	This material is valid for beams, shells, and Lagrangian solids.
Yield Model	Choose among Von Mises, Bilinear, True Stress vs Strain, Engineering Stress vs Strain, True Stress vs Plastic Strain, and Plastic Modulus vs Plastic Strain. The appearance of the rest of the form will vary depending on the selection made.
Strain Rate Model	Choose between Cowper-Symonds, Table, and None to select the strain rate model.
Failure Mode	Choose between Maximum Plastic Strain and None.
True Stress vs. Strain	This entry depends upon the choice of Yield Model. This is defined as a strain dependent field, in which Crush or Strain are both treated as strains.
Cowper Symonds D Factor **Cowper Symonds P Factor**	These entries apply when the Cowper-Symonds rate model is selected. The Table definition method uses a strain rate dependent field.
Maximum Plastic Strain **Max. Comp. Plastic Strain**	These entries, if present, depend upon the choice of Failure Model.

ElastoPlastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the ElastoPlastic (DMATEP) option is the selected Constitutive Model on the Input Options form. Use this form to define the DMATEP model.

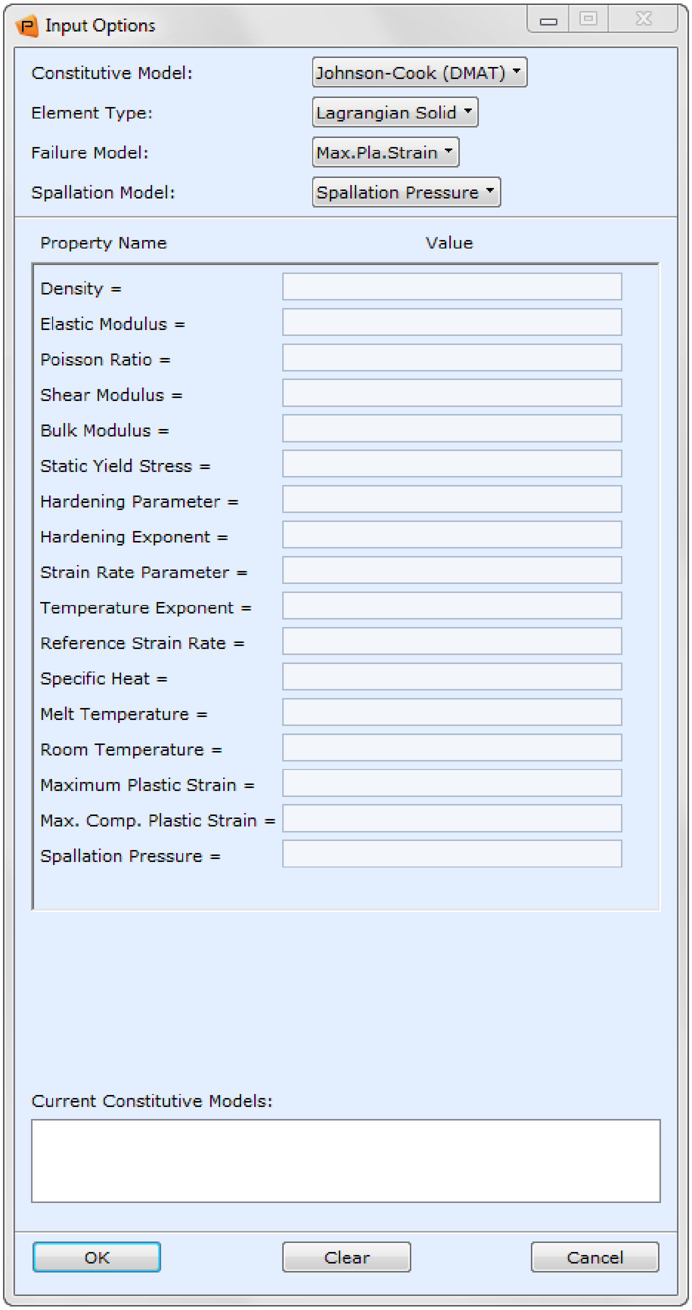


Parameter	Description
Valid For	This material is valid for beams and shells.
Yield Model	Choose among Von Mises, Bilinear, True Stress vs Strain, Engineering Stress vs Strain, True Stress vs Plastic Strain, and Plastic Modulus vs Plastic Strain. If the Valid For option is set to Shell, choices include Johnson-Cook, Rate Power Law, Tanimura-Mimura, and Zerilli-Armstrong. The appearance of the rest of the form will vary depending on the selection made.

Parameter	Description
Strain Rate Model	Choose between Cowper-Symonds, Table, and None to select the strain rate model.
Failure Mode	Choose between Maximum Plastic Strain, User Subroutine, and None. If the subroutine option is used then the name of the subroutine must be EXFAIL.
True Stress vs. Strain	This entry depends upon the choice of Pressure Variation definition. This is defined as a strain dependent field, in which Crush or Strain are both treated as strains.
Cowper Symonds D Factor **Cowper Symonds P Factor**	These entries apply when the Cowper-Symonds rate model is selected. The Table definition method uses a strain rate dependent field.
Max. Comp. Plastic Strain	This entry, if present, depends upon the choice of Cutoff Pressure method.

Johnson-Cook

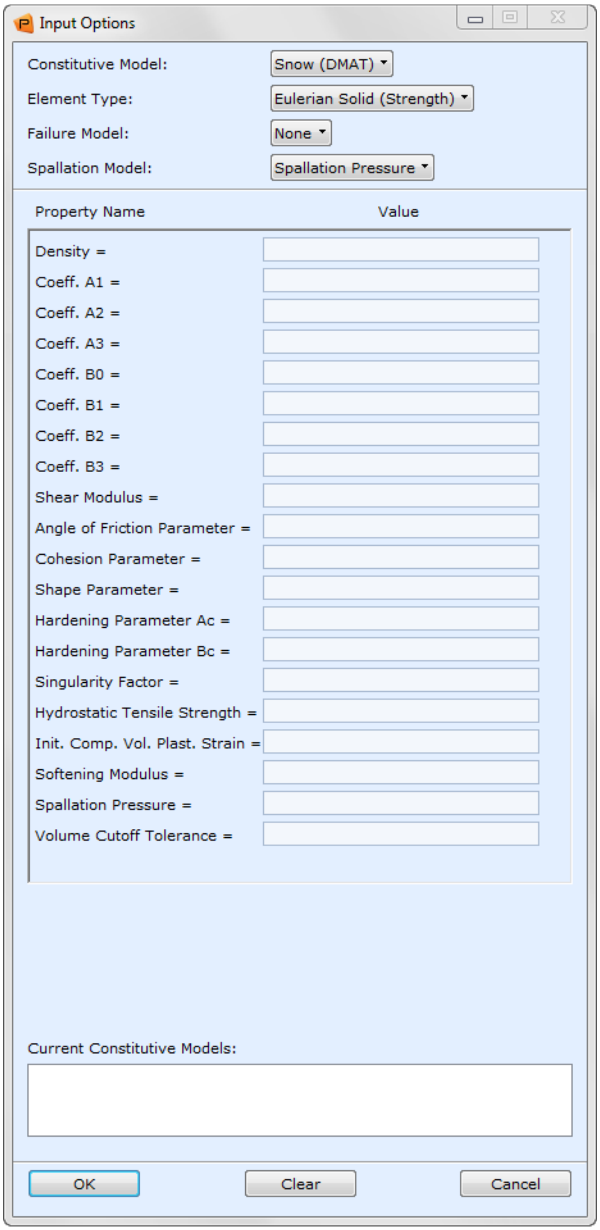
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Johnson-Cook (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define an ElastoPlastic material using the Johnson-Cook model using the DMAT description.


Input Options
Constitutive Model:
Johnson-Cook (DMAT)
Element Type:
Lagrangian Solid
Failure Model:
Max.Pla.Strain
Spallation Model:
Spallation Pressure
Property Name
Value
Density =
Elastic Modulus =
Poisson Ratio =
Shear Modulus =
Bulk Modulus =
Static Yield Stress =
Hardening Parameter =
Hardening Exponent =
Strain Rate Parameter =
Temperature Exponent =
Reference Strain Rate =
Specific Heat =
Melt Temperature =
Room Temperature =
Maximum Plastic Strain =
Max. Comp. Plastic Strain =
Spallation Pressure =
Current Constitutive Models:
OK
Clear
Cancel

Parameter	Description
Element Type	The material is valid for Lagrangian and Eulerian (strength) solids.
Failure Mode	Choose between Maximum Plastic Strain, Maximum Equivalent Stress, Maximum Plastic Strain (and Minimum) Time Step, User Subroutine, and None. The appearance of the rest of the form will vary depending on the selection made. If the subroutine option is used then the name of the subroutine must be EXFAIL.
Spallation Mode	Choose between Spallation Pressure and None.
Static Yield Stress **Hardening Parameter** **Hardening Exponent** **Strain Rate Parameter** **Temperature Exponent** **Reference Strain Rate** **Specific Heat**	These are the Johnson-Cook model parameters. Note: Missing entries depend upon the failure and spallation models selected.

Snow

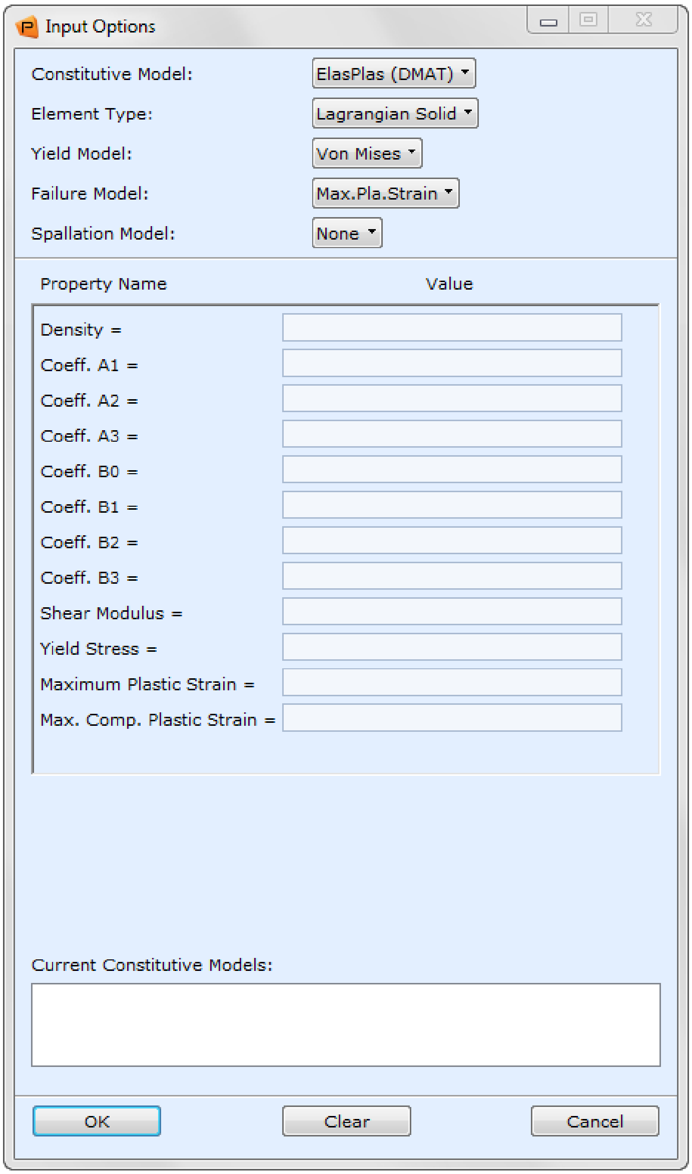
This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Snow (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define the Snow model using the DMAT description.



Parameter	Description
Element Type	This material is valid for Eulerian Solid (Strength) only.
Failure Model	No Failure model.
Spallation Model	Choose between Spallation Pressure and None.

ElastoPlastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the ElastoPlastic (DMAT) option is the selected Constitutive Model on the Input Options form. Use this form to define the DMAT model.

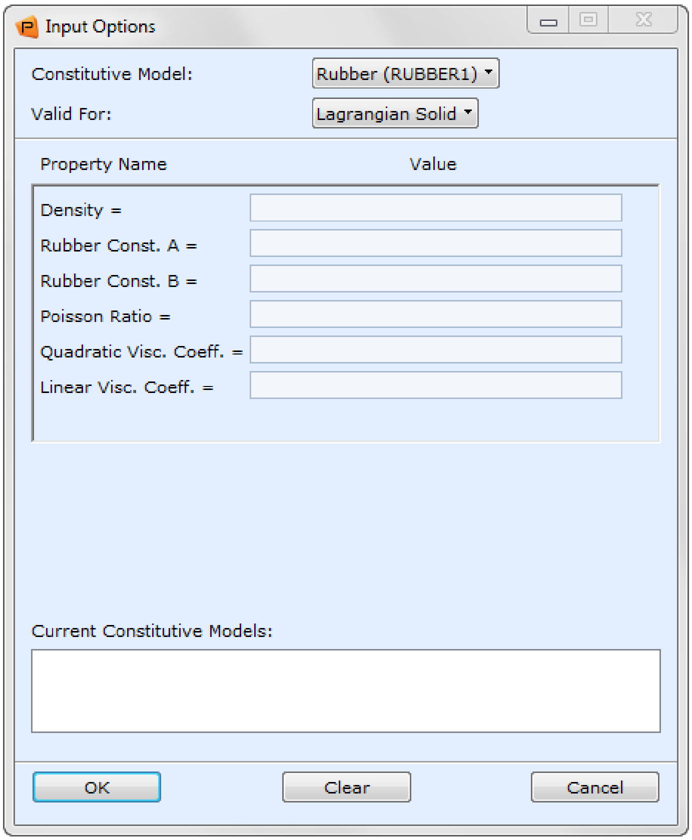


Parameter	Description
Element Type	This material is valid for Lagrangian and Eulerian (strength) solids.
Yield Model	Choose between Von Mises, Johnson-Cook, Rate Power Law, Tanimura-Mimura, Zerilli-Armstrong, Mohr-Coulomb and Multi-Surf Plast. The appearance of the rest of this form will vary depending on the selection made.
Failure Model	Choose between Maximum Plastic Strain, Maximum Equivalent Stress, Maximum Plastic Strain (and Minimum), Time Step, User Subroutine, and None. The appearance of the rest of this form will vary depending on the selection made. If the subroutine option is used then the name of the subroutine must be EXFAIL.
Spallation Model	Choose between Spallation Pressure and None.
Coeff A1	These are the parameters for a Polynomial Equation of State.
Coeff A2	
Coeff A3	
Coeff B0	
Coeff B1	
Coeff B2	
Coeff B3	

Note:	Missing entries depend upon the selections made above.

Rubber

This subordinate form appears when the Input Properties button is selected on the Materials form, when Isotropic is selected on the Material form, and the Rubber (RUBBER1) option is the selected Constitutive Model on the Input Options form. Use this form to define a rubber material using a RUBBER1 description.

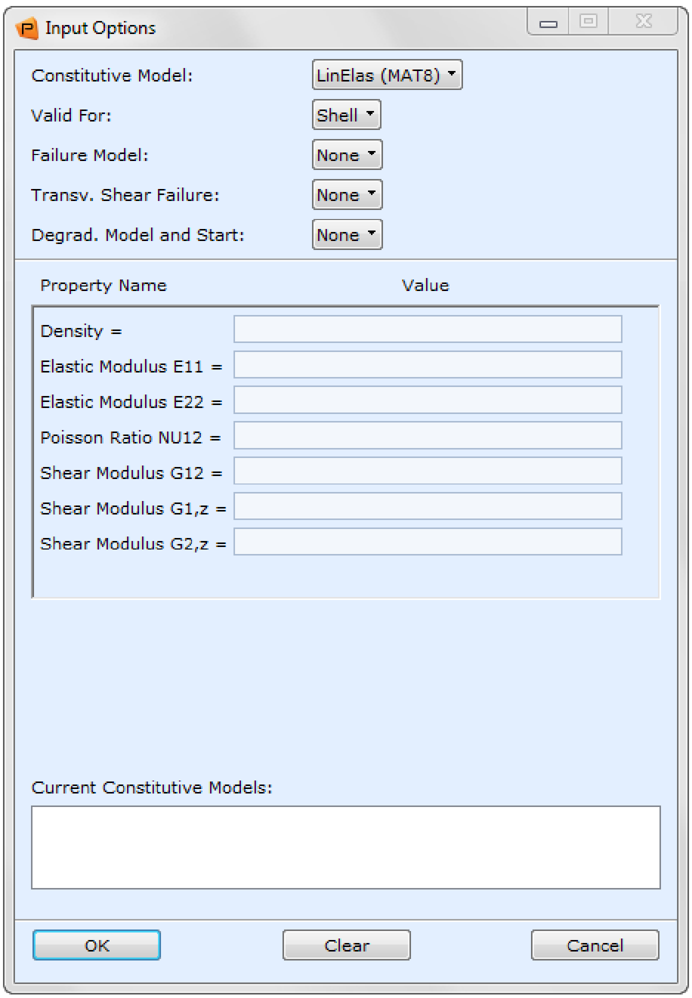


Parameter	Description
Valid For	This model is only applicable for Lagrangian solid elements.

2D Orthotropic

Linear Elastic

This subordinate form appears when the Input Properties button is selected on the Materials form, when 2D Orthotropic is selected on the Material form, and the Linear Elastic (MAT8) option is the Constitutive Model on the Input Options form. Use this form to define a linear elastic, orthotropic material using a MAT8 and MAT8A description.



Parameter	Description
Valid For	This model is only applicable for shell elements.
Failure Mode	Choose between None, Tsai-Hill (1), Tsai-Wu (2), Modified Tsai-Wu (3), Maximum Stress (4), Chang-Chang (5), Hashin (6), Combination and User Subroutine.
Transv. Shear Failure	If Failure option is set to None or User-Subroutine the only option is None. Otherwise, choose between Sublayer or Element.
Degrad. Model and Start	If Failure option is not set to Combination the only option is None. Otherwise choose between the Time Steps/Indiv. Const., Time Steps/All Const., Time/Indiv. Const., Time/All Const., Velocity/Indiv. Const., Velocity/All Const.

Note: Missing entries depend upon the selections made above.

1-For Failure Model databoxes use the number as shown in Failure options.
2-For Prop. Deg databoxes use four digit integers as 1111, 1110, 0111, 0001.

Woven Fabric

This subordinate form appears when the Input Properties button is selected on the Materials form, when 2D Orthotropic is selected on the Material form, and the Woven Fabric(FABRIC) option is the selected Constitutive Model on the Input Options form. Use this form to define a woven fabric using the FABRIC model.

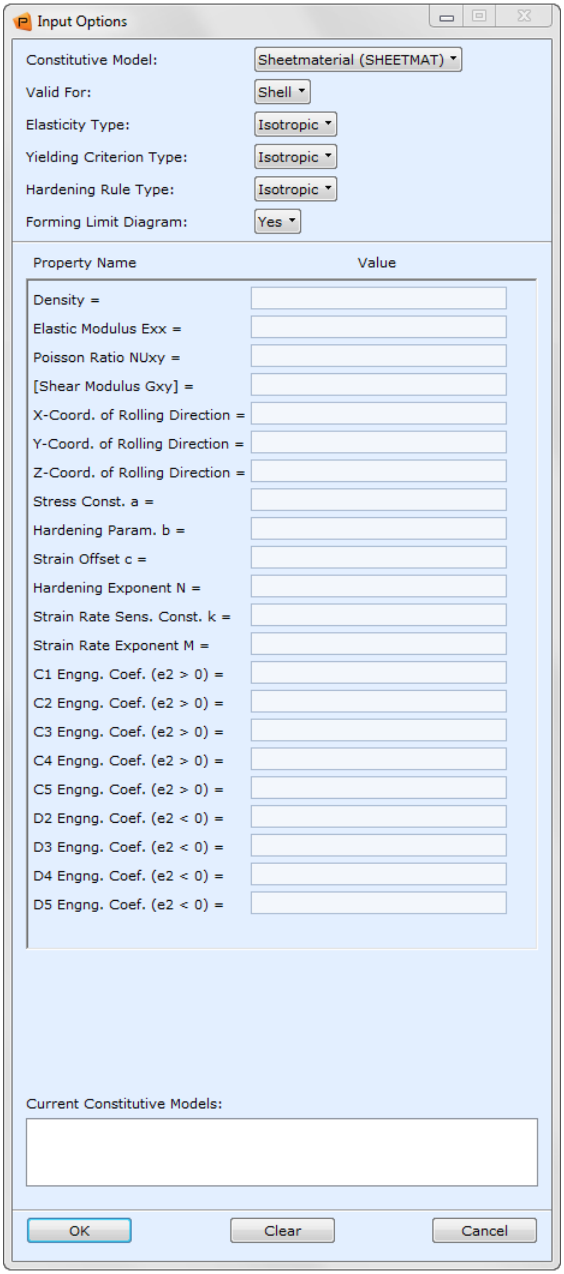

Input Options
Constitutive Model:
Woven Fabric (FABRIC)
Valid For:
Shell
Coating:
Partial Coating
Property Name
Value
Density =
Thickness Perc. of Coating =
Elastic Modulus (Coating) =
Poisson's Ratio (Coating) =
Shear Modulus (Coating) =
Damping Factor (Coating) =
Comp.Str.Scale Fact.(Coating) =
Elas. Mod. Linear Coeff.(Warp) =
Elas. Mod. Quadr. Coeff.(Warp) =
Orientation Angle (Warp) =
X Dir. of the Fabric (Warp) =
Y Dir. of the Fabric (Warp) =
Z Dir. of the Fabric (Warp) =
Elas. Mod. Linear Coeff.(Weft) =
Elas. Mod. Quadr. Coeff.(Weft) =
Orientation Angle (Weft) =
X Dir. of the Fabric (Weft) =
Y Dir. of the Fabric (Weft) =
Z Dir. of the Fabric (Weft) =
Shear Coeff. of Frict.(Fabric) =
Shear Modulus (Fabric) =
Damping Factor (Fabric) =
Comp.Str.Scale Fact. (Fabric) =
Locking Angle 1 (Fabric) =
Locking Angle 2 (Fabric) =
Current Constitutive Models:
OK
Clear
Cancel

Parameter	Description
Valid For	This material is valid for shell elements only.
Coating	Choose between Partial Coating, Only Coating, and No Coating.
Thickness Perc. of Coating	For Partial Coating only.
Elastic Modulus (Coating)	For Partial Coating or Only Coating only.
Poisson’s Ratio (Coating)	
Shear Modulus (Coating)	
Damping Factor (Coating	
Elas. Mod. Linear Coeff. (Warp)	For Partial Coating or No Coating only.

3D Orthotropic

Sheet Metal

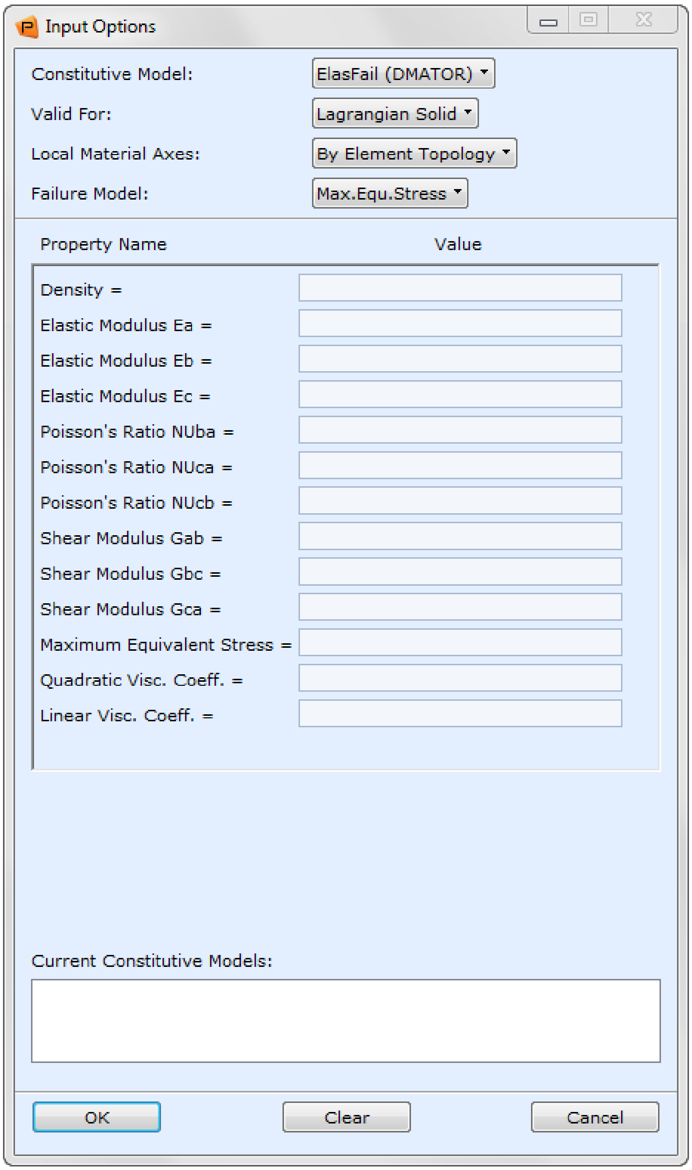
This subordinate form appears when the Input Properties button is selected on the Materials form, when 3D Orthotropic is selected on the Material form, and the Sheet Metal (SEETMAT) option is the selected Constitutive Model on the Input Options form. Use the form on the following page to define a linear elastic, orthotropic plate material using a SHEETMAT description.


Input Options
Constitutive Model:
Sheetmaterial (SHEETMAT)
Valid For:
Shell
Elasticity Type:
Isotropic
Yielding Criterion Type:
Isotropic
Hardening Rule Type:
Isotropic
Forming Limit Diagram:
Yes
Property Name
Value
Density =
Elastic Modulus Exx =
Poisson Ratio NUxy =
[Shear Modulus Gxy] =
X-Coord. of Rolling Direction =
Y-Coord. of Rolling Direction =
Z-Coord. of Rolling Direction =
Stress Const. a =
Hardening Param. b =
Strain Offset c =
Hardening Exponent N =
Strain Rate Sens. Const. k =
Strain Rate Exponent M =
C1 Engng. Coef. (e2 > 0) =
C2 Engng. Coef. (e2 > 0) =
C3 Engng. Coef. (e2 > 0) =
C4 Engng. Coef. (e2 > 0) =
C5 Engng. Coef. (e2 > 0) =
D2 Engng. Coef. (e2 < 0) =
D3 Engng. Coef. (e2 < 0) =
D4 Engng. Coef. (e2 < 0) =
D5 Engng. Coef. (e2 < 0) =
Current Constitutive Models:
OK
Clear
Cancel

Parameter	Description
Valid For	This material is valid for shells only.
Elasticity Type	Choose Isotropic and Planar Isotropic.
Yielding Criteria Type	Choose Isotropic Normal Anisotropic and Planar Anistropic.
Hardening Rule Type	Choose Isotropic and Normal Anisotropic.
Forming Limit Diagram	Select Yes to request output of the forming limit diagram data file.
Elastic Module Exx	These entries define the material parameters. The entries depend upon selections made above.
Poisson Ratio NUxy	
[Shear Modulus Gxy]	
X-Coord of Rolling Direction	
Y-Coord of Rolling Direction	
Z-Coord of Rolling Direction	
Stress Constant a	
Hardening Param. b	
Strain Offset c	
Hardening Exponent N	
Strain Rate Sens. Const. k	
Strain Rate Exponent M	

Elastic Failure

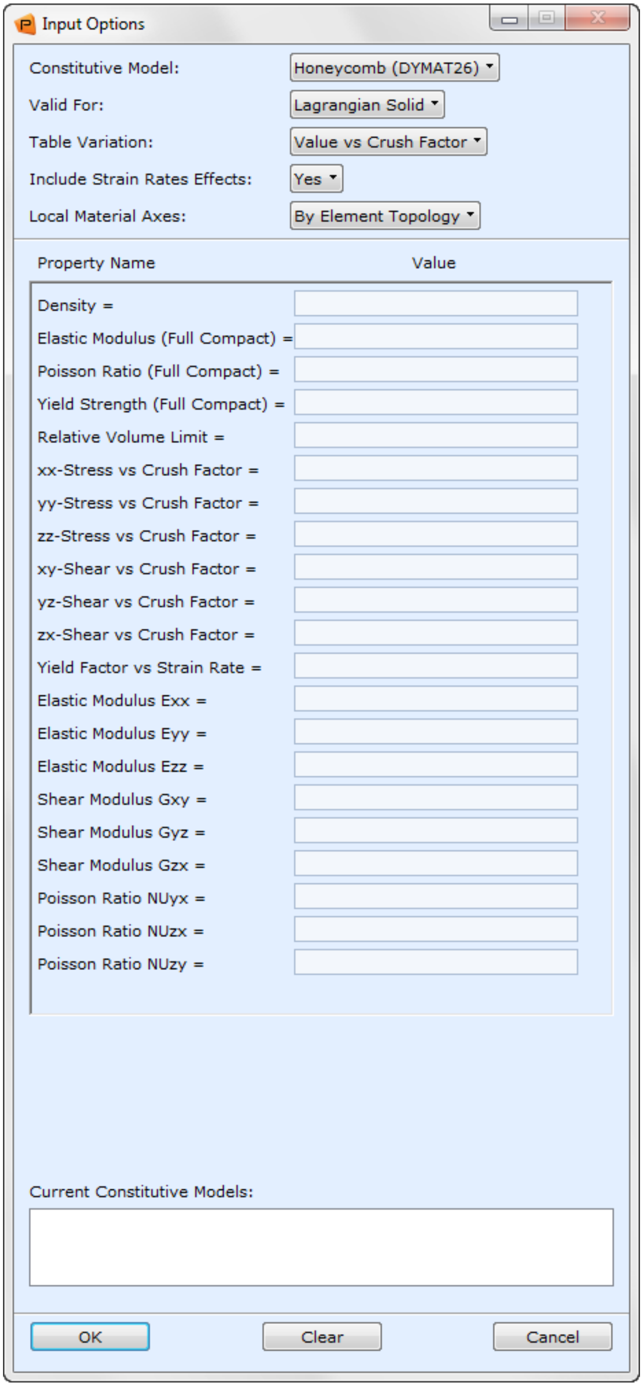
This subordinate form appears when the Input Properties button is selected on the Materials form, when 3D Orthotropic is selected on the Material form, and the Elastic Failure (DMATOR) option is the selected Constitutive Model on the Input Options form. Use this form to define an orthotropic elastic material, with failure, using the DMATOR description.



Parameter	Description
Valid For	This material is only valid for Lagrangian solids.
Local Material Axes	Choose between: By Two Vectors, By Element Topology, By Element Material and By Element Property. The appearance of the rest of the form will vary depending on the selection made.
Failure Mode	Choose between: Max. Equivalent Stress, Pressure, Maximum Equivalent Stress, and Minimum Time Step, User Subroutine, Extended User Subroutine, and None. If the subroutine option is used then the name of the subroutine must be EXFAIL.
Elastic Modulus Ea	These are the parameters defining an orthotropic elastic material.
Elastic Modulus Eb	
Elastic Modulus Ec	
Poisson's Ratio NUba	
Poisson's Ratio NUca	
Poisson's Ratio NUcb	
Shear Modulus Gab	
Shear modulus Gbc	
Shear Modulus Gca	
Maximum Equivalent Stress	The failure parameter will depend on the Failure Model selected above.

Honeycomb

This subordinate form appears when the Input Properties button is selected on the Materials form, when 3D Orthotropic is selected on the Material form, and the Honeycomb (DYMAT26) option is the selected Constitutive Model on the Input Options form. Use this form to define a honeycomb using the DYMAT26 model.

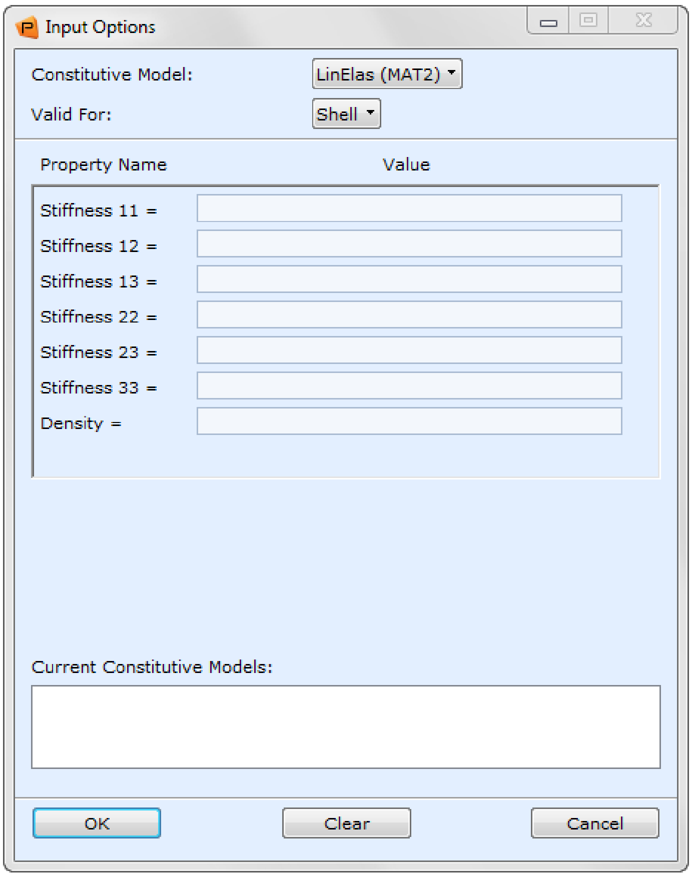

Input Options
Constitutive Model:
Honeycomb (DYMAT26)
Valid For:
Lagrangian Solid
Table Variation:
Value vs Crush Factor
Include Strain Rates Effects:
Yes
Local Material Axes:
By Element Topology
Property Name
Value
Density =
Elastic Modulus (Full Compact) =
Poisson Ratio (Full Compact) =
Yield Strength (Full Compact) =
Relative Volume Limit =
xx-Stress vs Crush Factor =
yy-Stress vs Crush Factor =
zz-Stress vs Crush Factor =
xy-Shear vs Crush Factor =
yz-Shear vs Crush Factor =
zx-Shear vs Crush Factor =
Yield Factor vs Strain Rate =
Elastic Modulus Exx =
Elastic Modulus Eyy =
Elastic Modulus Ezz =
Shear Modulus Gxy =
Shear Modulus Gyz =
Shear Modulus Gzx =
Poisson Ratio NUyx =
Poisson Ratio NUzx =
Poisson Ratio NUzy =
Current Constitutive Models:
OK
Clear
Cancel

Parameter	Description
Valid For	This material is valid for Lagrangian solids only.
Table Variation	Choose between Value vs Crush Factor and Value vs Relative Volume.
Include Strain Rate Effects	Choose to define a yield factor as a function of strain rate.
Local Material Axes	Choose between: By Two Vectors and By Element Topology.
xx-Stress vs. Crush Factor	These entries depend upon the choice of Table Variation method. They are defined as strain dependent fields, in which Volume or Relative Volume are both treated as strains.
yy-Stress vs. Crush Factor	
zz- Stress vs. Crush Factor	
xy-Shear vs. Crush Factor	
yz-Shear vs. Crush Factor	
zx-Shear vs. Crush Factor	

2D Anisotropic

Linear Elastic

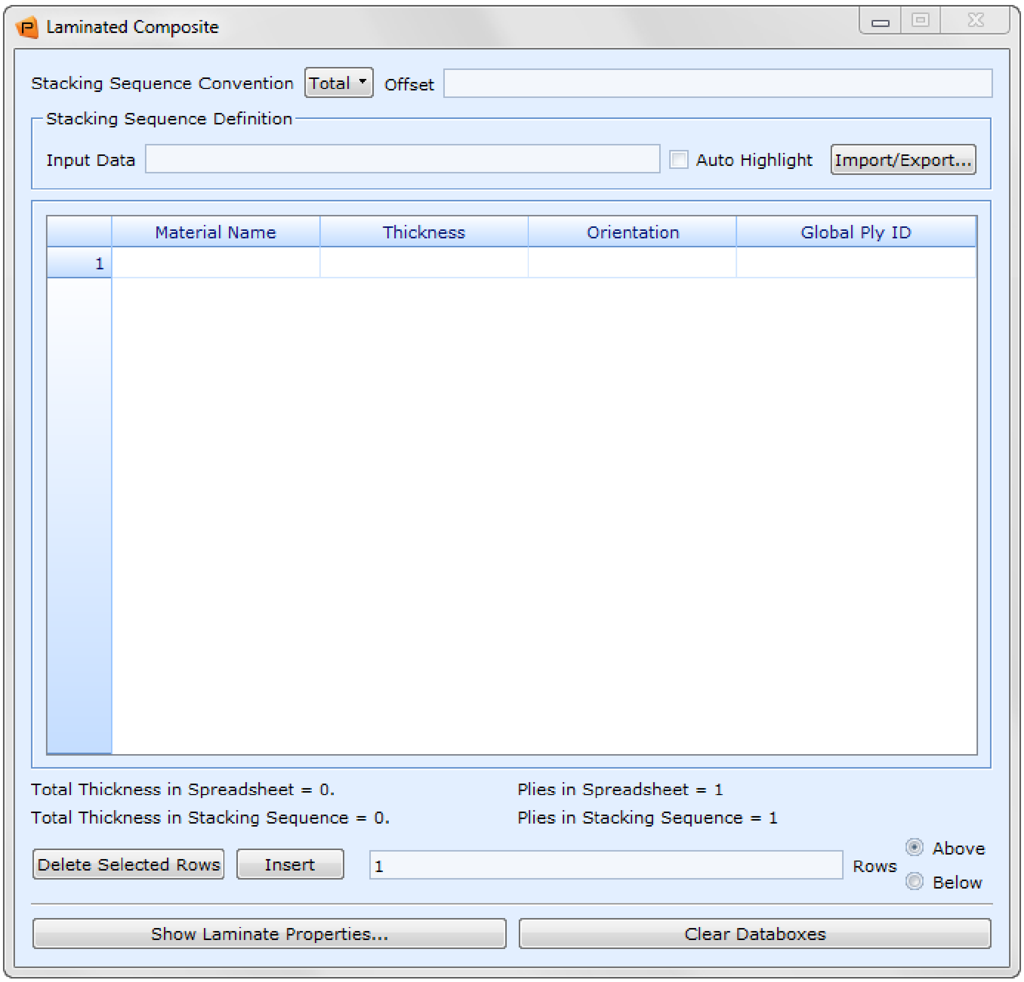
This subordinate form appears when the Input Properties button is selected on the Materials form, when 2D Anisotropic is selected on the Material form, and the Linear Elastic (MAT2) option is the Constitutive Model on the Input Options form. Use this form to define a linear elastic, anisotropic material using a MAT2 description.



Parameter	Description
Valid For	This model is only applicable for shell elements.

Composite and Laminate

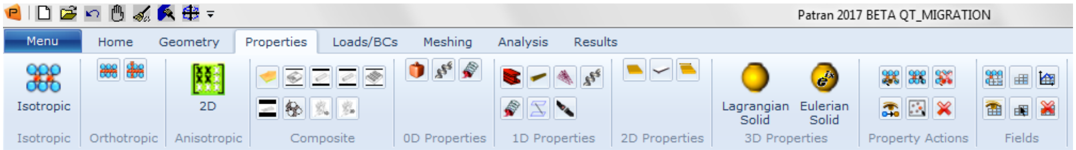
This subordinate form appears when the Input Properties button is selected on the Materials form, and when Composite and Laminate are selected on the Material form. Use this form to define a linear elastic laminated composite material.



Element Properties

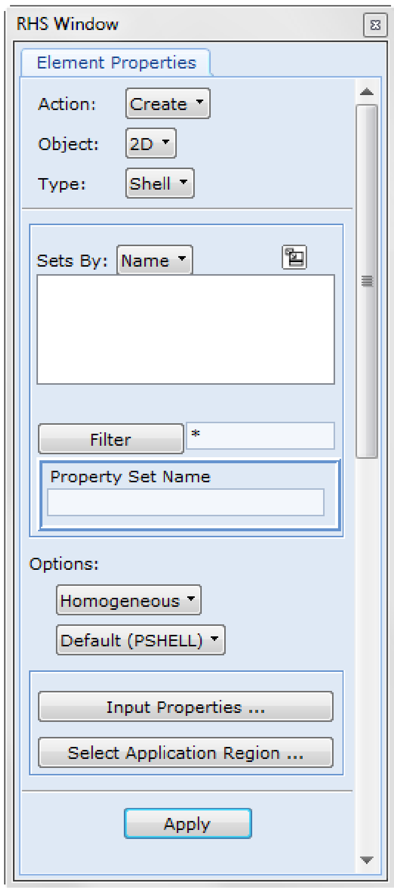
The Element Properties form appears when the Element Properties toggle, located on the Patran main form, is chosen.There are several option menus available when creating element properties. The selections made on the Element Properties menu will determine which element property form appears, and ultimately, which Dytran element will be created.

The following pages give an introduction to the Element Properties form, and details of all the element property definitions supported by the Patran Dytran Preference.



Element Properties Form

This form appears when Element Properties is selected on the main form. There are *four option menus* on this form, each will determine which Dytran element type will be created, and which property forms will appear. The individual property forms are documented later in this section. For a full description of this form, see Element Properties Forms (p. 63) in the *Patran Reference Manual*.



Parameter	Description
Object	Use this option menu to define the element's dimension. The options are: 0D (point elements) 1D (bar elements) 2D (tri and quad elements) 3D (tet, wedge, and hex elements)
Type	This option menu depends on the selection made in the Object option menu. Use this menu to define the general type of element, such as: Shell versus Membrane.
Options	These option menus may or may not be present and their contents depend heavily on the selections made for Dimension and Type. See Page 69 for more help. Note that special attention has been paid to defining the actual Bulk Data entry that will result from the property set being defined.

The following table outlines the option menus when Analysis Type is set to Structural.

Table 2-3 Element Properties

Degree	Type	Option 1
0D	■ Mass	MASS (CONM2) Scalar GrSpr. (PELAS) NonLinear GrSpr.(PELAS1) User Def.Sc.GrSpr. (PELASEX) Scalar Gr.Damp. (PDAMP)
1D	■ Beam	Simple Beam (PBEAM) Hughes-Liu Beam (PBEAM1) Bely-Schwer Beam (PBEAM1) Predefined HL Beam (PBEAML) Lumped Section (PBEAMP)

Degree	Type	Option 1
	■ Rod	Rod (PROD)
	■ Spring	Scalar Spring (PELAS) User Defined Scalar (PELASEX) Linear Spring (PSPR) NonLinear Spring (PSPR1) User Defined Spring (PSPREX) NonLinear Spring (PELAS1)
	■ Damper	Scalar Damper (PDAMP) Linear Damper (PVISC) NonLinear Damping (PVISC1) User Defined Damper (PVISCEX)
	■ Bar	Bar (PBAR) Bar (PBEAM)
	■ Seat Belt	Belt (PBELT)
	■ Spotweld	Simple (PWELD) Rupture (PWELD1) Delamination (PWELD2)
2D	■ Shell	Default (PSHELL/HGSUPPR) Default (PSHELL1/HGSUPPR) BLT (PSHELL1/HGSUPPR) KeyHoff (PSHELL1/HGSUPPR) HughesLiu (PSHELL1/HGSUPPR) C0-Triangle (C0TRIA) Laminate (PCOMP/PCOMPA/HGSUPPR) Equivalent Section (PSHELL1/HGSUPPR)
	■ Membrane	Membrane (PSHELL1)

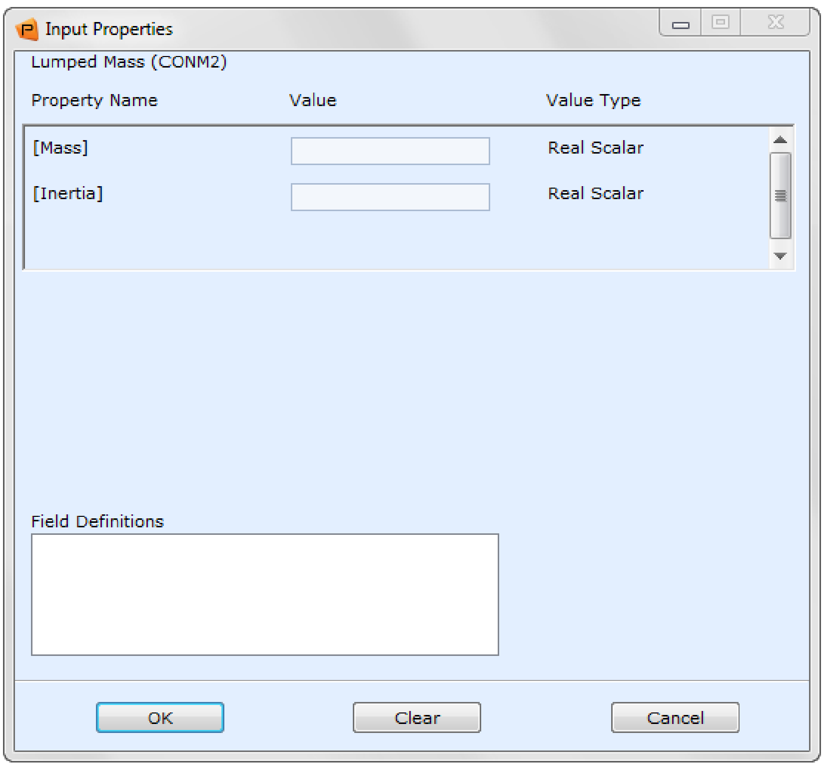
Degree	Type	Option 1
	■ Dummy Shell	Dummy property
3D	■ Lagrangian Solid	Lagrangian Solid (PSOLID/HGSUPPR)
	■ Eulerian Solid	Hydro (PEULER) Strength (PEULER) MM/Hydro (PEULER) MM/Strength (PEULER) Hydro (PEULER1) Strength (PEULER1) MM/Hydro (PEULER1) MM/Strength (PEULER1)

0D Mass

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	0D	0Mass (CONM2)	Point

Use this form to create a CONM2 Bulk Data entry. This defines a lumped mass at a geometric point of the structural model.



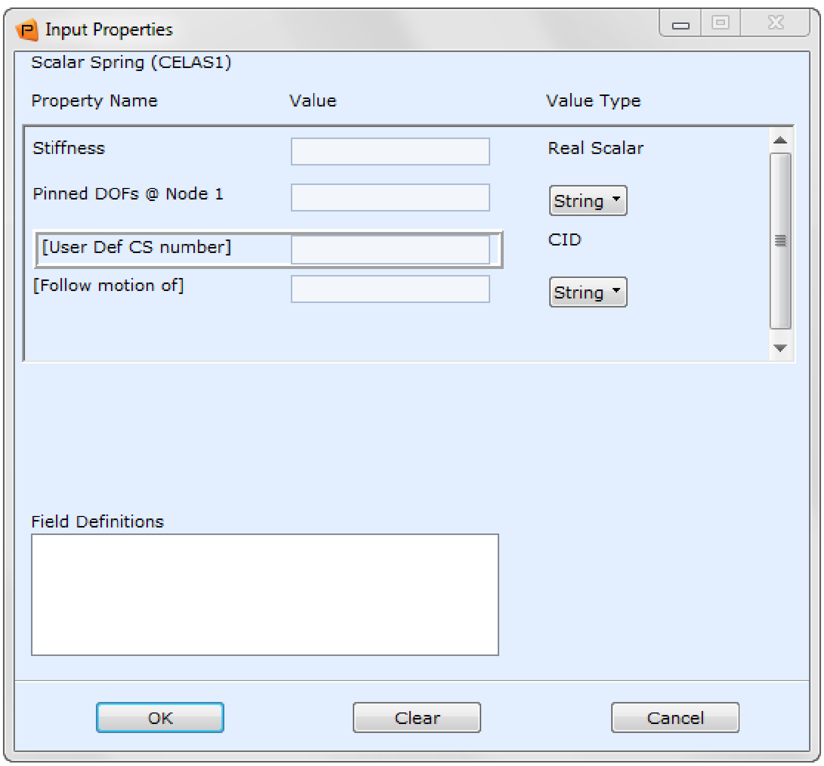
Parameter	Description
Mass Inertia	Defines the mass and inertia values assigned to the point. These properties are both optional.

Grounded Spring

One of three subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	0D	Grounded Spring	Scalar GrSpr.(PELAS + CELAS1) NonLinear GrSpr (PELAS1 + CELAS1) User Def. Sc.GrSpr (PELASEX + CELAS1)	Bar/2

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form.



Parameter	Description
Stiffness	Defines the relationship between the spring deflection and the stresses within the spring.
Pinned DOFs @ Node 1	Defines the orientation of the spring by allowing one degree of freedom at the first node to be pinned.

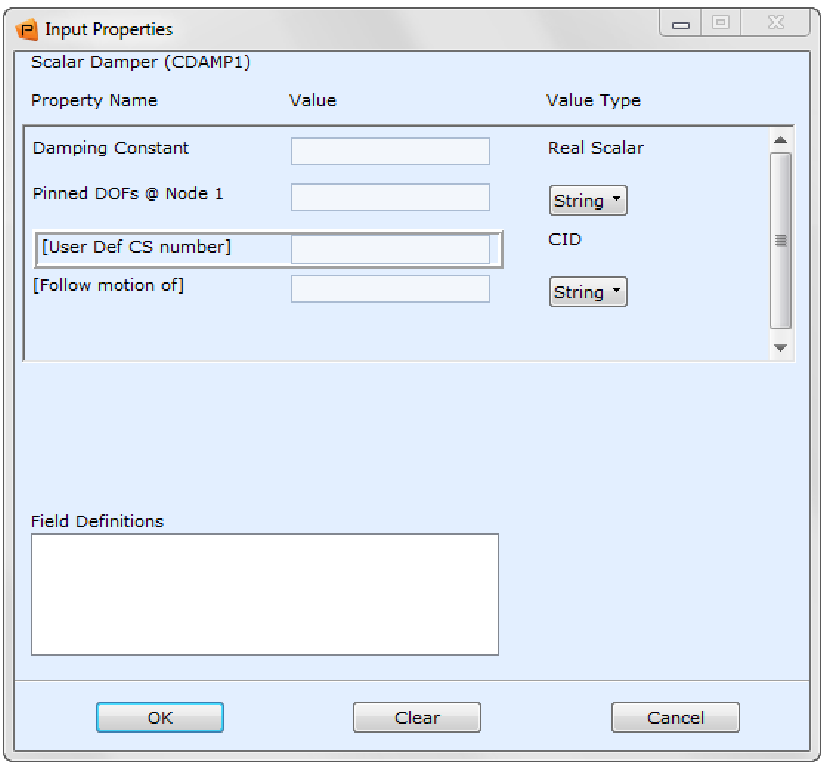
Parameter	Description
[**User Def CS Number**]	Number of a User Defined Coordinate system, used in conjunction with follower option below. This property is optional.
[**Follow motion of**]	This optional entry is used when the motion is to be constrained. Follow the motion in a user defined coordinate system or follow one of the grid points.

Grounded Damper

This subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	0D	Grounded Damper	Scalar Gr.Damp. (PDAMP + CDAMP1)	Bar/2

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form.



Parameter	Description
Damping Constant	Defines the relationship between the spring deflection and the stresses within the string.
Pinned DOFs @ Node 1	Defines the orientation of the spring by allowing one degree of freedom at the first node to be pinned.

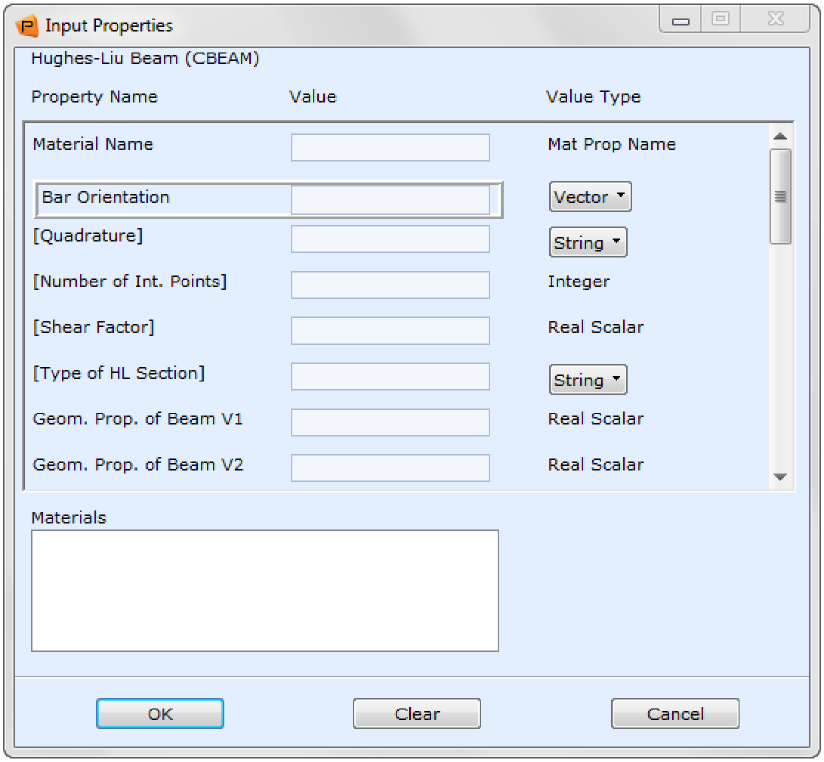
Parameter	Description
[**User Def CS Number**]	Number of a User Defined Coordinate System, used in conjunction with follower option below. This property is optional.
[**Follow motion of**]	This optional entry is used when the motion is to be constrained. Follow the motion in a user defined coordinate system or follow one of the grid points.

Beam

One of five subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Option	Topologies
Create	1D	Simple Beam (PBEAM + CBEAM)	Bar/2
		Hughes-Liu Beam (PBEAM1 + CBEAM)	
		Belytschko-Schwer (PBEAM1 + CBEAM)	
		Predefined HL Beam (PBEAML + CBEAM)	
		Lumped Section (PBCOMP + CBEAM)	

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form. The most general description of a beam is provided by the Hughes-Liu, which permits definition of standard sections.



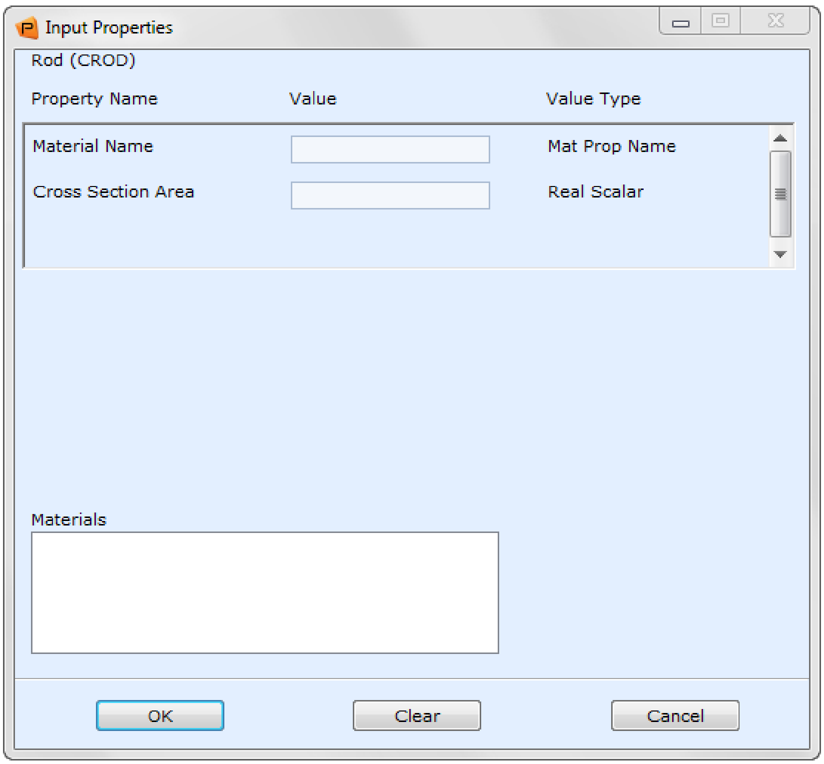
Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database is displayed when data is entered. Either select from the list using the mouse or type in the name. This property is required.
Bar Orientation	Defines the local element coordinate system to be used for any cross sectional properties. Define a vector or give the ID of the orientation node, using the node select tool.
Quadrature	Defines the integration method, which may be either Gauss or Lobatto. Gauss is default.
Shear Factor	Defines the shear factor, recommended value is 5/6. This property is optional.
Type of HL Section	Defines the section type, which may be Rectangular, Tubular, Trapezium, T Section, L Section, U Section, Z Section, or I Section. See the Dytran User's Manual for the definition of the 4 geometric parameters that define the section. Scroll down to enter that data.

Rod

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	1D	Rod (PROD + CROD)	Bar/2

Use this form to create PROD and CROD Bulk Data entries. This defines a tension-compression-torsion element of the structural model.



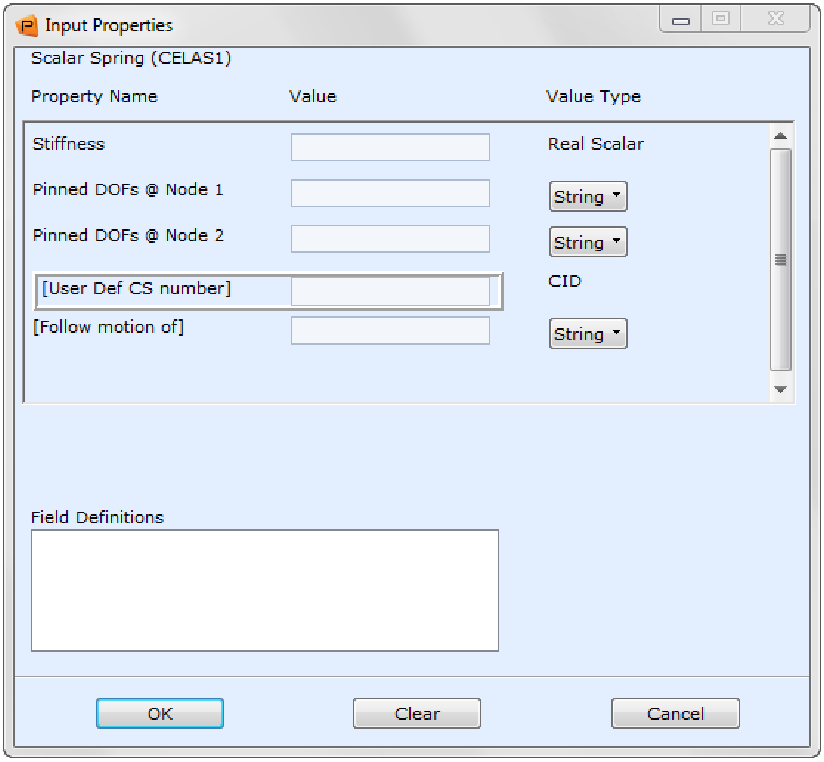
Parameter	Description
Material Name	Defines the material to be used. A list of materials currently in the database is displayed when data is entered. Either select from the list using the mouse or type in the name. This property is required.
Cross Section Area	Defines the cross-sectional area of the element. This value can either be a real value, or a reference to an existing field definition. This property is required.

Spring

One of six subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	1D	Spring	Scalar Spring (PELAS + CELAS1) User Defined Scalar (PELASEX + CELAS1) Linear Spring (PSPR + CSPR) NonLinear Spring (PSPR1 + CSPR) User Defined Spring (PSPREX + CSPR) NonLinear Spring (PELAS1 + CELAS1)	Bar/2

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form.



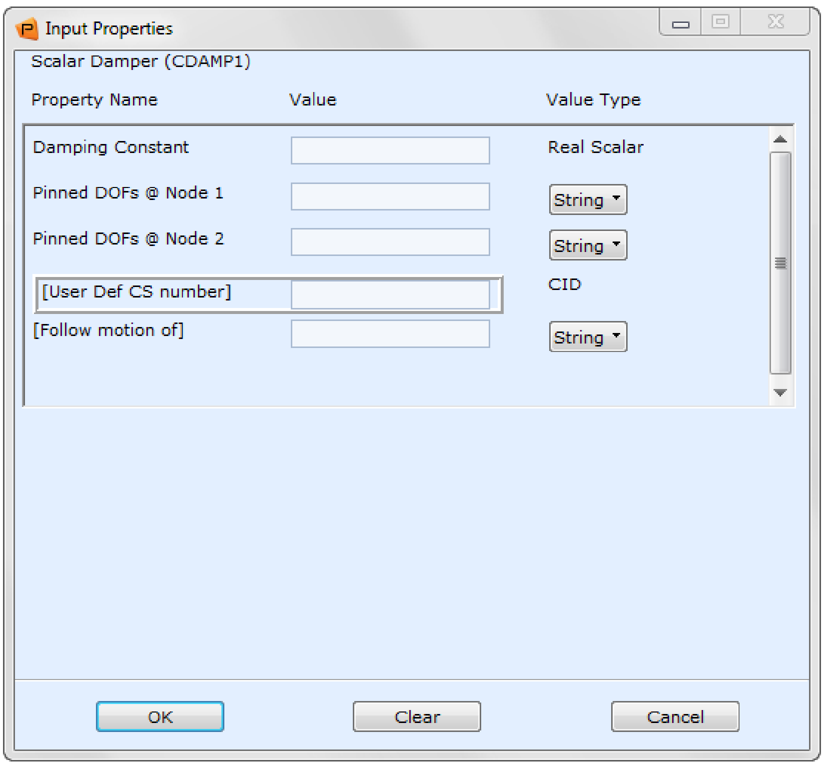
Parameter	Description
Stiffness	Defines the relationship between spring deflection and the stresses within the spring.
Pinned DOFs @ Node 1	Defines the orientation of the spring by allowing one degree of freedom at the first node to be pinned.
Pinned DOFs @ Node 2	Defines the orientation of the spring by allowing one degree of freedom at the second node to be pinned.
[User Def CS Number]	Number of a User Defined Coordinate system, used in conjunction with follower option below. This property is optional.
[Follow motion of]	This optional entry is used when the motion is to be constrained. Follow the motion in a user defined coordinate system or follow one of the grid points.

Damper

One of 4 subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	1D	Damper	Scalar Damper (PDAMP + CDAMP1) Linear Damper (PVISC + CVISC) NonLinear Damper (PVISC1 + CVISC) User Defined Damper (PVISCEX + CVISC)	Bar/2

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form.



Parameter	Description
Stiffness	Defines the relationship between spring deflection and the stresses within the spring.
Pinned DOFs @ Node 1	Defines the orientation of the spring by allowing one degree of freedom at the first node to be pinned.

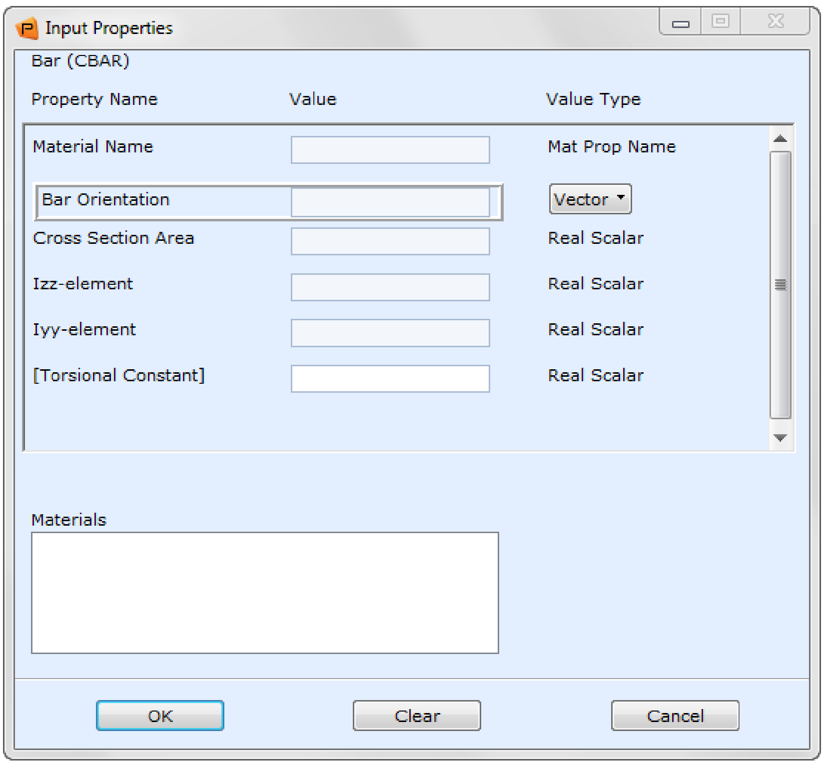
Parameter	Description
Pinned DOFs @ Node 2	Defines the orientation of the spring by allowing one degree of freedom at the second node to be pinned.
[**User Def CS Number**]	Number of a User Defined Coordinate system, used in conjunction with follower option below. This property is optional.
[**Follow motion of**]	This optional entry is used when the motion is to be constrained. Follow the motion in a user defined coordinate system or follow one of the grid points.

Bar

One of two subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	1D	Bar	Bar (PBAR + CBAR) Bar (PBEAM + CBAR)	Bar/2

Use this form to create the Bulk Data entries indicated above.



Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database, is displayed when data is entered. Either select from the list using the mouse or type in the name. This property is required.
Bar Orientation	Defines the local element coordinate system to be used for any cross sectional properties. Define a vector or give the ID of the orientation node, using the node selector tool.

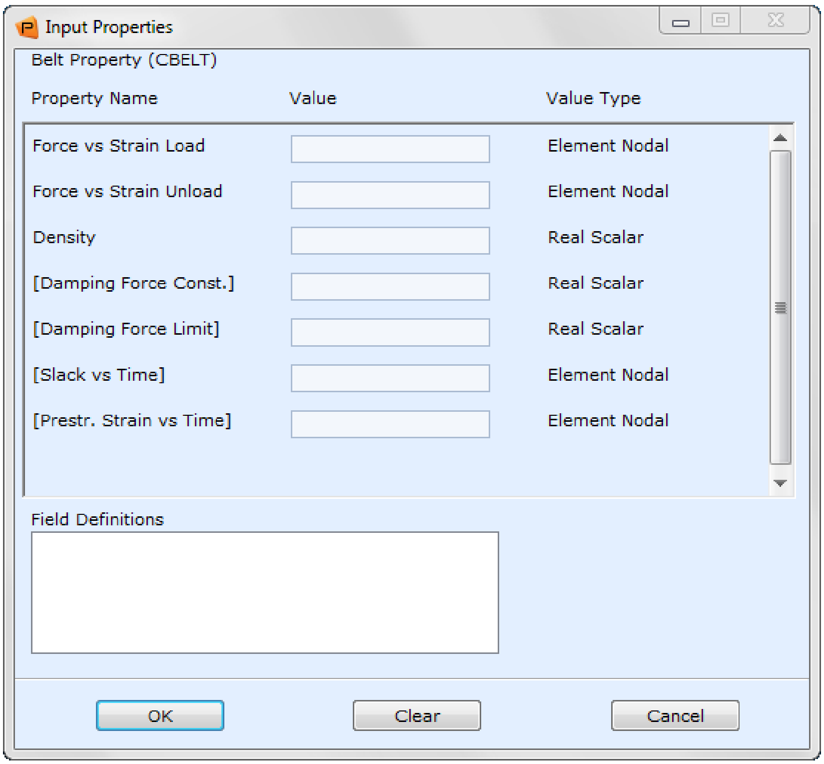
Parameter	Description
Izz Element	Defines the inertial properties of the beam in the local beam coordinate system.
Iyy Element	
[**Torsional Constant**]	Defines Torsional Constant. This property is optional.

Seat Belt

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	1D	SeatBelt (PBELT + CBELT)	Bar/2

Use this form to create the Bulk Data entries indicated above.

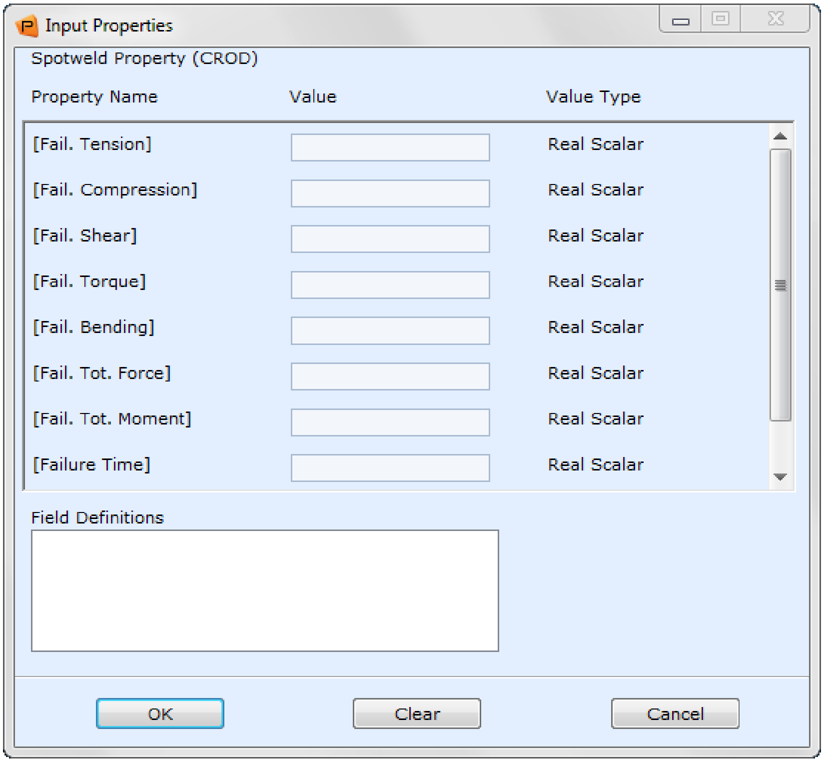


Spotweld

One of five subordinate forms appear when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	1D	Simple Rod (PWELD + CROD) Rupture Rod (PWELD1 + CROD) Delamination Rod (PWELD2 + CROD) Simple Bar (PWELD + CBAR) Rupture Bar (PWELD1 + CBAR)	Bar/2

Use this form to create the Bulk Data entries indicated above. The contents of the form will depend upon the selection made on the element properties form.

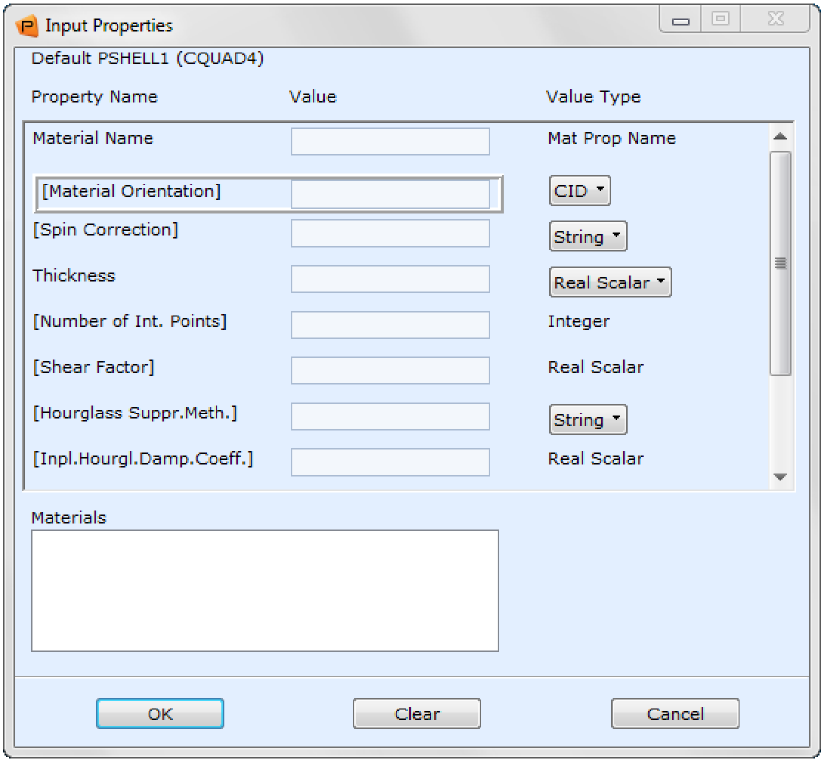


Shell

One of eight subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	2D	Shell	Default (PSHELL + CQUAD4) Default (PSHELL1 + CQUAD4) BLT (PSHELL1 + CQUAD4) KeyHoff (PSHELL1 + CQUAD4) Hughes Liu (PSHELL1 + CQUAD4) Co-Triangle (C0-TRI + CTRI3) Laminate (PCOMP/PCOMPA + CQUAD4) Equivalent Section (PSHELL1 + CQUAD4)	Tri/3, Quad/4

Use this form to create the Bulk Data entries indicated above.



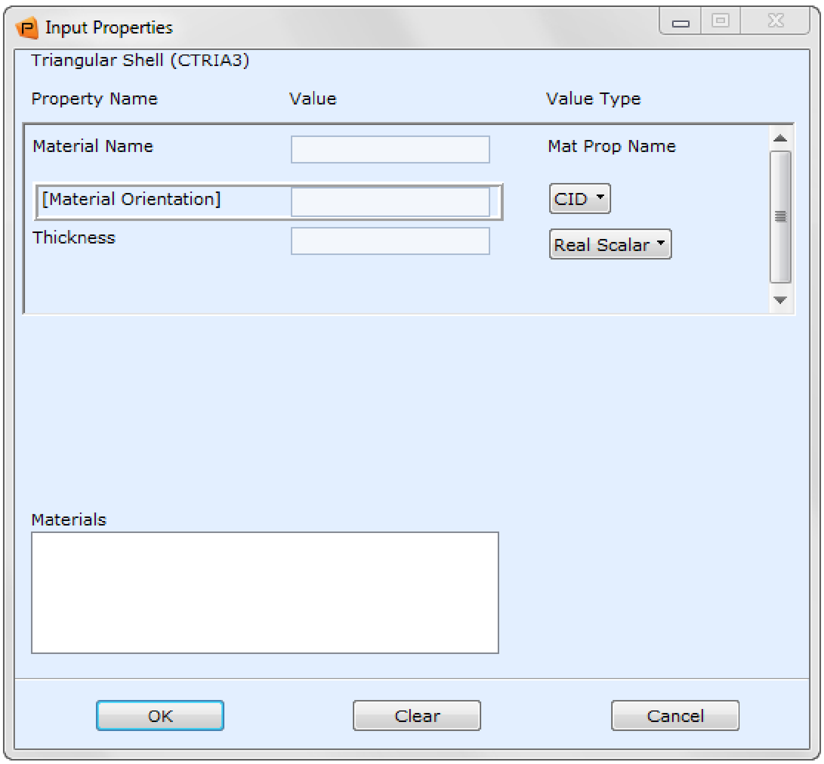
Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database, is displayed when data is entered. Either select one from the list using the mouse or type in the name. This property is required.
Material Orientation	Defines the material orientation. If the coordinate option (CID) is selected, then use the select tool to pick a coordinate system.
Spin Correction	Defines if the Spin Correction is applied. This property is optional.
Thickness	Defines the thickness which will be uniform over each element. This value can be a real value or a reference to an existing field definition.
Number of Int. points	Defines the number of integration points through the thickness of the shell. This property is optional.
Hourglass Suppr. Meth.	Defines hourglass method and coefficients. These properties are optional.
Inpl. Hourgl. Damp. Coeff	

Membrane

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	2D	Membrane (PSHELL1 + CTRIA3)	Tria/3

Use this form to create the Bulk Data entries indicated above.



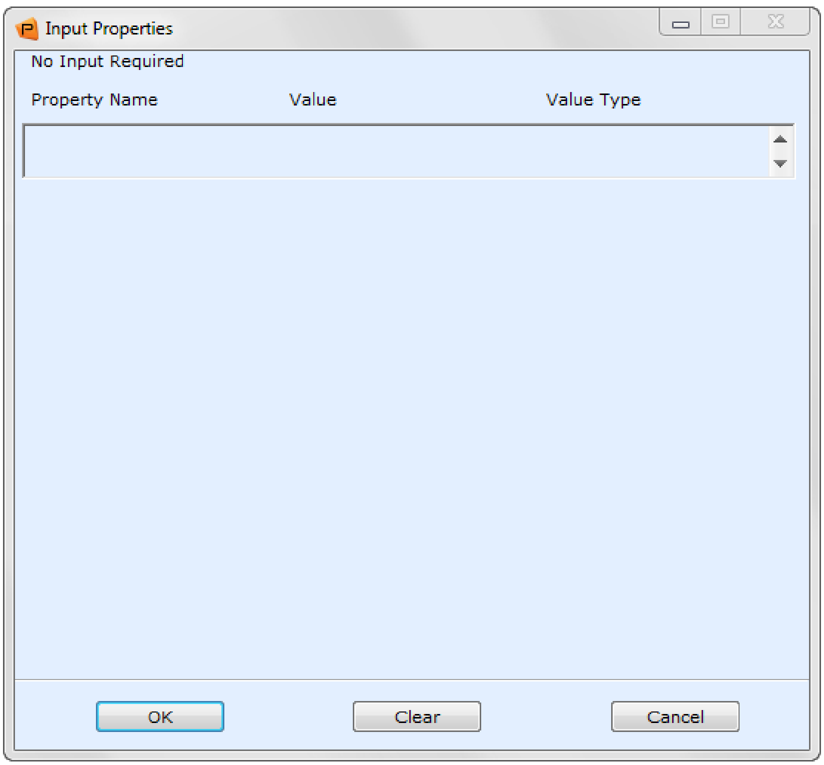
Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database is displayed when data is entered. Either select one from the list using the mouse or type in the name. This property is required.
[Material Orientation]	Defines the material orientation. If the coordinate option (CID) is selected then use the select tool to pick a coordinate system.
Thickness	Defines the thickness which will be uniform over each element. This value can either be a real value or a reference to existing field definitions.

Dummy Shell

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	2D	Dummy Shell (CQUAD4)	Quad/4

No data is required for the dummy shells.

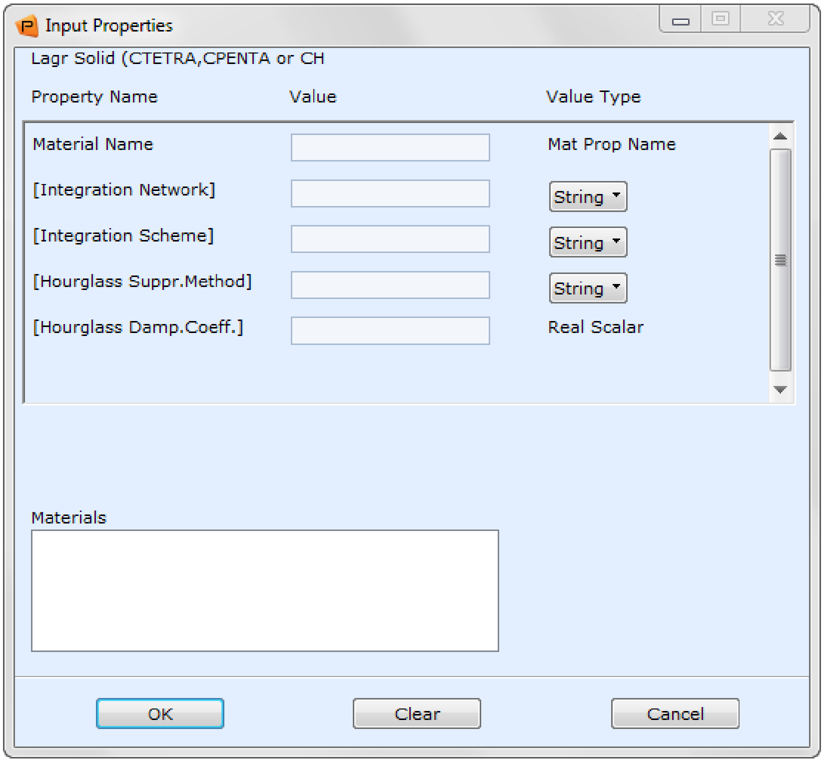


Lagrangian Solid

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Topologies
Create	3D	Lagrangian Solid (PSOLID + CTETRA/CPENTA/CHEXA)	Tet/4, PentA/6, Hex/8, Wedge

Use this form to create the Bulk Data entries indicated above.



Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database is displayed when data is entered. Either select one from the list using the mouse or type in the name. This property is required.
Integration Network	Defines the integration network, which may be One or Two.
Integration Scheme	Defines the integration scheme, which may be Reduced or Full.
Hourglass Suppr. Method **Hourglass Damp. Coeff.**	Define hourglass method and coefficient. These properties are optional.

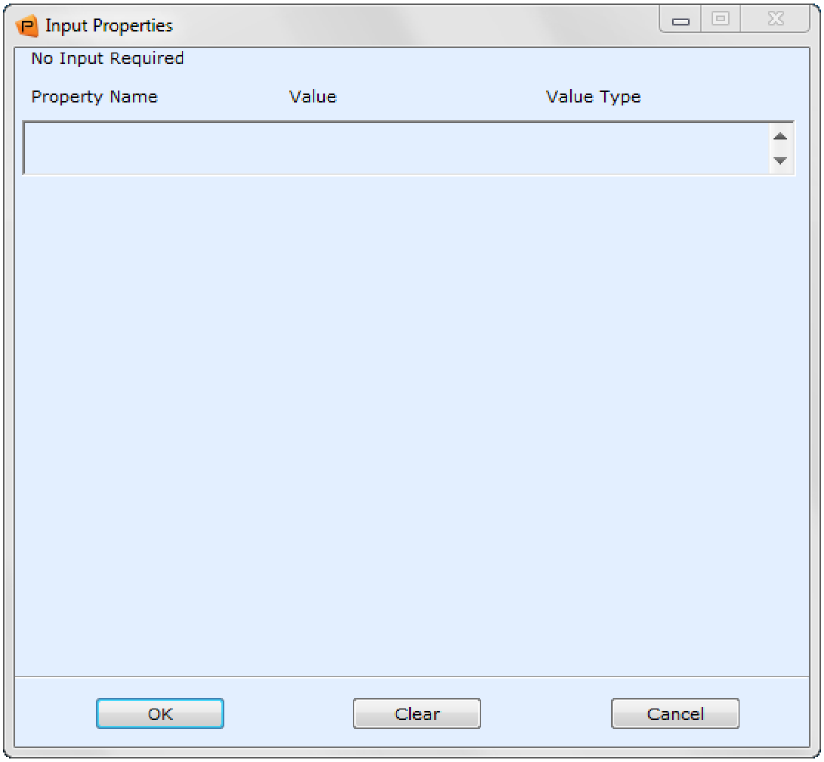
Eulerian Solid

One of eight subordinate forms appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Object	Type	Option	Topologies
Create	3D	Eulerian Solid	Hydro (PEULER + CTETRA/CPENTA/CHEXA)	Tet/4, Pent/6, Hex/8
			Strength (PEULER + CTETRA/CPENTA/CHEXA)	
			M/M Hydro (PEULER + CTETRA/CPENTA/CHEXA)	
			M/M Strength (PEULER + CTETRA/CPENTA/CHEXA)	
			Hydro (PEULER1 + CTETRA/CPENTA/CHEXA)	
			Strength (PEULER1 + CTETRA/CPENTA/CHEXA)	
			M/M Hydro (PEULER1 + CTETRA/CPENTA/CHEXA)	
			M/M Strength (PEULER1 + CTETRA/CPENTA/CHEXA)	

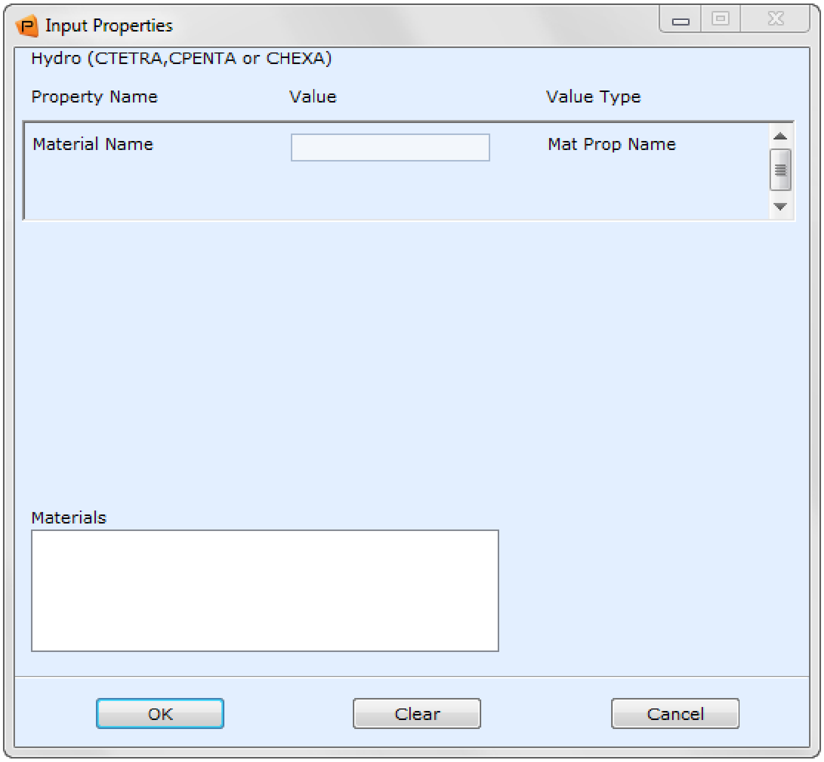
PEULER1 Property Definition

In contrast with traditional Lagrangian solids, Eulerian elements can contain multiple materials in one volume element. Therefore no materials have to be assigned on the Input Properties form, when the general initial condition generation using the PEULER1 option is used. Material and initial condition for Eulerian element property sets are then assigned by selecting Object: Init. Cond. Euler under Loads/BCs.



PEULER Property Definition

Use this form to create Eulerian properties, according to the Lagrangian approach. Each Eulerian element contains only one material. The material is selected on the Input Properties form.

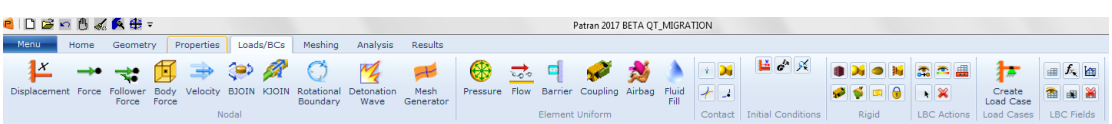


Parameter	Description
Material Name	Defines the material to be used. A list of all materials currently in the database is displayed when data is entered. Select one from the list using the mouse or type in the name. This property is required.

Loads and Boundary Conditions

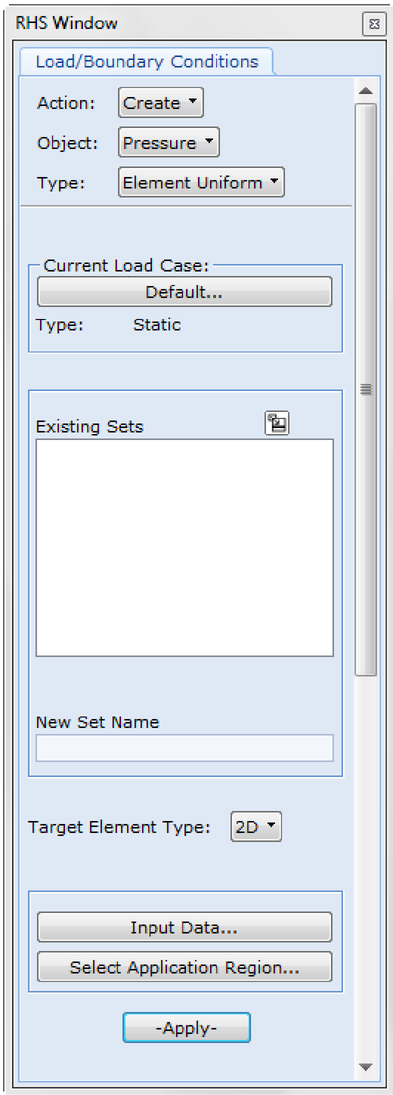
The Loads and Boundary Conditions form will appear when the Loads/BCs toggle, located on the Patran application selections, is chosen. When creating a loads and boundary condition there are several option menus. The selections made on the Loads and Boundary Conditions menu will determine which loads and boundary conditions form appears, and ultimately, which Dytran loads and boundary conditions will be created.

The following pages give an introduction to the Loads and Boundary Conditions form, and details of all the loads and boundary conditions supported by the Patran Dytran Analysis Preference.



Loads & Boundary Conditions Form

This form appears when Loads/BCs is selected on the main form. The Loads and Boundary Conditions form is used to provide options to create the various Dytran loads and boundary conditions. For a definition of full functionality, see Loads and Boundary Conditions Form (p. 18) in the *Patran Reference Manual.*



Parameter	Description
Object	Defines the general load type to be applied. Object choices are Displacement, Force Pressure, Initial Velocity, Follower Force, Contact, Planar Rigid Wall, Nodal Rigid Body, KJOIN, BJOIN, Rigid Ellipsoid, Init. Cond. Euler, Flow, Barrier, Rigid Body Object, Detonation Wave, Rigid Connection, Rigid Body Hinge, Init. Rotation Field, Rotational Boundary, Coupling, Airbag, Fluid Filled Containers.
Type	Defines what type of region is to be loaded. The available options here depends on the selected Object. The general selections can be Nodal, Element Uniform, or Element Variable. Nodal is applied explicitly to nodes. Element Uniform defines a constant value to be applied over an entire element, element face, or element edge.
Current Load Case	Current Load Case type is set on the Load Case menu. When the Load Cases toggle (located on the main form) is chosen, the Load Cases menu will appear. Under Load Case Type, select either Static or Time Dependent, then enter the name of the case, and click on the apply button.
Target Element Type	Defines the target element type to which this load will be applied. This only appears if the type is Element Uniform. This can be 2D or 3D.
Input Data	Generates either a Static or Transient Input Data form, depending on the current Load Case Type.

The following table outlines the options when Create is the selected action.

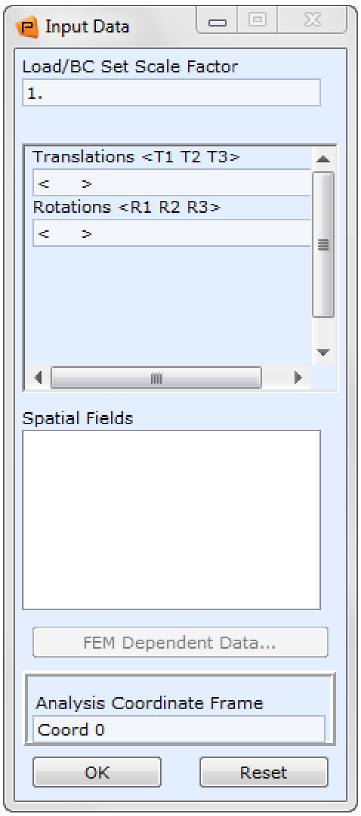
Table 2-4 Loads and Boundary Conditions

Object	Type
Displacement	Nodal
Force	Nodal
Pressure	Element Uniform
Initial Velocity	Nodal
Velocity	Nodal
Follower Force	Nodal
Contact	Element Uniform
Planar Rigid Wall	Nodal
Nodal Rigid Body	Nodal
KJOIN	Nodal
BJOIN	Nodal

Object	Type
Rigid Ellipsoid	Nodal
Init. Cond. Euler	Element Uniform
Flow	Element Uniform
Barrier	Element Uniform
Rigid Body Object	Nodal
Detonation Wave	Nodal
Rigid Connection	Element Uniform
Rigid Body Hinge	Nodal
Init. Rotation Field	Nodal
Rotational Boundary	Nodal
Coupling	Element Uniform/Nodal
Airbag	Element Uniform
Fluid Filled Containers	Element Uniform
Body Force	Nodal
Rigid Surface	Element Uniform
Mesh Generator	Nodal
Rigid Joint Constraint	Nodal

Static (Not Time Varying)

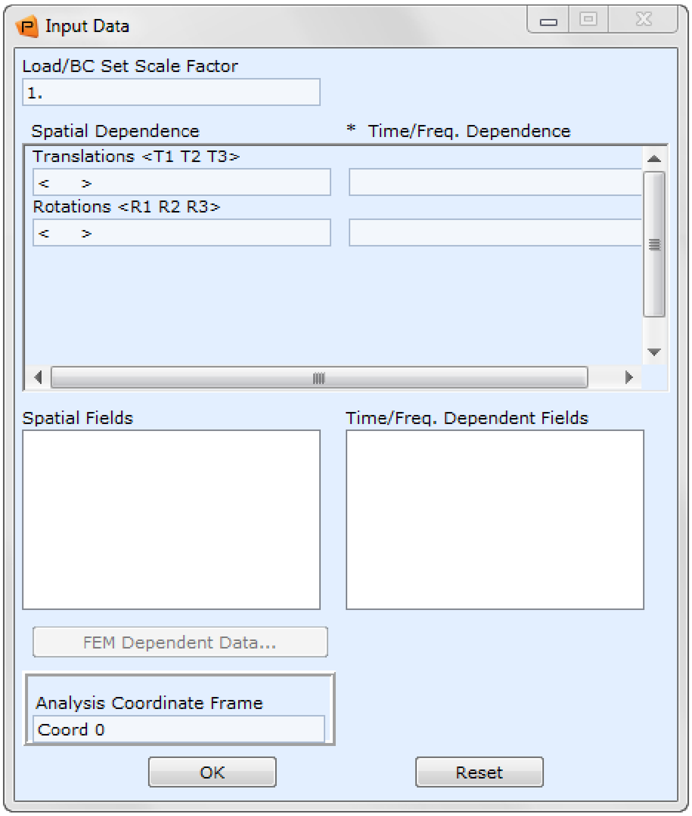
This subordinate form appears when the Input Data button is selected on the Loads and Boundary Conditions form when the Current Load Case Type is *Static*. The Current Load Case Type is set on the Load Case form. For more information, see Loads & Boundary Conditions Form. The information on the Input Data form will vary depending on the selected Object. Defined below is the standard information found on this form. Note that this form is not used with the Patran Dytran preference.



Parameter	Description
Load/BC Set Scale Factor	Defines a general scaling factor for all values defined on this form. The default value is 1.0. Primarily used when field definitions are used to define the load values.
Translations **Rotations**	Input Data in this section will vary. See also Eulerian Initial Conditions (p. 115) for detailed information.
Spatial Fields	Used when specifying real values in the Input Data entries, spatial fields can be referenced. All defined spatial fields currently in the database are listed. If the input focus is placed in the Input Data.

Transient (Time Varying)

This form appears when the Input Data button is selected on the Loads and Boundary Condition form when the Current Load Case Type is Time Dependent. The Current Load Case Type is set on the Load Case form. For more information, see Loads & Boundary Conditions Form and Load Cases. The information on the Input Data form will vary, depending on the selected Object. Defined below is the standard information found on this form.



Parameter	Description
Load/BC Set Scale Factor	Defines a general scaling factor for all values defined on this form. The default value is 1.0. Primarily used when field definitions are used to define the load values.
Spatial Dependence	Input Data in this section will vary. See Eulerian Initial Conditions (p. 115) for detailed information.
Spatial Fields	When specifying real values in the Input Data entries, spatial fields can be referenced. All defined spatial fields currently in the database are listed. If the input focus is placed in the Input Data entry, and a spatial field is selected by double clicking in this list, a reference to that field will be entered in the Input Data entry.

Parameter	Description
Time/Freq. Dependent Field	When specifying time dependent values in the Input Data entries, time dependent fields can be referenced. All defined time dependent fields currently in the database are listed. If the input focus is placed in the Input Data entry, and a time dependent field is selected by double clicking in this list, a reference to that field will be entered in the Input Data entry.
FEM Dependant Data	This button will display a Discrete FEM Fields input form to allow field creation and modification within the loads/bcs application. Visible only when focus is set in a databox which can have a DFEM field reference.
Analysis Coordinate Frame	Defines the coordinate frame to be used to interpret the degree-of-freedom data defined on the form. This only appears on the form for Nodal type loads. This can be a reference to any existing coordinate frame definition.

Object Tables

There are areas on the static and transient input data forms where the load data values are defined. The data fields which appear depend on the selected load Object and Type. In some cases, the data fields also depend on the selected Target Element Type. The following Object Tables outline and define the various input data that *pertains* to a specific selected object:

Displacement

Object	Type	Analysis Type
Displacement	Nodal	Structural

If the displacement/rotational component is one, this will result in generation of SPC, SPC1 or SPC3 Bulk Data entries, which defines translational and rotational constraints in the prescribed coordinate system. The standard Patran convention is used. Separate entries by commas, with a space denoting an unconstrained degree of freedom.

Input Data	Description
Translations (T1,T2,T3)	Defines the enforced translational displacement values. These are in *model length units*.
Rotations (R1,R2,R3)	Defines the enforced rotational displacement values. These are in *degrees*.

Force

Object	Type	Analysis Type
Force	Nodal	Structural

This defines a FORCE entry for transient load cases. Individual entries will be created if their defined loads do not have the same load curve.

Input Data	Description
Force (F1,F2,F3)	Defines the applied forces in the translation degrees-of-freedom.
Moment (M1,M2,M3)	Defines the applied moments in the rotational degrees-of-freedom.

Pressure

Object	Type	Analysis Type	Dimension
Pressure	Element Uniform	Structural	2D

Creates a PLOAD Bulk Data entry.

Input Data	Description
Top Surf Pressure	Defines the top surface pressure load on shell elements. If a scalar field is referenced, it will be evaluated once at the center of the applied region.
Bot Surf Pressure	Defines the bottom surface pressure load on shell elements. If a spacial field is referenced, it will be evaluated once at the center of the applied region.
Edge Pressure	This data is ignored by Dytran.

Object	Type	Analysis Type	Dimension
Pressure	Element Uniform	Structural	3D

Creates a PLOAD Bulk Data entry.

Input Data	Description
Pressure	Defines the face pressure value on solid elements. If a spacial field is referenced, it will be evaluated once at the center of the applied region.

Initial Velocity

Object	Type	Analysis Type
Initial Velocity	Nodal	Structural

Creates a TICGP Bulk Data entry.

Input Data	Description
Trans Veloc (v1,v2,v3) **Rot Veloc (w1,w2,w3)**	Defines the V0 fields for translational degrees-of-freedom. Defines the V0 fields for rotational degrees-of-freedom.

Velocity

Object	Type	Analysis Type
Velocity	**Nodal**	**Structural**

Creates a FORCE Bulk Data entry.

Input Data	Description
Trans Veloc(v1,v2,v3)	Defines the enforced translational velocity values. These are in model length units per unit time.
Rot Veloc (w1,w2,w3)	Defines the enforced rotational velocity values. These are in degrees per unit time.

Follower Force

Object	Type	Analysis Type
Follower Force	Nodal	Structural

This defines FORCE1, FORCE2, MOMENT1 or MOMENT2 Bulk Data entries. The data varies depending upon the entry type.

Input Data	Description
Force vs Time	Defines the applied force at each node in the application region.
Grid Point 1 **Grid Point 2** **Grid Point 3** **Grid Point 4**	Grid point list.

Contact

Object	Type	Analysis Type	Dimension
Contact	Element Uniform	Structural	Dual Application

Six types of contact exist:

1. Self Contact

2. Subsurface
3. Master Slave Surface
4. Master Slave Node
5. Adaptive Self Contact
6. Adaptive Master Slave Surface

These are seen as options on the Loads/Boundary Conditions form. The "Subsurface" type exists only as a mechanism for defining parts of a larger surface and has no associated data. The content of the Input Data form will depend upon whether the "Basic" or "Advanced" form type is chosen. Refer to the Dytran User's Manual for information on the various options available when the "Advanced" features are used. See also Contact for a more elaborate description of the different contact forms. The following data are used to complete the CONTACT and CONTFORC Bulk Data entries.

Input Data	Description
Static Friction Coefficient	Static coefficient of friction.
Kinetic Friction Coefficient	Kinetic coefficient of friction.
Exponential Decay Coefficient	Exponential decay coefficient EXP.
Contact Activation Time	Time at which the contact is activated.
Contact Deactivation Time	Time at which the contact is deactivated.
Contact Thickness	For shell elements this is a multiplier on the actual thickness.
Gap	Artificial contact thickness.
Penetration Tolerance	Tolerance for the initial penetration check.
Initial Monitoring Distance	Defines the fixed part of the monitoring distance.
Penetration Depth/Factor	Value of the allowed penetration.
Monitoring Region Velocity	Scale factor (MONVEL) on relative velocity.
Contact Force Scale Factor	Scale factor for the contact forces.
Monitoring Distance Factor	This defines the fixed part of the monitoring distance.
Projection Tolerance	Project tolerance for inside and outside corners.
View Angle	View angle of edges (in degrees)
Contact Force Stiffness	Contact force stiffness
Load (Force vs Depth)	Force vs penetration depth for the loading phase
Unload (Force vs Depth)	Force vs penetration depth for the unloading phase
Damper Stiffness	Damper stiffness

Planar Rigid Wall

Object	Type	Analysis Type
Planar Rigid Wall	Nodal	Structural

Two different planar rigid wall options exist:

1. Kinematic rigid wall without friction
2. Penalty method based rigid wall with friction

These are seen as options at the top of the Input Data form. The user must select which wall will be used. Both wall's position and orientation are defined by selecting a coordinate system which has its origin on the plane and the local z axis as the outward normal from the contact surface. This defines a WALL Bulk Data entry. There are only parameters associated with the penalty based planar rigid wall.

Input Data	Description
Static Friction Coefficient	Static coefficient of friction.
Kinetic Friction Coefficient	Kinetic coefficient of friction.
Exponential Decay Coefficient	Exponential decay coefficient EXP.

Nodal Rigid Body

Object	Type	Analysis Type
Nodal Rigid Body	Nodal	Structural

This defines a rigid body whose properties will be computed by Dytran. There is no data associated with this type of rigid body. This defines a RBE2 FULLRIG Bulk Data entry.

KJOIN

Object	Type	Analysis Type
KJOIN	Nodal	Structural

This defines a KJOIN Bulk Data entry. In addition to the data below the user can control whether "Interference" is "Strong" or not (Weak).

Input Data	Description
Tolerance	Tolerance for joining grid points.
Stiffness	Relative stiffness of the kinematic joint.

BJOIN

Object	Type	Analysis Type
BJOIN	Nodal	Structural

Four failure options exist:

1. Constant Force/Moment
2. User Defined
3. At Specified Time
4. Component Failure
5. Spotweld
6. Rupture

These are seen as options at the top of the Input Data form. The user must also select whether grid point positions are to be equivalenced and whether multiple breaks are allowed. The defaults are Yes and No respectively.

The other option on the form and the following data are used to complete the BJOIN Bulk Data entry.

Input Data	Description
Force at Failure **Moment at Failure**	(Option 1 only)
EXBRK Subroutine Name	(Option 2 only)

Input Data	Description
Time of Failure	(Option 3, 5 and 6 only)
x-Force at Failure **y-Force at Failure** **z-Force at Failure** **x-Moment at Failure** **y-Moment at Failure** **z-Moment at Failure**	(Option 4 only)
Tension Failure **Compression Failure** **Shear Failure** **Torque Failure** **Bending Failure** **Total Force Failure** **Total Moment Failure**	(Options 5 and 6 only)

Rigid Ellipsoid

Object	Type	Analysis Type
Rigid Ellipsoid	Nodal	Structural

An ellipsoidal rigid wall is defined by this entry. It may be static or have an initial motion. The selected coordinate system defines the centroid of the ellipsoid and its orientation. This defines a RELLIPS Bulk Data entry.

Input Data	Description
Mass	Mass of the rigid ellipse.
X-Dimension	Semi axis in local x direction.
Y-Dimension	Semi axis in local y direction.
Z-Dimension	Semi axis in local z direction.
Initial Velocity <vx,vy,vz>	Initial translational velocity in the local coordinate frame.
Initial Rotations <wx,wy,wz>	Initial rotational velocity in the local coordinate frame.

Init. Cond. Euler

Object	Type	Analysis Type	Dimension
Init. Cond. Euler	Element Uniform	Structural	3D

The initial condition and location of the materials in the Eulerian mesh is defined. See also for a more elaborate description of the forms. The initial condition of the materials is defined in geometrical regions, using the options:

1. Shape/Surface
2. Shape
3. Initial Values
4. Region Definition
5. Simple

Option: ShapeSurface

Defines a region inside or outside a multifaceted surface. This defines a MATINI Bulk Data entry.

Input Data	Description
Cover	Processing strategy for Eulerian elements that are inside or outside of the initialization surface.
Reverse Normals	Auto reverse switch for MATINI surface segments.
Check Normals	Checking switch for MATINI surface segments.

Option: Shape

Defines a spherical or cylindrical shape or a shape defined by an application region. This defines a SPHERE, CYLINDER or SET1 Bulk Data entry.

Input Data	Description
Radius of Sphere	Radius of a spherical region (Shape: Sphere).
Centroid	Local coordinate frame for the definition of a spherical or cylindrical region (Shape: Sphere and Shape: Cylinder).
Radius of Cylinder	Radius of cylindrical region (Shape: Cylinder).
Length of Cylinder	Length of the cylindrical region (Shape: Cylinder).

Option: Initial Values

Defines the initial values of the materials in the Eulerian mesh. This defines a TICVAL Bulk Data entry.

Input Data	Description
Material	Material for which initial values are defined.
Init. Velocity <Vx, Vy, Vz>	Initial velocity of the material.
Density	Initial density of the material
Specific Internal Energy	Initial specific internal energy of the material.

Option: Region Definition

Defines where materials and initial conditions are applied in the Eulerian mesh. This defines a TICEUL Bulk Data entry.

Input Data	Description
Existing PEULER1 Sets	Selection of the Eulerian property set.
Existing Shape Sets	Selection of geometrical region.
Existing Initial Value Sets	Selection of initial value set.
Level indicator	Hierarchy of the shape/initial value set.

Option: Simple

Defines the initial values of the materials in the Eulerian mesh. This defines a TICEL Bulk Data entry.

Input Data	Description
Material	Material for which initial values are defined.
Init. Velocity <Vx, Vy, Vz>	Initial velocity of the material.
Density	Initial density of the material
Specific Internal Energy	Initial specific internal energy of the material.

Flow

Object	Type	Analysis Type	Dimension
Flow	Element Uniform	Structural	3D

Creates a flow boundary for Eulerian meshes and defines a FLOW Bulk Data entry. See also Flow for a more elaborate description of the forms.

Input Data	Description
Material	Material that flows in or out the Eulerian mesh.
Velocity <Vx, Vy, Vz>	Material velocity at the flow boundary.
Pressure at the Boundary	Pressure at the flow boundary.
Density at the Boundary	Material density at the flow boundary.
Specific Internal Energy	Specific Internal Energy of the material at the flow boundary.

Barrier

Object	Type	Analysis Type	Dimension
Barrier	Element Uniform	Structural	3D

Creates a barrier boundary for Eulerian meshes through which no material can flow and defines a WALLET Bulk Data entry. There is no data associated with the Eulerian BARRIER definition. See also Barrier for a more elaborate description of the forms.

Rigid Body Object

Object	Type	Analysis Type
Rigid Body Object	Nodal	Structural

Defines the constraint, velocity or force (FORCE and MOMENT entries) applied to the center of gravity of the rigid body defined by the MATRIG entry. See also Rigid Body Object for a more elaborate description of the forms.

Input Data	Description
Material	Select Rigid Material.
Translations (T1,T2,T3)	Defines the enforced translational velocity values.
Rotations (R1,R2,R3)	Defines the enforced rotational velocity values.
Force (F1,F2,F3)	Defines the applied forces in the translation degrees-of-freedom.
Moment (M1,M2,M3)	Defines the applied moments in the rotational degrees-of-freedom.

Detonation Wave

Object	Type	Analysis Type
Detonation Wave	**Nodal**	**Structural**

Defines a ignition point in the Eulerian mesh from which a spherical detonation wave travel, causing the reaction of high explosive materials. Creates a DETSPH Bulk Data entry. See also Detonation Wave for a more elaborate description of the forms.

Input Data	Description
Material	Select JWL Material.
Wave velocity	Velocity of the spherical detonation wave.
Detonation Time	Time when the ignition point detonates.

Rigid Connection

Object	Type	Analysis Type	Dimension
Rigid Connection	Element Uniform	Structural	Dual Application

Defines a rigid connection between different parts of Lagrangian meshes.

Three types of connections can be used:

1. Two Surfaces Tied Together;
2. Grid Points Tied to a Surface;
3. Shell Edge Tied to a Shell Surface

The user must also select whether the gaps should be automatically closed (options are Default, Yes and No) and define the type of gap tolerance (options Distance and Factor (option 1 only)).

The following data are used to complete the RCONN Bulk Data entry. See also Rigid Connection for a more elaborate description of the forms.

Input Data	Description
Gap Tolerance Value	Value of the gap tolerance (option Distance).
Gap Tolerance Factor	Factor to calculate the gap tolerance (option Factor).

Rigid Body Hinge

Object	Type	Analysis Type
Rigid Body Hinge	Nodal	Structural

Defines a hinge between rigid body and a deformable structure.

Two types of connections can be used:

1. Rigid Material;
2. Nodal Rigid Body;

Creates a RBHINGE Bulk Data entry. See also Rigid Body Hinge for a more elaborate description of the forms.

Input Data	Description
Material	Select Rigid Material (option 1 only).
Nodal Rigid Body	Select Nodal Rigid Body (option 2 only).
Hinge Component	Rotation of the hinge (RX, RY or RZ).

Initial Rotation Field

Object	Type	Analysis Type
Init. Rotation Field	Nodal	Structural

Defines a velocity field of grid points consisting of a rotation and a traslation specification.

Creates a TIC3 Bulk Data entry. See also Initial Rotation Field for a more elaborate description of the forms.

Input Data	Description
Trans Veloc(v1,v2,v3)	Defines the initial translational velocity values. These are in model length units per unit time.
Rot Veloc (w1,w2,w3)	Defines the initial rotational velocity values. These are in degrees per unit time.
Rotation Center	Defines a point at the center of rotation.

Rotational Boundary

Object	Type	Analysis Type
Rotational Boundary	Nodal	Structural

Defines a rotational boundary constraint on grid points.

Two types of radial velocity can be used:

1. Free;
2. Constraint;

Creates a SPC2 Bulk Data entry. See also Rotational Boundary for a more elaborate description of the forms.

Input Data	Description
Angular Velocity	Defines the rotational (angular) velocity value.
Rotation Vector (v1,v2,v3)	Defines the rotation vector.
Rotation Center	Defines a point at the center of rotation.

Coupling

Object	Type	Analysis Type	Dimension
Coupling	Element Uniform or Nodal	Structural	Dual Application

Defines a coupling surface that acts as the interface between an Eulerian and a Lagrangian domain. Interaction between two coupling surfaces can also be defined.

Five types of coupling exist:

1. General Subsurface
2. Subsurface
3. General
4. With Failure
5. Interaction
6. ALE
7. ALE Grid1
8. ALE Grid

General Subsurface should be used in General Coupling while Subsurface should be used in coupling with failure.

There is no input data associated with subsurface and iteraction definition.

The Fast General Coupling is generated as General Coupling on the Loads/Boundary Conditions form with an additional definition as Fast General Coupling on the “Coupling Parameters” form under “Execution Controls”.

For the Fast General Coupling an additional PARAM, FASTCOUP Bulk Data entry is defined.

The following Bulk Data entries can be defined within this lbc:

COUPLE, COUPOR, COUOPT, COUPLE1, COUP1FL, COUP1INT, PORFLOW, ALE, ALEGRID, ALEGRID1

See also Coupling for a more elaborate description of the forms.

Airbag

Object	Type	Analysis Type	Dimension
Airbag	Element Uniform	Structural	Dual Application

Defines porosity, inflator and Heat Transfer in Air Bags.

Two types of airbag exist:

1. Subsurface
2. Surface

Subsurface should be used in surface.

The following Bulk Data entries can be defined within this lbc:

GBAG, GBAGPOR, GBAGHTR, GBAGINFL, GBAGCOU, COUPLE, COUPOR, COUHTR, COUOPT, COUINFL, PERMEAB, PERMGBG, PORFCPL, PORFGBG, PORFLCPL, PORFLGBG, PORHOLE, PORLHOLE, HTRCONV, HTRRAD, INFLATR, INFLATR1, INFLFRAC, INFLGAS, INFLHYB, INFLHYB1, INFLTANK, INITGAS

See Airbag for a more elaborate description of the forms.

Fluid Filled Containers

Object	Type	Analysis Type	Dimension
Fluid Filled Containers	Element Uniform	Structural	Dual Application

Defines the pressure within a closed volume in the Eulerian mesh.

Two types of Fluid Filled Containers exist:

1. Subsurface
2. Surface

Subsurface should be used in surface.

Creates a FFCONTR Bulk Data entry. See also Fluid Filled Containers for a more elaborate description of the forms.

Input Data	Description
Fluid Volume	Fluid Volume in the container.
Atmospheric Pressure	Atmospheric Pressure.

Body Force

Object	Type	Analysis Type
Body Force	Nodal	Structural

Defines a body force loading.

It can be applies to four types of entities:

1. Lagrangian;
2. Eulerian

3. Ellipsoid
4. Grid;

Creates a BODYFOR Bulk Data entry. See Body Force for a more elaborate description of the forms.

Input Data	Description
Element Type	Select Element type (options 1 and 2 only).
Rigid Ellipsoid	Select Rigid Ellipsoid (option 3 only).
Refer. Coordinate Frame	Coordinate Frame.
Scale Factor	Defines a constant scale factor or a tabular field
Load Direction <N1,N2,N3>	Defines the load direction vector.

Rigid Surface

Object	Type	Analysis Type
Rigid Surface	Element Uniform	Structural

Defines a rigid surface.

Two types of Rigid Surface exist:

1. Subsurface
2. Surface

Subsurface should be used in surface.

Creates a RIGID Bulk Data entry. See Rigid Surface for a more elaborate description of the forms.

Input Data	Description
Center of Gravity	Defines a point at the center of gravity..
Mass	Mass of the Rigid Body.
Trans Veloc(v1,v2,v3)	Defines the initial translational velocity values. These are in model length units per unit time.
Rot Veloc (w1,w2,w3)	Defines the initial rotational velocity values. These are in degrees per unit time.
Inertia Ixx about CG	Defines the Inertia Ixx about the center of gravity
Inertia Ixy about CG	Defines the Inertia Ixy about the center of gravity
Inertia Ixz about CG	Defines the Inertia Ixz about the center of gravity
Inertia Iyy about CG	Defines the Inertia Iyy about the center of gravity

Input Data	Description
Inertia Iyz about CG	Defines the Inertia Iyz about the center of gravity
Inertia Izz about CG	Defines the Inertia Izz about the center of gravity
Refer. Coordinate Frame	Coordinate Frame.

Mesh Generator

Object	Type	Analysis Type
Mesh Generator	Nodal	Structural

Defines a mesh.

Two types of Mesh exist:

1. Box
2. Adaptive

Creates a MESH Bulk Data entry. See Mesh Generator for a more elaborate description of the forms.

Input Data	Description
Origin	Defines the coordinates of the point of origin (option 1 only).
Box Size	Vector defining the size of the box (option 1 only).
Number of Elem in the X dir	Defines the number of elements in the X dir. (option 1 only).
Number of Elem in the Y dir	Defines the number of elements in the Ydir. (option 1 only).
Number of Elem in the Z dir	Defines the number of elements in the Z dir. (option 1 only).
Reference Point	Defines the coordinates of the Reference Point (option 2 only).
Euler Elem. Mesh Size	Vector defining the size of the Euler Element (option 2 only).
Starting Node ID	Starting ID for the nodes
Starting Elem ID	Starting ID for the elements
Resize Method	Defines the method used for resizing (option 2 only).
Resize in the X dir	Resize in the X direction (option 2 only).
Resize in the Y dir	Resize in the Y direction (option 2 only).
Resize in the Z dir	Resize in the Z direction (option 2 only).
Coupling lbc	Select Coupling lbc
3D Property	Select 3D Property

Rigid Joint Constraint

Object	Type	Analysis Type
Rigid Joint Constraint	Nodal	Structural

Defines a rigid joint.

Six types of Rigid Joint exist:

1. Cylindrical
2. Planar
3. Revolute
4. Spherical
5. Translational
6. Universal

Depending on the option selected, creates a RJCLY, RJPLA, RJREV, RJSPH, RJTRA or RJUNI Bulk Data entry. See Rigid Joint Constraint for a more elaborate description of the forms.

Input Data	Description
Stiffness	Defines the stiffness of the joint.
Node G1	Node ID of grid point 1
Node G2	Node ID of grid point 2
Node G3	Node ID of grid point 3 (options 1,2,3,5,6)
Node G4	Node ID of grid point 4 (options 1,2,3,5,6)
Node G5	Node ID of grid point 5 (option 5 only)
Node G6	Node ID of grid point 6 (option 5 only)

Contact

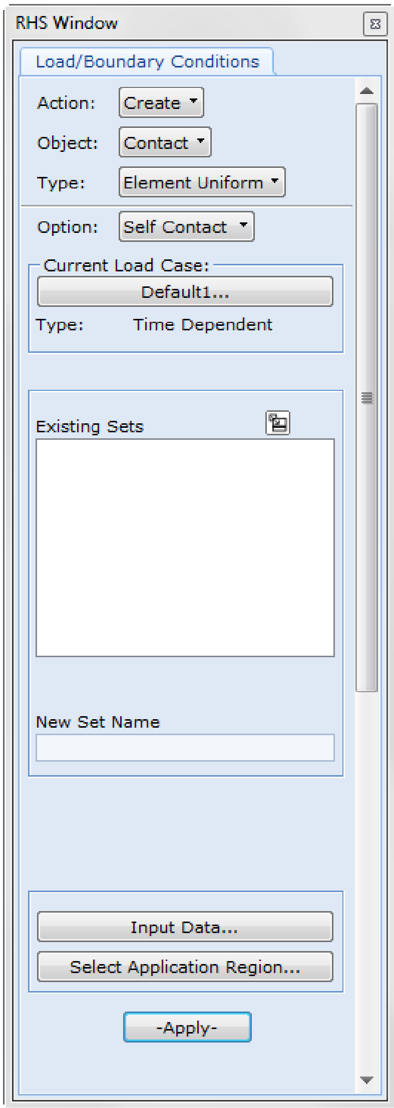
Introduction

This section describes the user interface provided by Patran to access the contact features of explicit dynamics finite element codes. This interface is used during definition of the Contact LBC types: Self Contact, Subsurface, Master/Slave Surface, Master/Slave Node, Adaptive Self Contact and Adaptive Master/Slave Surface.

Tools have been provided to enable the user to quickly and easily define contact conditions. Specification of contact is conceptually simple, involving either one or two contact surfaces, and a set of contact parameters which control the interaction of the surfaces.

Contact Types

A contact condition in which a single logical surface may come into contact only with itself is described as self-contact, and requires the specification of a single Application Region. A contact condition in which two logical surfaces may contact each other is described as Master/Slave contact, and requires specification of two Application Regions. Master/Slave contact is further subdivided by the definition of Master/Slave Surface and Master/Slave Node. Master/Slave Surface describes the condition in which both the master and slave surfaces are described using element faces, whereas Master/Slave Node describes the condition in which the Slave surface is described using only nodes.


RHS Window
Load/Boundary Conditions
Action: Create
Object: Contact
Type: Element Uniform
Option: Self Contact
Current Load Case:
Default1...
Type: Time Dependent
Existing Sets
New Set Name
Input Data...
Select Application Region...
-Apply-

Parameter	Description
Option	Choose between: -Self Contact -Subsurface -Master-Slave Surface -Master-Slave Node -Adaptive Self Contact -Adaptive Master-Slave Surface

Contact Construction

Tools are provided to enable the construction of contact surfaces, using the standard Patran select tool mechanisms (2D elements, 3D element faces), or groups. Contact subsurfaces can also be constructed using these tools, and later used to define a complete logical contact surface. This functionality allows the user to use the select tool to specify application regions on Patran geometry or the associated FEM entities or to define a more complex contact surface that is assembled from a mixture of 2D and 3D element faces, and to simply combine groups of 2D elements taking into account the direction of the contact outward normal. (For 2D elements, the outward normal can be reversed for contact purposes without modifying the underlying element topology.) Use of the group select mechanism is restricted to FEM entities only. Visualization of the specified contact condition is provided by graphically previewing but is not currently supported for geometry entities.

"Simple" contact surfaces include surfaces which may be described entirely by the faces of 3D elements, or by 2D elements whose outward normals are aligned with the desired contact normal direction. These contact surfaces may be constructed entirely using a single select mechanism (either Select Tool or Group method). Simple contact surfaces may not include a mixture of 3D element faces and 2D elements, or 2D elements whose outward normals are not all aligned with the desired contact normal direction.

"Complex" contact surfaces are defined as those surfaces which consist of a mixture of 2D elements and 3D element faces, or all 2D elements but with some of the outward normal incorrectly aligned. Contact conditions which include complex contact surfaces must be constructed using "Subsurfaces," where each subsurface is a "Simple" contact surface. Definition of contact surfaces is limited to one method, i.e., it is not permissible to mix, "Select Tool," "Group," or "Subsurface" within the definition of a contact surface.

The following section describes how each of the contact surface creation methods is used to describe a simple contact surface.

Use of the Select Tool

The select tool is use to graphically select the desired entities from the model. When this method is selected, the user must specify which dimensionality the intended object has, i.e., 3D, 2D, or Nodal. If the selected dimensionality is 2D, then the user can further specify whether the top, bottom, or both surfaces are required. Selection of top will result in a contact surface whose outward normal is coincident with the element outward, whereas selection of bottom will result in a contact surface whose outward normal is in the opposite direction to the element outward normal. The user can toggle between Top, Bottom, or Both at any time during selection, however all of the selected entities will be assigned the same logical direction. Selection of 3D allows the user to select either all faces or all free faces of 3D elements. No user specification of the contact normal direction is required for 3D elements since the program automatically specifies this direction. No contact direction is applicable to Nodal contact surfaces.

It is not permissible to mix 3D, 2D, and Nodal entities within a single Application Region. (This functionality is provided through the use of contact subsurfaces). The select tool can be used to select on the basis of either FEM or Geometry entities.

Use of the Group Tool

The Group tool is used to define simple contact surfaces on the basis of Patran group names. When this method is selected, the user must specify which dimensionality the intended object has, i.e., either 3D, 2D, or Nodal. The entities which will be selected for use in the contact surface in this case are either all 3D free surfaces in the group, all 2D elements, or all nodes contained in the selected group. In the case of 2D elements, the user may specify whether the contact normal direction is coincident with the element top, bottom, or both faces. Multiple groups may be selected. However, it should be noted that both the selected element dimensionality and contact normal direction apply across all selected groups.

Use of the Subsurface Tool

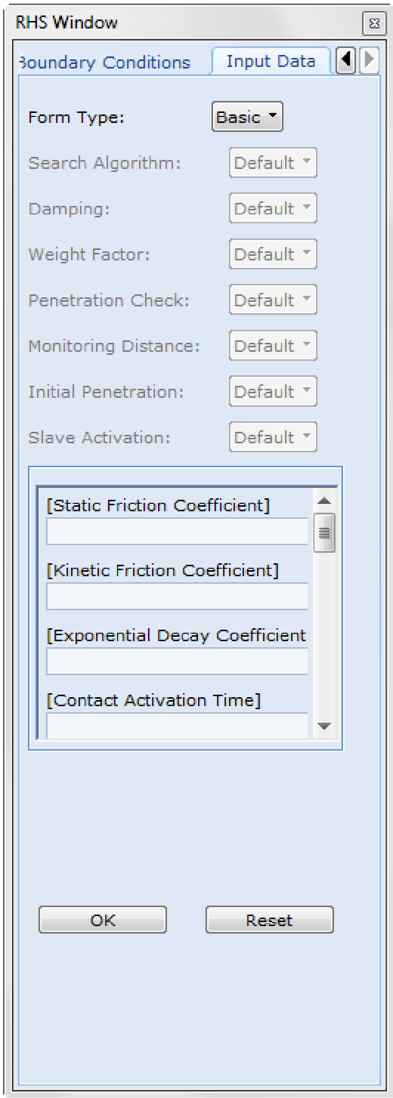
Contact Subsurfaces may be defined using either of the above methods. Subsurfaces may then be used in the specification of Master, Slave, or Self contact surfaces. When this option is used, the user may not specify element dimensionality or contact normal direction since this information has already been defined during subsurface definition. As many subsurfaces as required may be selected to form the desired complex contact subsurface.

Use of the Property Tool

The Property tool is used only in the adaptive contacts. Properties may then be used in the specification of Master, Slave, or Self contact surfaces. As many properties as required may be selected to form the desired complex contact set.

Contact: Input data

This form is used for the input data of a contact lbc.



Parameter	Description
Form Type	Choose between: - Basic (Default) and Advanced
Search Algorithm	Choose between: (Advanced only) - Default, Full, Slide, and B-Spline

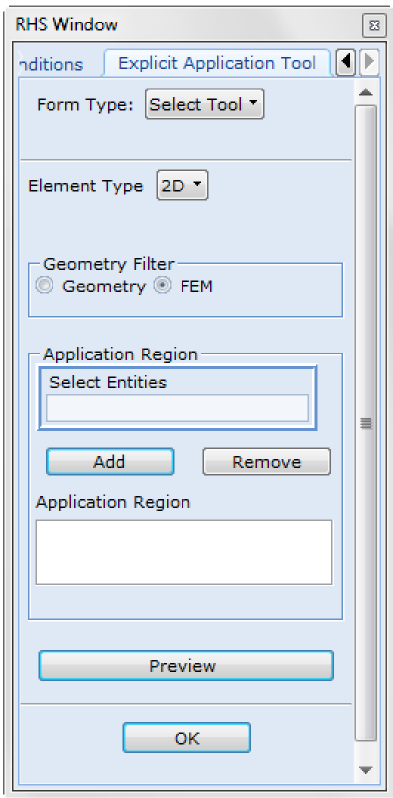
Parameter	Description
Damping	Choose between: (Advanced only) - Default, Yes, and No
Weight Factor	Choose between (Advanced only) - Default, Both, Slave, Master, and None
Penetration Check	Choose between: (Advanced only) - Default, Distance, and Factor
Monitoring Distance	Choose between: (Advanced only) - Default, Distance, and Factor
Initial Penetration	Choose between: (Advanced only) -Default, Check, and No Check
Slave Activation	Choose between: (Advanced adaptive only) -Default, Method 1, ... , Method 4A
Additional Databoxes	Additional databoxes: Basic: - Contact Deactivation Time - Contact Thickness - Gap Advanced: - Contact Deactivation Time - Contact Thickness - Gap - Penetration Tolerance - Initial Monitoring Distance - Penetration Depth/Factor - Monitoring relative Velocity - Contact Force Scale Factor - Monitoring Distance Factor - Projection tolerance - View Angle - Contact Force Stiffness - Load (Force vs depth) - Unload (Force vs depth) - Damper Stiffness

Contact: Application Region

This form is used to define contact surfaces. The form will vary depending upon which options are selected; however, two basic configurations are used depending on whether the contact condition requires specification of a single contact surface or two contact surfaces.

Single Application Region

The following form is used to define a single surface contact or a subsurface.



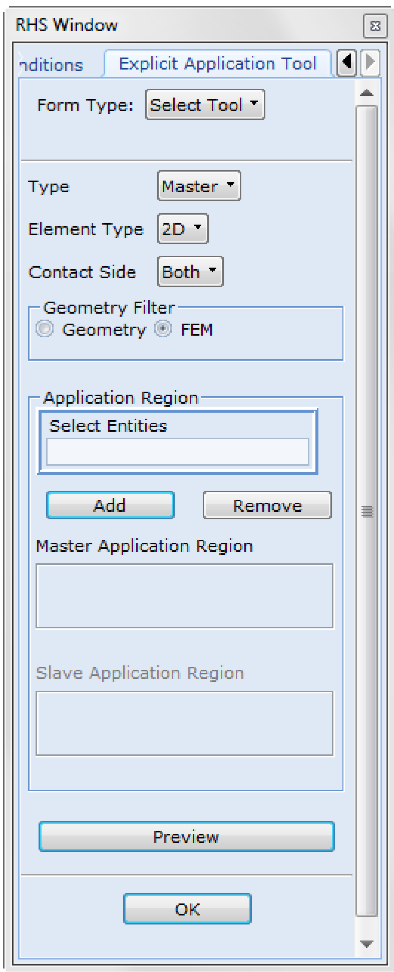
Parameter	Description
Form Type	Choose between: - Select Tool - Group - Subsurface (non-adaptive contact only) - Properties (adaptive Contact only)
Element Type	Choose between -2D -3D
Geometry Filter	Filter for picking Geometry or FEM entities.

Parameter	Description
Select Entity	Entity select databox. Entities appearing here may be added or removed from the active application region.
Application Region	List of entities in application region.
Preview	Preview contact surface graphically.

Note:	This form varies depending on the selection in the Form Type. Shown here is for type Select Tool.

Dual Application Region

The following form is used to define either of the master-slave contact types



Parameter	Description
Form Type	Choose between: - Select Tool - Group - Subsurface (non-adaptive contact only) - Properties (adaptive contact only)
Type	Choose between: - Master - Slave
Element Type	Choose between: -2D -3D
Contact Side	Choose between: (for Master 2D only) - Both (Default) - Top - Bottom
Geometry Filter	Filter for picking Geometry or FEM entities.
Select Entities	Entity select databox- Entities appearing here may be **Added** or **Removed** from the active application region.
Master Application Region	List of entities in application region.
Slave Application Region	Titles grey out when region is inactive.
Preview	Preview contact surface graphically.

Note: This part of the form varies depending on the selection in the Form Type. Shown here is for type Select Tool.

Eulerian Initial Conditions

Introduction

This section describes the user interface provided by Patran to model the initial state of the Eulerian part of the analysis model prior to running the analysis. It is important to recognize the difference between initial condition and enforced condition for an Eulerian model. Enforced conditions specify the loading and constraints of material throughout the transient analysis. The Eulerian loading and constraints are the flow boundary and barrier. Initial conditions, on the other hand, specify the state of the material only at the beginning of the analysis. Thereafter, the material state is determined by the calculation and the applied boundary and/or barrier conditions.

The definition of initial conditions within Eulerian meshes is somewhat different than that in a Lagrangian solid mesh. In a Lagrangian solid mesh the elements are completely filled with one material. The mesh is attached to the material, and each Property ID is linked to one specific material. However, in an Eulerian mesh the material can flow through the mesh from element to element, and a single element can contain up to five different materials. The Property ID of an element is not linked to a single material, but a number of different materials.

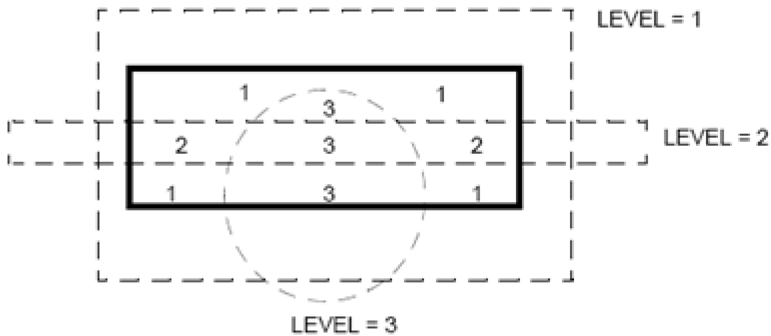
The initial conditions for an Eulerian analysis can be defined on geometrical regions within the Eulerian mesh, and on an element basis. The interface in Patran to define initial conditions for Eulerian analysis allows the generation of initial conditions in cylindrical or spherical geometry shapes and sets of elements. The interface is used during definition of the Contact LBC types: Self Contact, Master/Slave Surface, Master/Slave Node, and Subsurface.

Initial Condition Generation

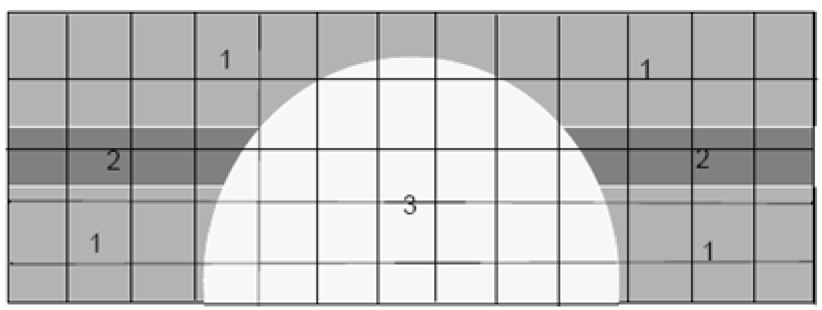
Each geometrical region (cylinder, sphere or set of elements) has a level number. This allows the creation of regions of arbitrary shape by allowing the regions to overlap. The part of an element that lies in two or more geometrical regions is assigned to the region that has the highest level number.

Think of geometrical regions as shapes cut out of opaque paper. Position the region of the lowest level number on the mesh. Then, place the next higher region on top of the first and continue until all the regions are in place. When the last region is placed, you have a map indicating to which region each element in the problem is assigned.

The following figure shows how three different geometrical regions can be used to create regions of arbitrary shape. The solid line represents the boundary of the Eulerian mesh. Region one (LEVEL = 1) is the large dashed rectangle. Region two (LEVEL = 2) is the long narrow rectangle. Region three (LEVEL = 3) is a circular region. Each geometrical region has its own specific material.



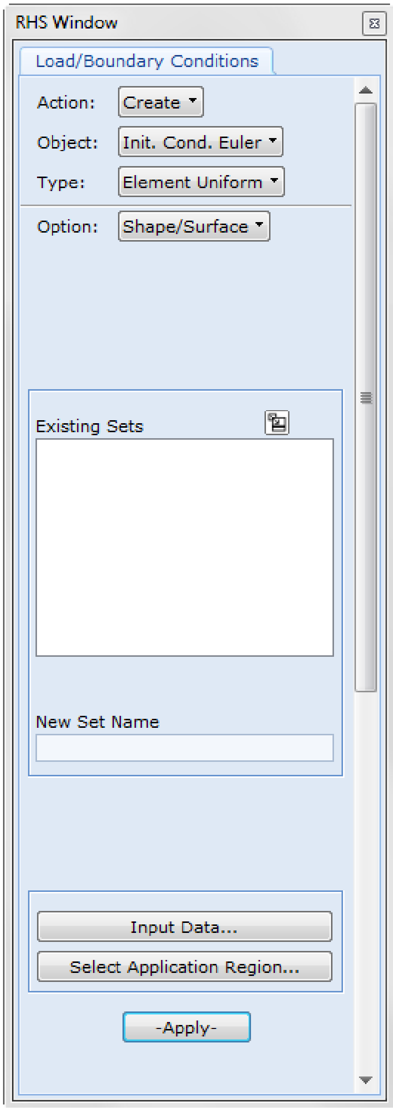
Below the results of the assignment of the different geometrical regions to each Eulerian element in the rectangular mesh is shown. Many elements have one material assigned, some have 2 materials assigned and 4 elements have 3 different materials assigned to them.



Initial Condition Construction

Tools are provided to enable the construction of initial conditions. The following steps need to be taken to fully define the assignment of material and initial conditions to the elements of an Eulerian mesh with a certain defined property:

1. Define the different geometrical regions by using Option:Shape/Surface or Option:Shape after selection of Object: Init. Cond. Euler.
2. Define the initial conditions of the material in the different geometrical regions by using Option: Initial Values. Each geometrical region can only have one material and initial value definition.
3. Define the assignment of material and initial conditions for a selected Eulerian Property set by selecting Option: Region Definition.

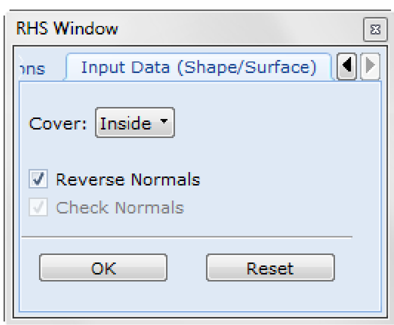

RHS Window
Load/Boundary Conditions
Action:
Create
Object:
Init. Cond. Euler
Type:
Element Uniform
Option:
Shape/Surface
Existing Sets
New Set Name
Input Data...
Select Application Region...
-Apply-

Parameter	Description
Option	Available are the following options: Shape/Surface Shape Initial Values Region Definition Simple

Shape Definition

Shape - Surface

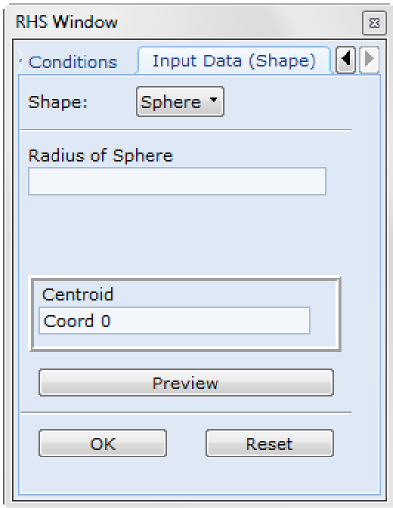
Defines the surface shape within a closed volume for the construction of the Eulerian mesh.

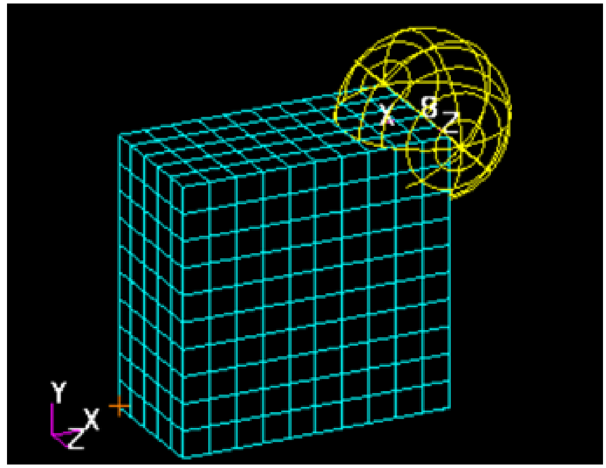


Parameter	Description
Cover	Choose between: - Inside (Default) - Outside
Reverse Normal	If needed, the normals are reversed to obtain a positive surface volume, depending on the selected cover option (inside/outside).
Check Normal	The normals of the faces of the surface are checked to see whether they all point in the same direction and give a positive closed volume.

Shape - Sphere

This shape selection creates a spherical Eulerian region

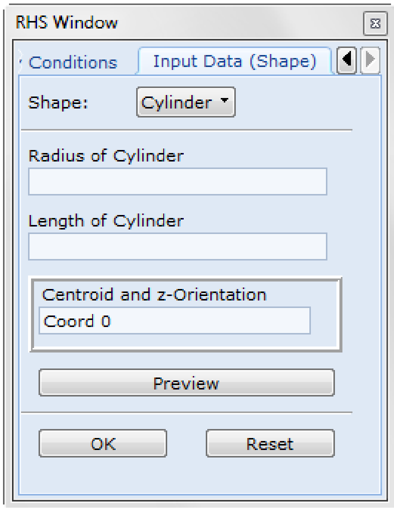


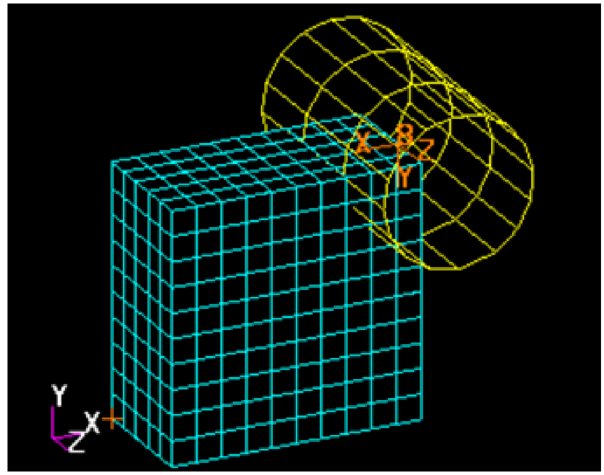


Parameter	Description
Radius of Sphere	The spherical region is defined by selecting a coordinate system for the centroid and defining a radius.
Centroid	
Preview	Upon selection of the Preview option the spherical shape is visualized in the viewport (as shown in the picture below.

Shape - Cylinder

This shape selection creates a cylindrical Eulerian region.

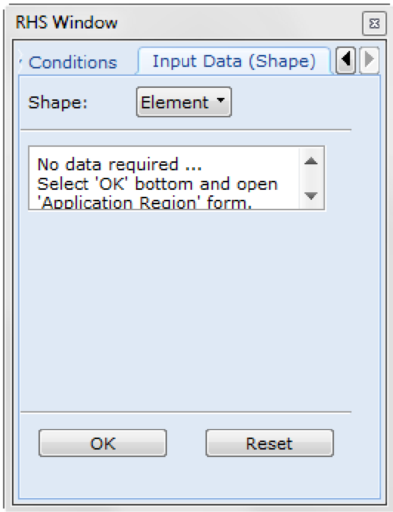


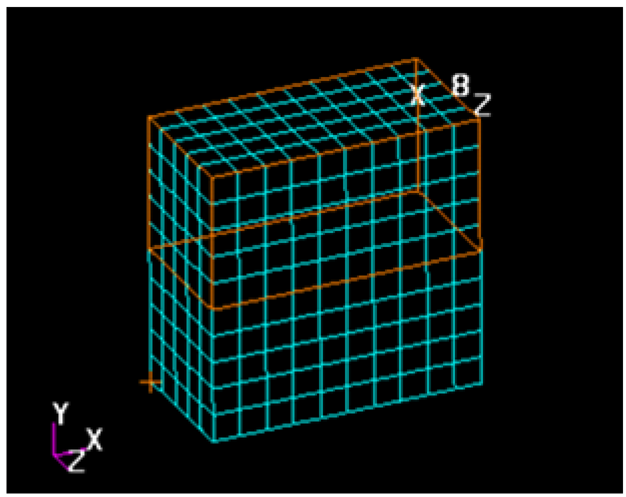


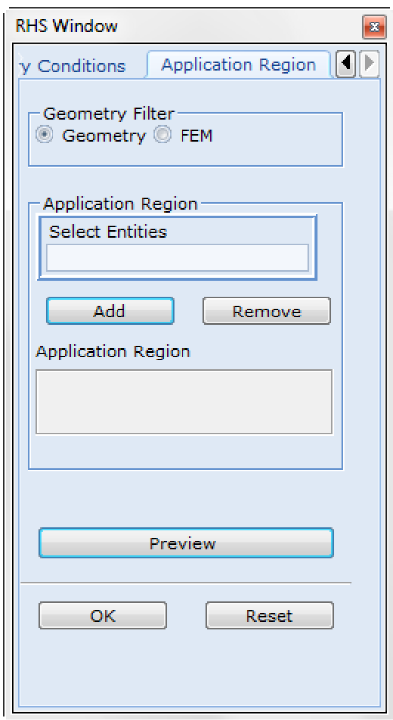
Parameter	Description
Radius of Cylinder	The cylindrical region is defined by selecting a coordinate system and defining the radius and the length of the cylinder. Note that the coordinate system should be located at the centroid of the cylinder with the z axis pointing from point 1 to 2 of the CYLINDER card.
Length of Cylinder	
Centroid and z-Orientation	
Preview	Upon selection of the Preview option the cylindrical shape is visualized in the viewport (as shown in the picture below).

Shape - Element

This shape selection creates an Eulerian region based upon element selection. On the Input Data form only the Shape: Element is selected and Select Application Region on the LOAD/LBC form defines the region.



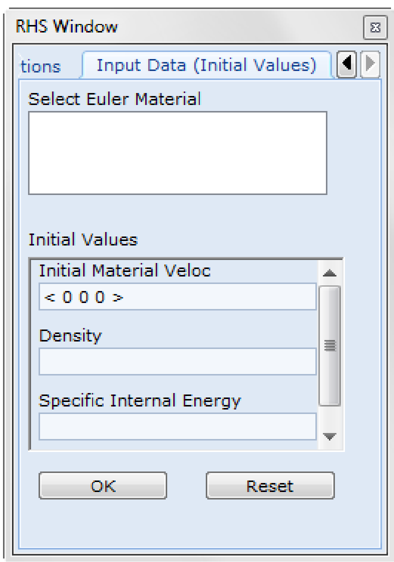




Parameter	Description
Shape	Select Shape: Element and define the element selection with the Application Region option on the Load/LBC form.
Preview	Upon selection of the Preview option the selected Application Region shape is highlighted in the viewport (as shown in the picture above).

Initial Values Definition

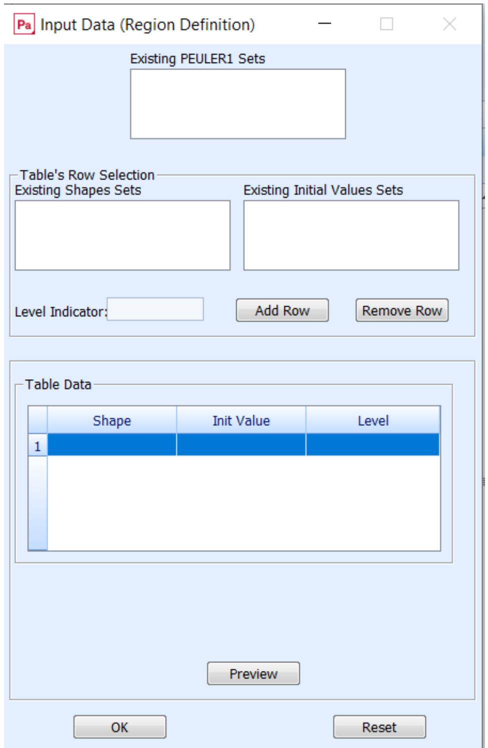
Defines the initial values of an Eulerian geometric region. This form defines the values of the initial condition, and the value set is assigned to a particular region using Option: Region definition.

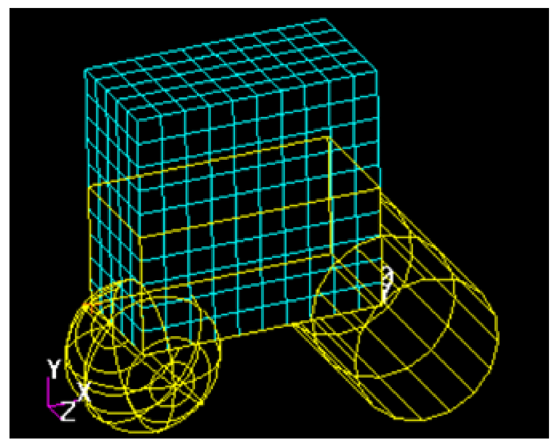


Parameter	Description
Select Euler Material	Only Eulerian materials that have been defined are shown. If no material is selected for the initial value set, a void region is assumed in which no material is present.
Initial Values	Initial conditions for the material velocity and specific internal enenergy that are not specified are assumed to be zero. Density is set to the reference density, as defined on the material form.

Region Definition

Defines for a selected Eulerian element property set, the geometrical region and associated initial value set, and their hierarchical levels.

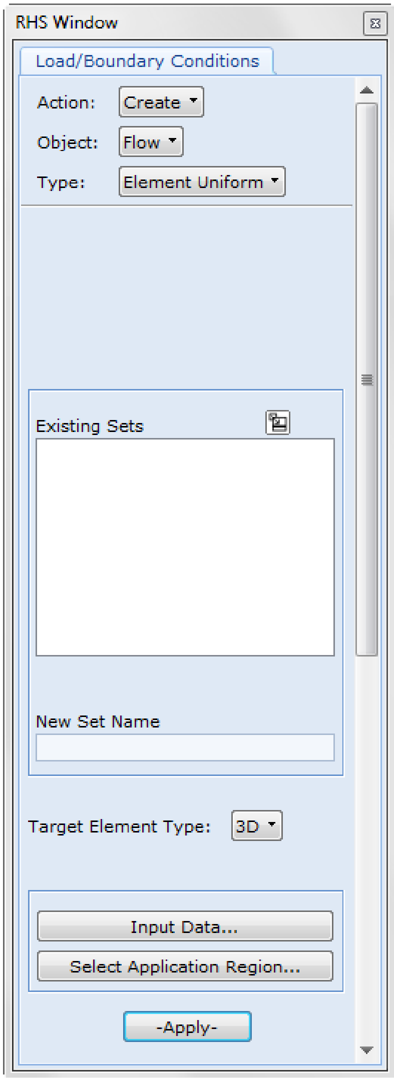


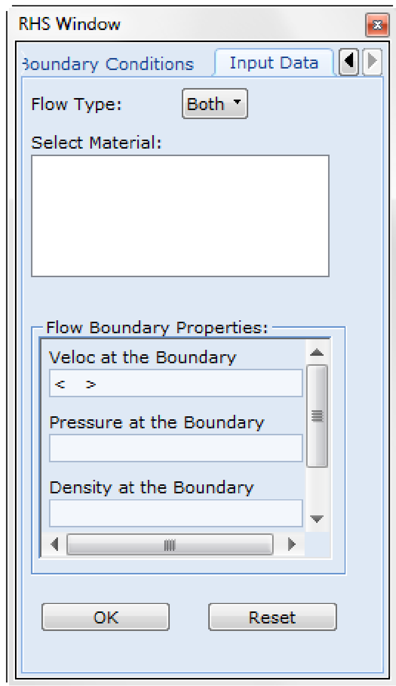


Parameter	Description
Level Indicator	Each combination of Shape and Initial Value set must have a unique Level Indicator in the table. For different Eulerian element property sets, the use of non-unique Level Indicators is allowed, as long as the Level Indicators are unique within one set.
Table Data: Shape	Each shape can only be used once in the table.
Preview	The Preview option will visualize the definition of the Eulerian region, as shown in the example below.

Flow

The Eulerian flow boundary defines the properties of the material at the boundaries of an Eulerian mesh. Any material properties at the boundaries not specifically defined will have the same value as the material in the element at the flow boundary during the analysis.


RHS Window
Load/Boundary Conditions
Action: Create
Object: Flow
Type: Element Uniform
Existing Sets
New Set Name
Target Element Type: 3D
Input Data...
Select Application Region...
-Apply-

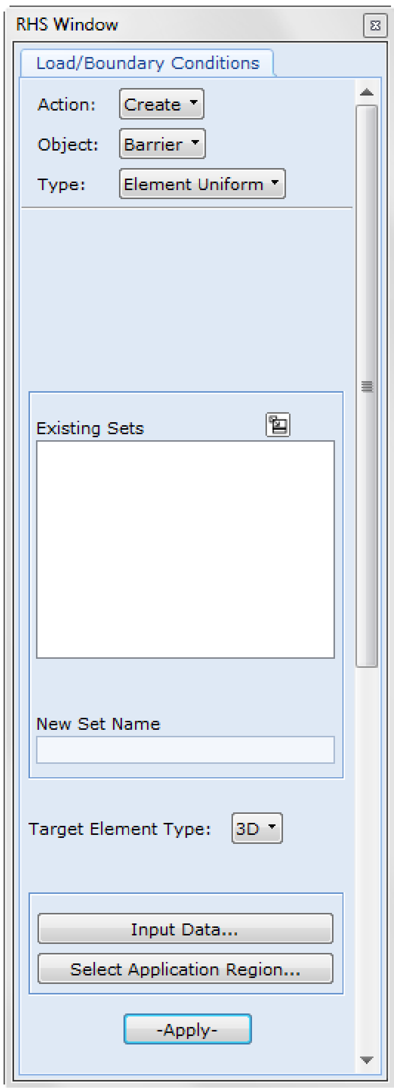

RHS Window
Boundary Conditions
Input Data
Flow Type: Both
Select Material:
Flow Boundary Properties:
Veloc at the Boundary
< >
Pressure at the Boundary
Density at the Boundary
OK
Reset

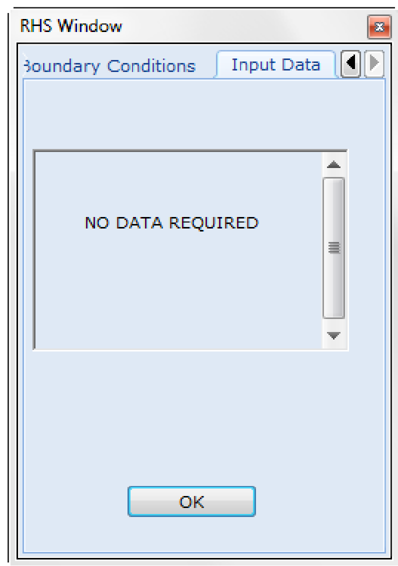
Parameter	Description
Flow Type	The Flow Type can be either: In Flow -inflow boundary Out Flow -outflow boundary Both -inflow or outflow boundary depending on the physical problem during the analysis.
Select Material	Only Eulerian materials that have been defined are shown.
Flow Boundary Properties	Material properties at the flow boundary that can be defined are: velocity, pressure, density and internal energy.

Note: In the case of multi-material flow into or out of a multi-material Eulerian mesh, the material flowing into or out of the mesh is assumed to be the same as in the elements adjacent to the boundary. For this material flow, only velocity and pressure are prescribed. Both the density and specific internal energy of the mixed materials are assumed to be the same as those of the mixed materials in the element adjacent to the boundary.

Barrier

A barrier defines a rigid wall in the Eulerian mesh, through which no material transport takes place. By default the exterior faces of an Eulerian mesh that do not have a FLOW boundary condition specified are barriers. However, the BARRIER boundary can be used to specify internal rigid walls in the Eulerian mesh.

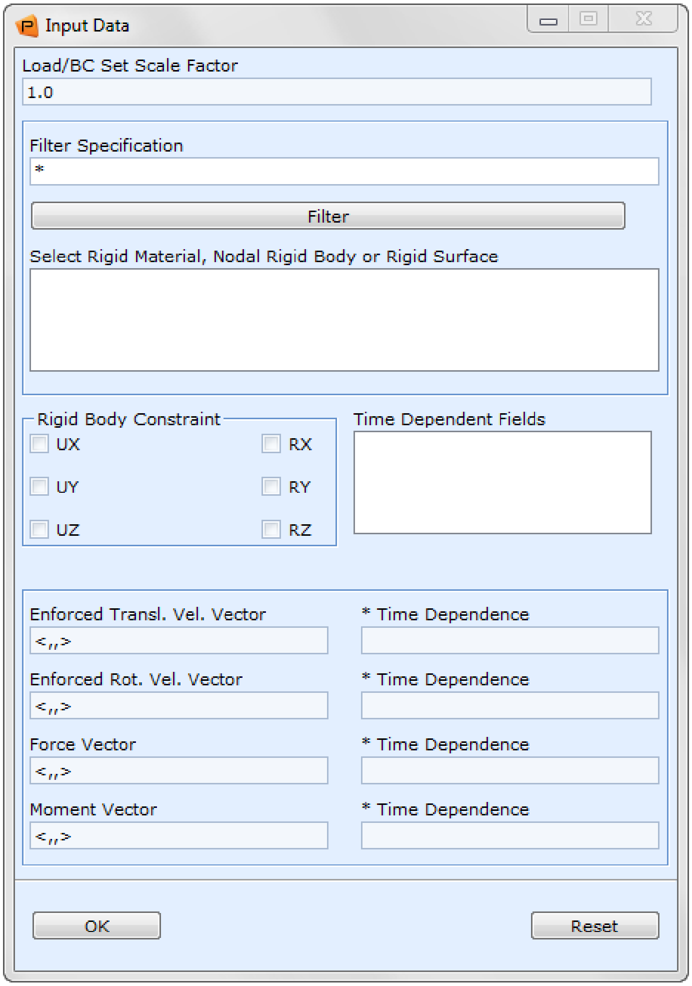


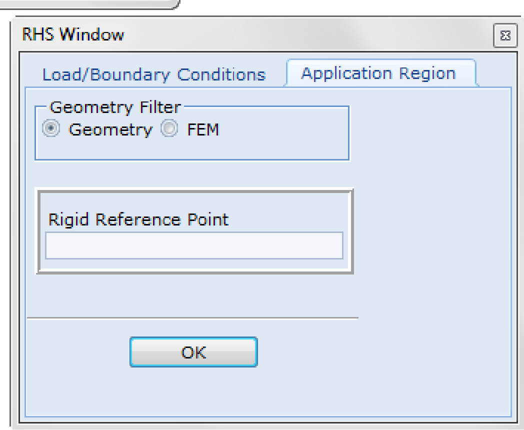


Rigid Body Object

In Dytran, rigid body can be constrained or have forces acting on their center of gravities. The Rigid Body Object LBC allows you to: (a) constrain the body, (b) specify a predefined velocity field, and (c) apply forces and moments on the center of gravity.

The vectors are visualized by using a reference node in the application region.

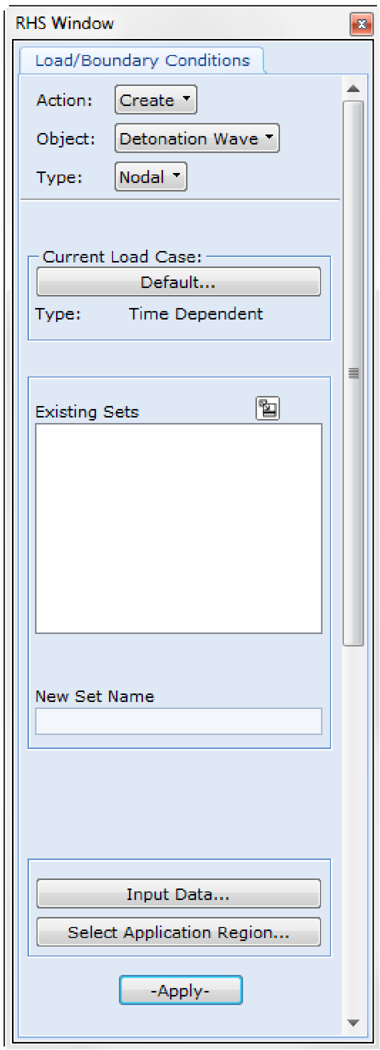

Input Data
Load/BC Set Scale Factor
1.0
Filter Specification
*
Filter
Select Rigid Material, Nodal Rigid Body or Rigid Surface
Rigid Body Constraint
UX
RX
UY
RY
UZ
RZ
Time Dependent Fields
Enforced Transl. Vel. Vector
<,,>
* Time Dependence
Enforced Rot. Vel. Vector
<,,>
* Time Dependence
Force Vector
<,,>
* Time Dependence
Moment Vector
<,,>
* Time Dependence
OK
Reset

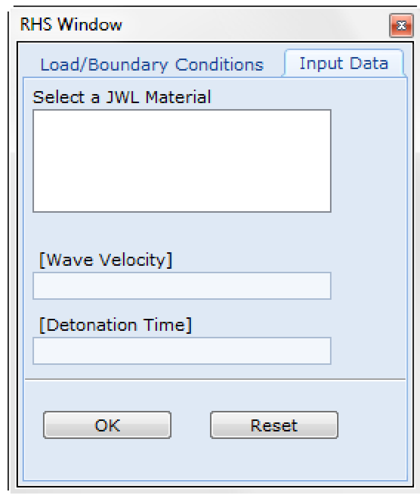

RHS Window
Load/Boundary Conditions
Application Region
Geometry Filter
Geometry
FEM
Rigid Reference Point
OK

Parameter	Description
Select Rigid Material, Nodal Rigid Body or Rigid Surfaces	List of existing MATRIG materials, Nodal Rigid Bodies, and Rigid Surfaces. Only one material or LBC name can be selected.
Time Dependent Fields	List of existing non-spatial fields.
Rigid Body Constraints	Toggles for constraining the rigid body on its center of gravity. When toggling on in one direction it will place a zero on the appropriate place in the velocity vector field. Any existing value will be overwritten. When toggling off, the value will be overwritten with a blank.
Rigid Reference Point	Only allowed is Point ID or Node ID. Any other input will be automatically erased. Also in case more than one entity is selected.

Detonation Wave

Defines a spherical detonation wave originating from the ignition point at the specified time. This is an initial boundary condition for detonation wave for a JWL material.



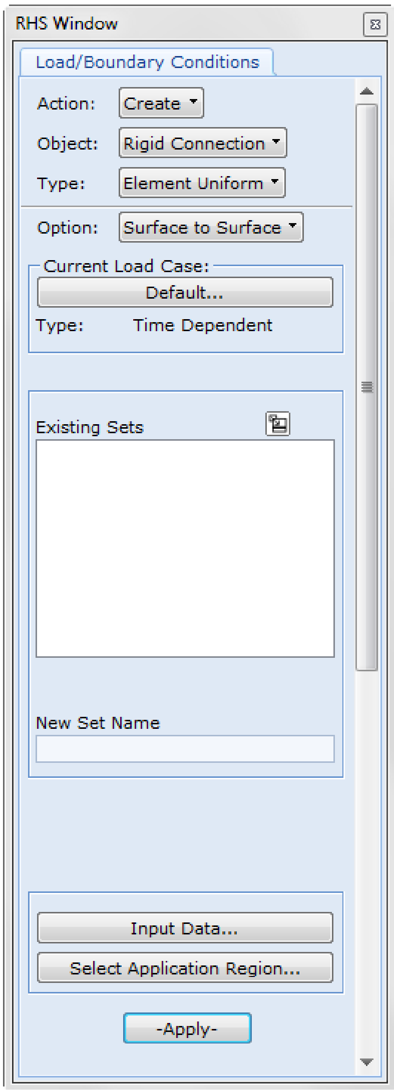


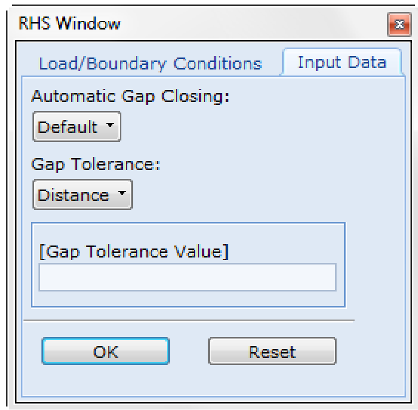
Parameter	Description
Select a JWL Material	Only JWL materials that have been defined are shown. Single material selection.
Wave Velocity	Velocity of the detonation wave and detonation time that are not specified are assumed to be zero.
Detonation Time	

Note:	Standard Application Region form for Nodal type is used (single nodal selection).

Rigid Connection

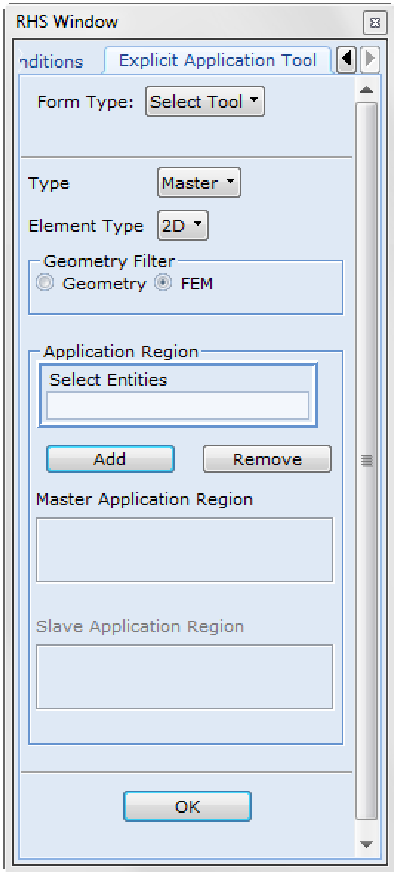
Defines a rigid connection between different parts of Lagrangian meshes. Three types of connections can be used: 1- Two Surfaces Tied Together; 2- Grid Points Tied to a Surface; 3- Shell Edge Tied to a Shell Surface.





Parameter	Description
Option	Types of Connection: -Surface to Surface -Edges to Surface -Nodes to Surface
Automatic Gap Closing	Choose between: -Default (default) -Yes -No
Gap Tolerance	Choose between: -Distance (default) -Factor Note: option Factor only available for Surface to Surface connection type
Gap Tolerance Value	Shown only if the Gap Tolerance is set to Distance. If the Gap Tolerance is set to Factor, it will show the label Gap Tolerance Factor instead.

Application Region forms for Rigid Connection



Parameter	Description
Form Type	Choose between: -Select Tool -Groups
Type	Choose between: -Master -Slave

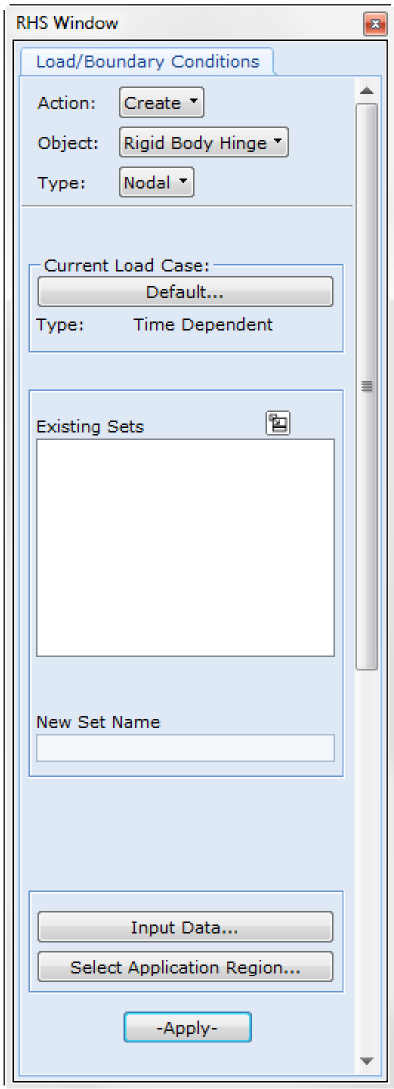
Parameter	Description
Element Type	For Master Type: -2D -3D For Slave Surface to Surface Type -2D -3D For Slave Edges to Surface Type -1D -2D For Slave Nodes to Surface Type -Nodal
Geometry Filter	This part of the form varies depending on the selection in the Form Type. Shown here is for type Select Tool.
Select Entities	
Mater Application Region	
Slave Application Region	

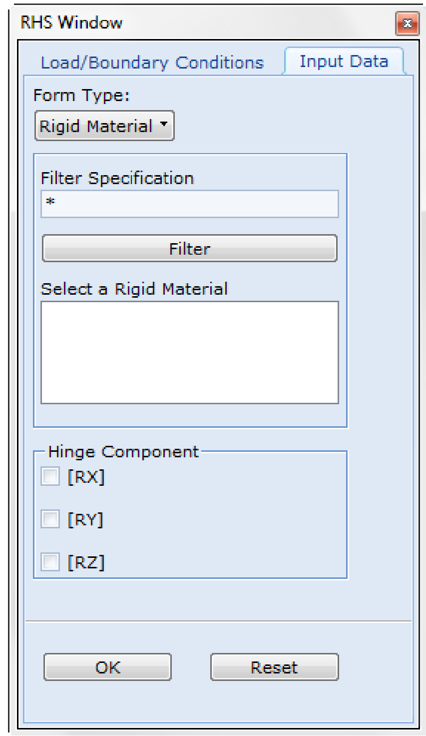
For Surface type, SURFACE entry is written. If the surface is 3D, CFACE entries are written, while if the surface is 2D, SET1 entries are written.

For 1D edges and nodes, only SET1 is also used.

Rigid Body Hinge

Defines a hinge between rigid body and a deformable structure.



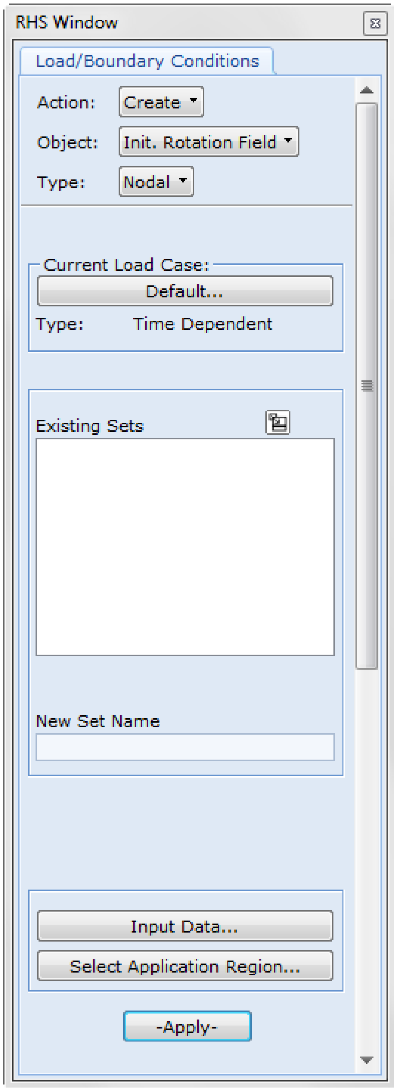


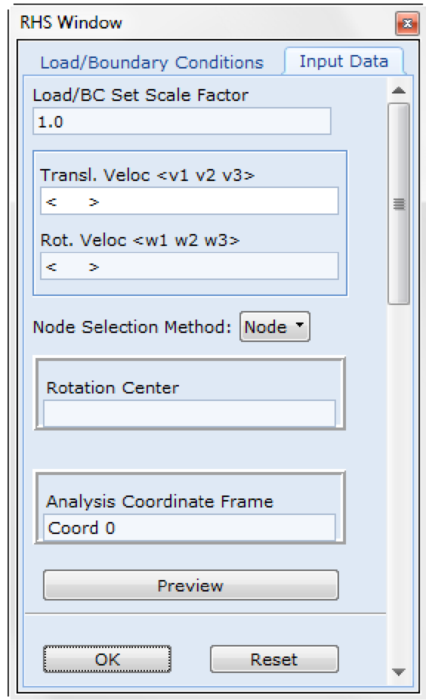
Parameter	Description
Form Type	Choose between: -Rigid Material (default) -Nodal Rigid Body
Select a Rigid Material	-For Rigid Material type: Only MATRIG materials that have been defined are shown. Single material selection. -For Nodal Rigid Body type: Only Nodal Rigid Body lbcs that have been defined are shown. Single lbc selection.
Hinge Component	Rotational components.

Note:	Standard Application Region form for Nodal type is used.

Initial Rotation Field

Defines a velocity field for a set of grid points consisting of a rotation and a translation specification.





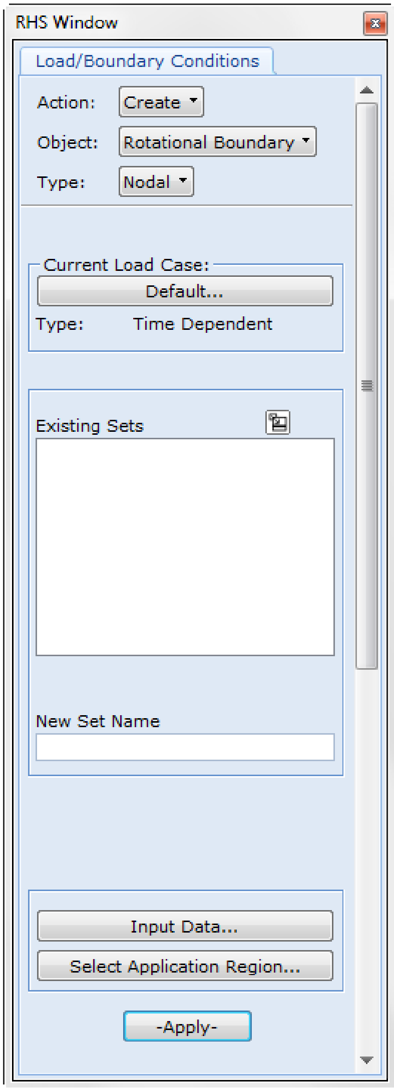
Parameter	Description
Transl. Veloc <v1 v2 v3>	Translational and rotational velocity vectors.
Rot. Veloc <w1 w2 w3>	

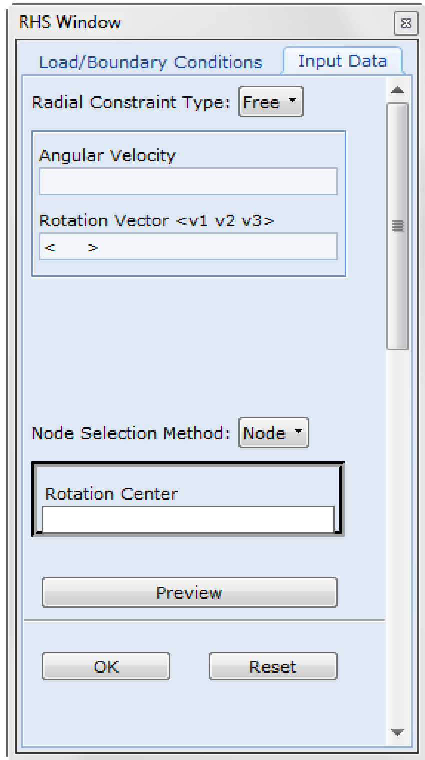
Parameter	Description
Node Selection Method	Choose between: -Node (default) -Point -Coord
Rotation Center	Center of rotation Node: If Coord or Point is selected as the method type, a new node is created in the model.
Preview	Shows the node selected with the translational and rotational vectors.

Note: Standard Application Region form for Nodal type is used.

Rotational Boundary

Defines a rotational boundary constraint on grid points.





Parameter	Description
Radial Constraint Type	Choose between: -Free (default) -Constraint
Angular Velocity	Rotational (angular) velocity and rotation vector.
Rotation Vector	

Parameter	Description
Node Selection Method	Choose between: -Node (default) -Point -Coord
Rotation Center	Center of rotation Note: If Coord or Point is selected as the method type, a new node is created in the model.
Preview	Shows the node selected with the rotational vector.

Coupling

Introduction

This section describes the user interface provided by Patran to model fluid-structure interaction between a structure and an Eulerian mesh. This interface is used during definition of the coupling LBC types: General Subsurface, Subsurface, General, with Failure, Interaction, ALE, ALE Grid1 and ALE Grid.

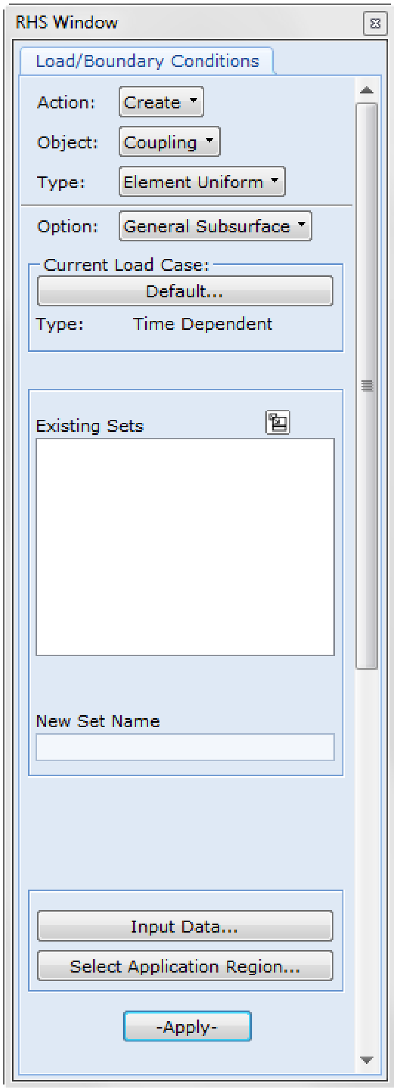
Tools have been provided to enable the user to quickly and easily define coupling conditions. Specification of coupling is conceptually simple, involving coupling surfaces, and a set of coupling parameters which control the interaction of the coupling surfaces with the Eulerian mesh. A fundamental assumption for the definition of a coupling surface is the concept that a coupling surface must be closed; this means that there are no holes in the coupling surface.

Coupling Types

A coupling condition in which a coupling surface is interacting with an arbitrary oriented HEX solid Eulerian mesh is described as General Coupling. Fluid-structure analysis using general coupling involve a large amount of geometrical intersection calculations during the analysis. To reduce the CPU time needed for these intersection calculations, a special coupling condition is available, which is described a Fast General Coupling. Fast general coupling can only be applied when the Eulerian solid HEX mash is aligned with the global coordinate frame. This means that all faces of the solid HEX elements are aligned with one of the coordinate planes of the global coordinate frame.

Coupling Construction

Tools are provided to enable the construction of coupling surfaces, using the standard Patran select tool mechanisms (2D elements, 3D element faces), or groups. Coupling subsurfaces can also be constructed using these tools, and later be used to define a complete logical coupling surface. This functionality allows the user to use the select tool to specify application regions on Patran geometry or the associated FEM entities or to define a more complex coupling surface that is assembled from a mixture of 2D and 3D element faces. Use of the group select mechanism is restricted to FEM entities only. Visualization of the specified coupling condition is provided by graphically previewing but is not currently supported for geometry entities.


RHS Window
Load/Boundary Conditions
Action: Create
Object: Coupling
Type: Element Uniform
Option: General Subsurface
Current Load Case:
Default...
Type: Time Dependent
Existing Sets
New Set Name
Input Data...
Select Application Region...
-Apply-

Parameter	Description
Option	Choose between: -General Subsurface -Subsurface -General -With Failure -Interaction -ALE -ALE Grid 1 -ALE Grid

"Simple" coupling surfaces include surfaces which may be described entirely by the faces of 3D elements, or by 2D elements. These coupling surfaces may be constructed entirely using a single select mechanism (either Select Tool or Group method). Simple coupling surfaces may not include a mixture of 3D element faces and 2D elements.

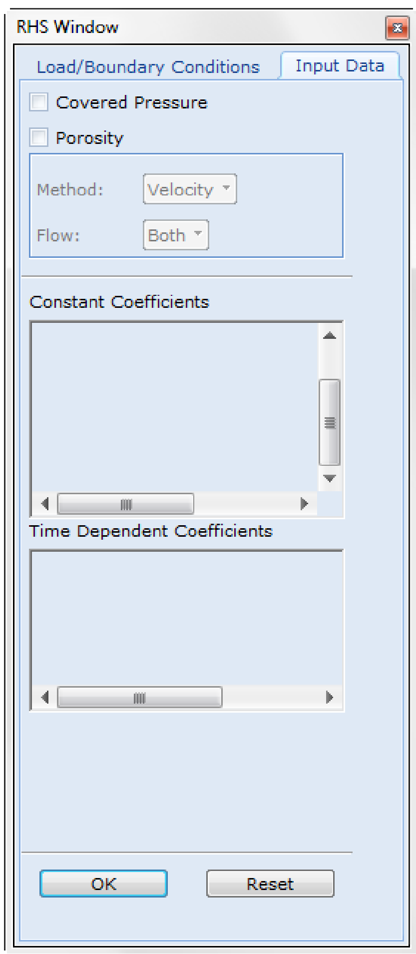
"Complex" coupling surfaces are defined as those surfaces which consist of a mixture of 2D elements and 3D element faces. Coupling conditions which include complex coupling surfaces must be constructed using "Subsurfaces," where each subsurface is a "Simple" coupling surface. The following sections describes how each of the coupling surface creation methods is used to describe a simple coupling surface.

Coupling: Input Data

The form is used to define a set of coupling parameters which control the interaction of the coupling surface with the Eulerian mesh. A coupling surface can consist of 2D elements, 3D element faces, or a mixture of 2D and 3D elements. The part of the structural mesh that is covered by the coupling surface depends on the direction of the normals of the faces of the coupling surface. The normals of the coupling surface must all be aligned in the same direction and can automatically be aligned in such a way that either all normal point outward or inward without modifying the underlying element topology. When all normals point outward, the inside of the coupling surface is covered. When all normals point inward, the outside of the coupling surface is covered.

Input data forms are provided for general subsurface, general coupling, coupling with failure, ALE GRID1 and ALE Grid.

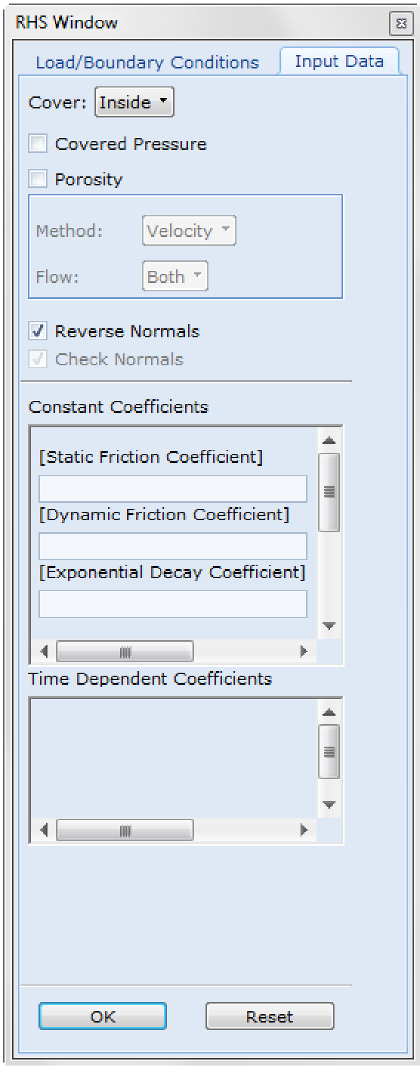
The following form is used for the input data of a general subsurface.



Parameter	Description
Method	Choose between: -Velocity (Default) -Pressure
Flow	Choose between: -Both (Default) -Out -In
Constant Coefficient: Additional Parameters	Additional parameters are: -Environmental Pressure -Environmental Density -Environmental Specific Energy -Flow Boundary Velocity
Time Dependent Coefficients: Additional Parameters	Additional parameters are: -Covered Pressure -Porosity Area Coefficient

Note: Titles grey out when porosity is inactive.

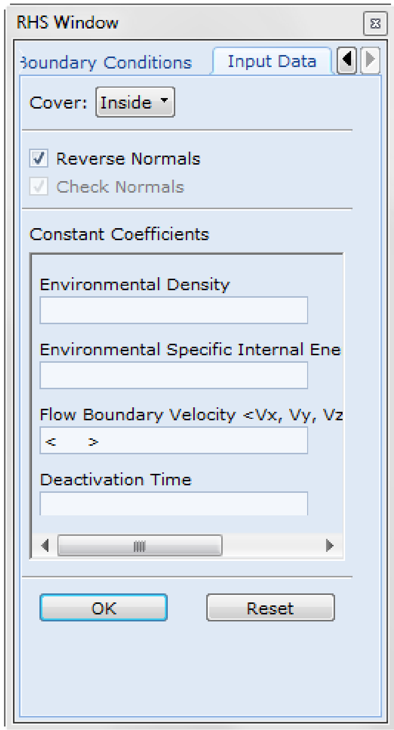
The following form is used for the input data of a general coupling.


RHS Window
Load/Boundary Conditions
Input Data
Cover: Inside
Covered Pressure
Porosity
Method: Velocity
Flow: Both
Reverse Normals
Check Normals
Constant Coefficients
[Static Friction Coefficient]
[Dynamic Friction Coefficient]
[Exponential Decay Coefficient]
Time Dependent Coefficients
OK
Reset

Parameter	Description
Cover	Choose between: -Inside (Default) -Outside
Method	Choose between: -Velocity (Default) -Pressure
Flow	Choose between: -Both (Default) -Out -In
Reverse Normals	If needed, the normals are reversed to obtain a positive coupling surface volume, depending on the selected cover option (inside/outside).
Check Normals	The normals of the faces of the coupling surface are checked to see whether they all point in the same direction and give a positive closed volume.
Constant Coefficient: Additional Parameters	Additional parameters are: -Environmental Pressure -Environmental Density -Environmental Specific Energy -Flow Boundary Velocity
Time Dependent Coefficients: Additional Parameters	Additional parameters are: -Covered Pressure -Porosity Area Coefficient

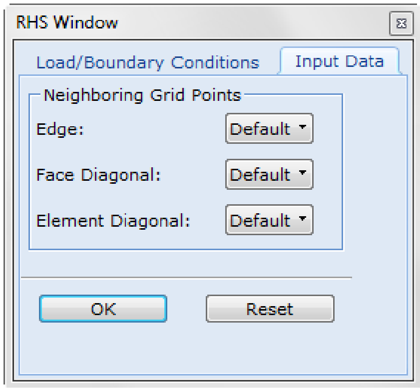
Note:	Titles grey out when porosity is inactive.

The following form is used for the input data of coupling with failure.



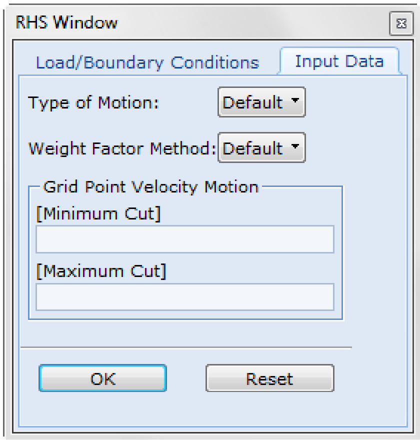
Parameter	Description
Cover	Choose between: -Inside (Default) -Outside
Reverse Normals	If needed, the normals are reversed to obtain a positive coupling surface volume, depending on the selected cover option (inside/outside).
Check Normal	The normals of the faces of the coupling surface are checked to see whether they all point in the same direction and give a positive closed volume.

The following form is used for the input data of ALE Grid 1.



Parameter	Description
Neighboring Grid Points	Choose between: -Default (default) -Yes -No

The following form is used for the input data of ALE Grid.



Parameter	Description
Type of Motion	Choose between: -Default -Standard -Free -Fixed -Flow -Special -User
Weight Factor Method	Choose between: -Default -Constant -Computed
Grid Point Velocity Motion	Shown only if the Type of Motion is not set to User. Parameters defining the minimum and maximum allowable velocity of ALE grip points.

Note:	For the option “User”, the preference sets the “NAME” field of the “ALEGRID” entry to ALEG_id where id is the identity number of the lbc.

Coupling: Application Region

This form is used to define coupling surfaces. The form will vary depending upon which options are selected; three basic configurations are used depending on whether the coupling surface is defined using the "Select Tool," "Group," or "Subsurface" option. Definition of coupling surfaces is limited to one method, i.e., it is not permissible to mix, "Select Tool," "Group," or "Subsurface" within the definition of a coupling surface.

Use of the Select Tool

The select tool is use to graphically select the desired entities from the model. When this method is selected, the user must specify which dimensionality the intended object has, i.e. 3D, or 2D.

It is not permissible to mix 3D and 2D entities within a single Application Region. (This functionality is provided through the use of coupling subsurfaces). The select tool can be used to select on the basis of either FEM or Geometry entities.

Use of the Group Tool

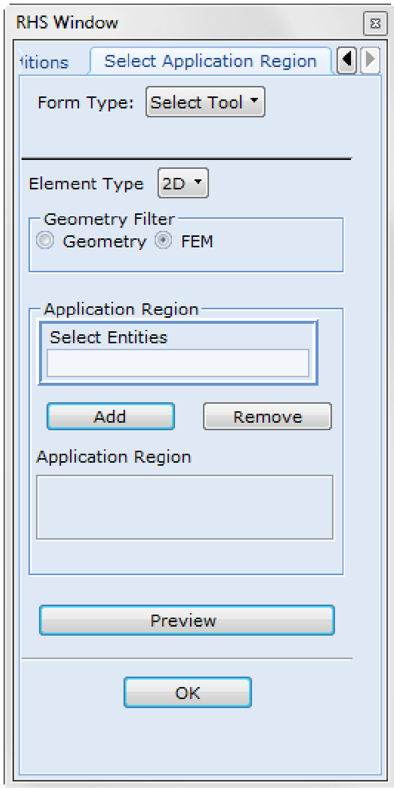
The Group tool is used to define simple coupling surfaces on the basis of Patran group names. When this method is selected, the user must specify which dimensionality the intended object has, i.e., either 3D, 2D, or Nodal. The entities which will be selected for use in the coupling surface in this case are either all 3D free surfaces in the group, or all 2D elements contained in the selected group. However, it should be noted that the selected element dimensionality must apply across all selected groups.

Use of the Subsurface Tool

Coupling Subsurfaces may be defined using either of the above methods. Subsurfaces may then be used in the specification of general coupling surfaces. When this option is used, the user may not specify element dimensionality since this information has already been defined during subsurface definition. As many subsurfaces as required may be selected to form the desired complex coupling subsurface.

Use of the Subsurface/Select Tool

Only available for coupling with failure. Coupling Subsurfaces may be defined using either of the two first methods. Subsurfaces may then be used in the specification of coupling with failure surfaces while Select Tool is used for the definition of the Euler Elements. When this option is used, the user may not specify element dimensionality since this information has already been defined during subsurface definition. As many subsurfaces as required may be selected to form the desired complex coupling subsurface.



Parameter	Description
Form Type	Choose between: -Select Tool -Group -Subsurface (only for general coupling) Note: The other options on this form varies depending on the selection above. Described below are the options for type Select Tool.
Element Type	Choose between: -2D -3D
Geometry Filter	Filter for picking Geometry or FEM entities.
Select Entities	Entity select databox. Entities appearing here may be added or removed from the active application region.

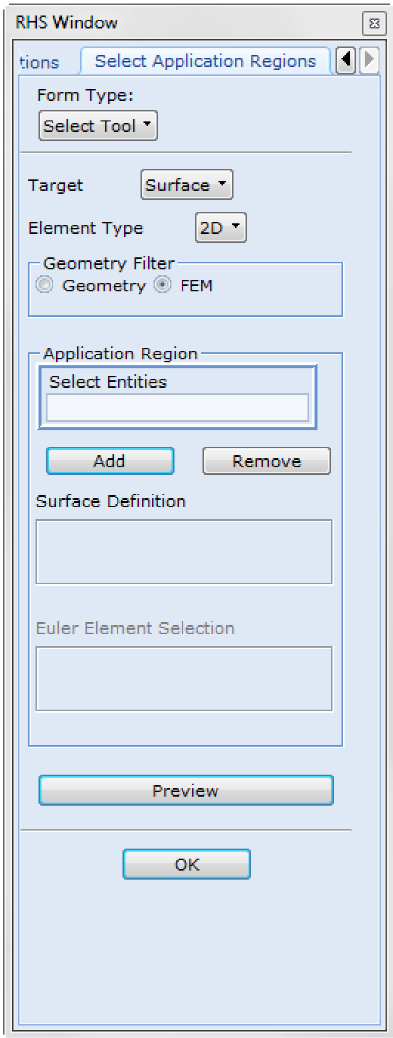
Parameter	Description
Application Region	List of entities in application region.
Preview	Preview coupling surface graphically.

Use of the Subsurface/Group Tool

Only available for coupling with failure. Coupling Subsurfaces may be defined using either of the two first methods. Subsurfaces may then be used in the specification of coupling with failure surfaces while Group Tool is used for the definition of the Euler Elements. When this option is used, the user may not specify element dimensionality since this information has already been defined during subsurface definition. As many subsurfaces as required may be selected to form the desired complex coupling subsurface.

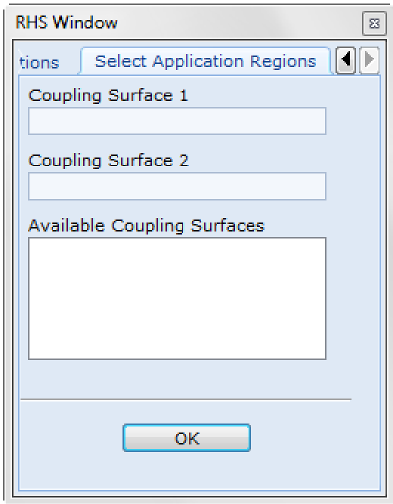
The application region form varies depending on the selection of coupling type.

The form used for the application region of general subsurface, subsurface, and general coupling appears above. The following form is used for the application region of coupling with failure.


RHS Window
tions
Select Application Regions
Form Type:
Select Tool
Target
Surface
Element Type
2D
Geometry Filter
Geometry
FEM
Application Region
Select Entities
Add
Remove
Surface Definition
Euler Element Selection
Preview
OK

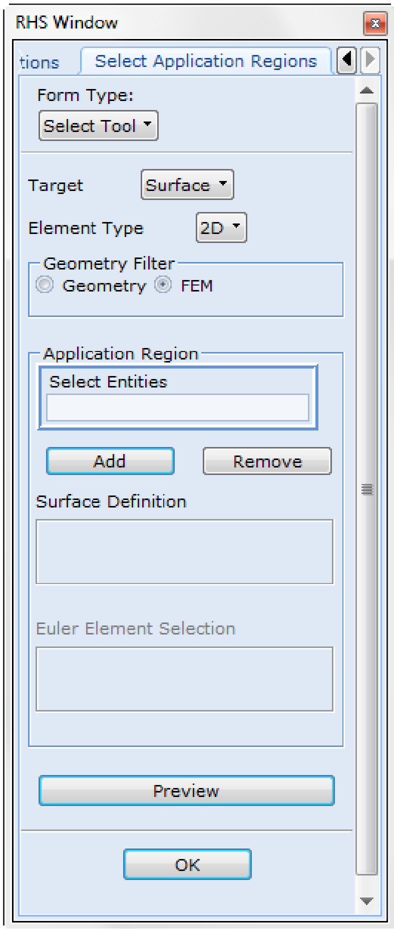
Parameter	Description
Form Type	Choose between: -Select Tool (Default) -Groups -Subsurface/Select Tool -Subsurface/Groups Note: The other options on this form varies depending on the selection above. Described below are the options for type Select Tool.
Target	Choose between: -Surface -Euler Element
Element Type	Choose between: -2D (for Surface only) -3D
Geometry Filter	Filter for picking Geometry or FEM entities.
Select Entities	Entity select databox. Entities appearing here may be added or removed from the active application region.
Surface Definition	List of entities in application region.
Euler Element Selection	
Preview	Preview coupling surface graphically.

The following form is used for the application region of interaction.



Parameter	Description
Available Coupling Surface	Lists all couplings with failure surfaces.

The following form is used for the application region of ALE.



Parameter	Description
Form Type	-Select Tool (only option).
Target	Choose between: -Surface -Euler Element
Element Type	Choose between: -2D (for Surface only) -3D
Preview	Preview coupling surface graphically.

Note:	The preview button checks if the grids of the Surface and the Euler element have one-to-one correspondence by using a tolerance of 1/20th of the selected element size. If not a warning message is given. The same check is carried out when the apply button is hit. If the check fails, the lbc will not be created.

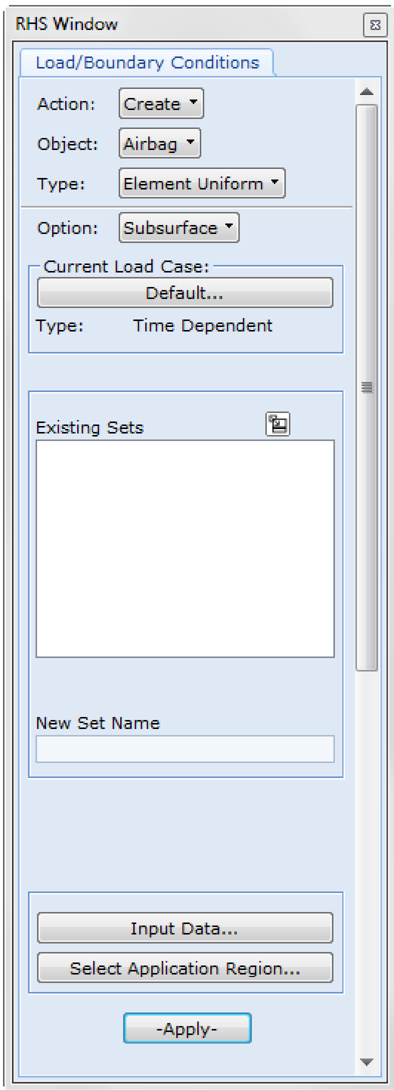
The Application Region forms for both ALE Grid 1 and ALE Grid are the standard form for nodal type.

You cannot have ALE Grid 1 and ALE Grid in the same analysis model. An error message is given when creating an ALE Grid lbc if an ALE Grid 1 lbc already exists and vice-versa.

Airbag

Introduction

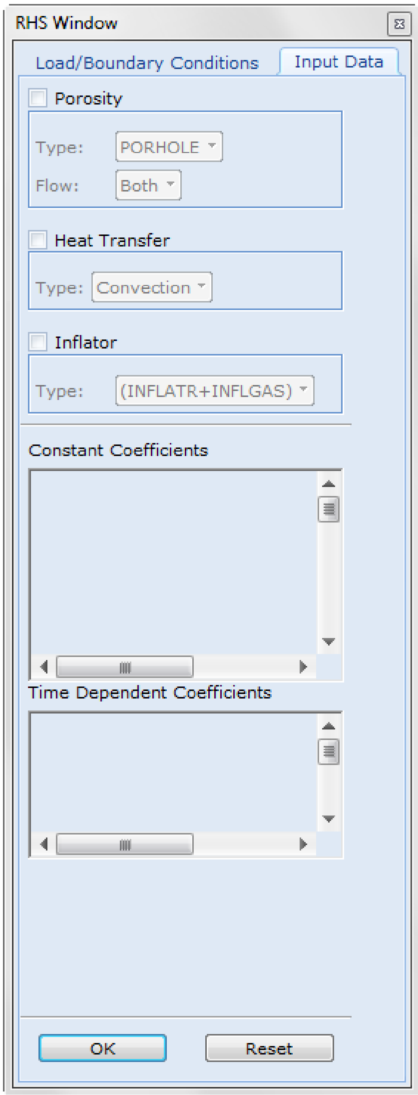
This section describes the user interface provided by Patran to model fluid-structure interaction especially for the air bags. Porosity, inflator and heat transfer can be defined in Air Bags. This interface is used during definition of the airbag LBC types: Subsurface and Surface.



Parameter	Description
Option	Choose between: -Subsurface -Surface

Airbag: Input data

Input data forms are provided for both subsurface and surface.The following form is used for the input data of a subsurface.

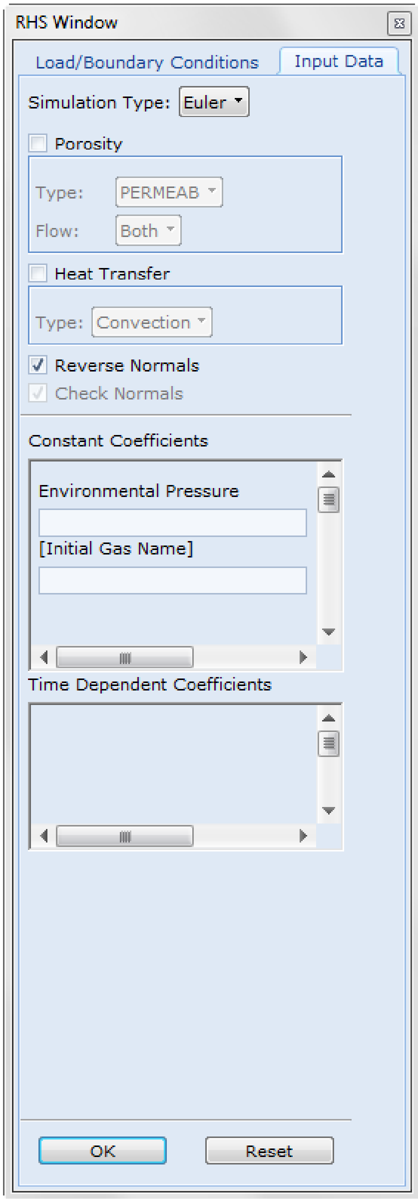


Parameter	Description
Porosity	
Type	Choose between: PORHOLE (Default) / PERMEAB / PORLHOLE / PERMGBG / PORFSCPL / PQRFLGBG

Parameter	Description
Flow	Choose between: -Both (Default) -Out -In Note: Titles grey out when porosity is inactive.
Heat Transfer: Type	Choose between: -Convection (Default) -Radiation -Convection and Radiation Note: Title greys out when heat transfer is inactive.
Inflator: Type	Choose between: Simple Dyn.(INFLATR) (DEF.) / (INFLATR+INFLGAS) / Simple Stat. (INFLATR1) / (INFLATR1+INFLGAS) / Hybrid Dyn. (INFLHYB) / Hybrid Stat. (INFLHYB1) / Tanktest Temp (INFLTANK) / Tanktest Pres (INFLTANK) Note: Note: Title greys out when Inflator is inactive.
Constant Coefficients: Additional Parameters	Additional parameters are: -Environmental Pressure/Density/Specific Internal Energy/Specific Heat/Temperature -Stephan-Boltzman Constant -Airbag Surface Name -Inflator Gas Name/Gas Fraction Name/Gamma/Gas Constant Cv/Gas Constant R/Gas Constant Cp/Tank Volume/Gas Mass -etc...
Time Dependent Coefficients: Additional Parameters	Additional parameters are: -Porosity Area Coefficient -Permeability Value -Heat Convection Transfer Area Coeff. -Convection Coefficient -Heat Radiation Transfer Area Coeff. -Gas Emissivity Coefficient -Inflator Area coeff. -Mass flow Rate Table -Inflator Gas Temperature -Tank Pressure -Inflator Pressure

Note:	If Porosity is On, Inflator is switched Off and if Inflator is On, Porosity is switched Off.

The following form is used for the input data of a surface

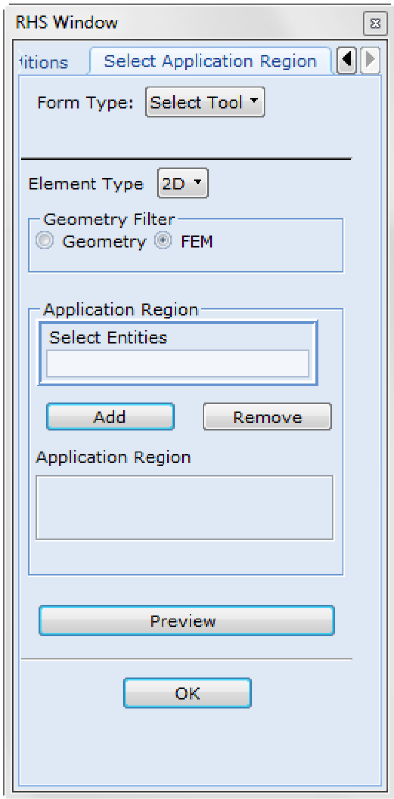


Parameter	Description
Uniform pressure	Choose between: -Euler (Default) -Uniform Pressure -Switch
Porosity	Note: Titles grey out when porosity is inactive.
Type	Choose between: -PERMEAB (Default) -PORHOLE -PORLHOLE
Flow	Choose between: -Both (Default) -Out -In
Heat Transfer	Note: Title greys out when heat transfer is inactive.
Type	Choose between: -Convection (Default) -Radiation -Convection and Radiation
Constant Coefficient: Additional Parameters	Additional parameters are: -Specific Heat Constant Cp -Gas Constant R -Euler to Unif. Press. Switch Time -Validity Check Percentage -Environmental Density -Environmental Specific Internal Energy -Environmental Specific Heat -Environmental Temperature -Stephan-Boltzman Constant
Time Dependent Coefficients: Additional Parameters	Additional parameters are: -Porosity Area Coefficient -Permeability Value -Heat Convection Transfer Area Coeff. -Convection Coefficient -Heat Radiation Transfer Area Coeff. -Gas Emissivity Coefficient

Airbag: Application Region

This form is used to define airbag surfaces. The form will vary depending upon which options are selected.

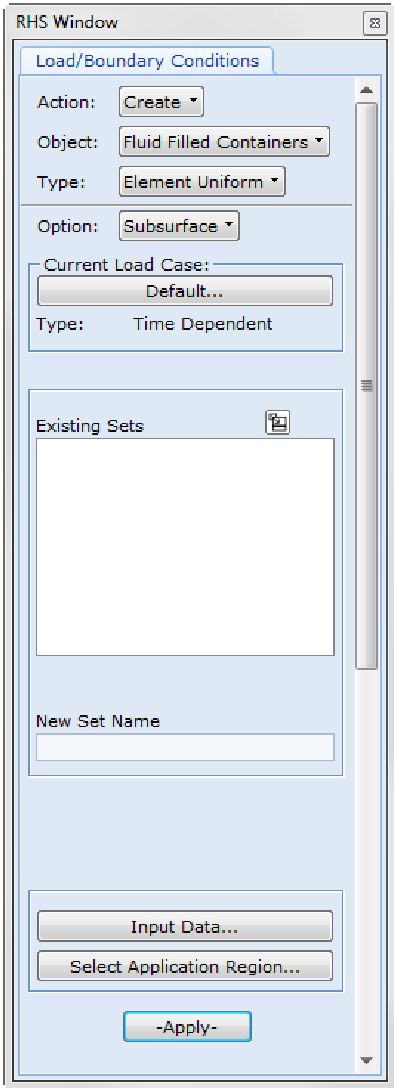
The following form is used for the application region of both surface and subsurface.

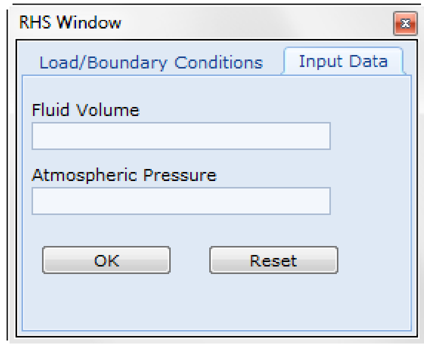


Parameter	Description
Form Type	Choose between: -Select Tool -Group -Subsurface (only for surface) Note: The other options on this form varies depending on the selection above. Described below are the options for type Select Tool.
Element Type	Choose between: -2D -3D
Preview	Preview airbag surface graphically.

Fluid Filled Containers

Defines the pressure within a closed volume in the Eulerian mesh. Intended for the use of (partially) filled containers.

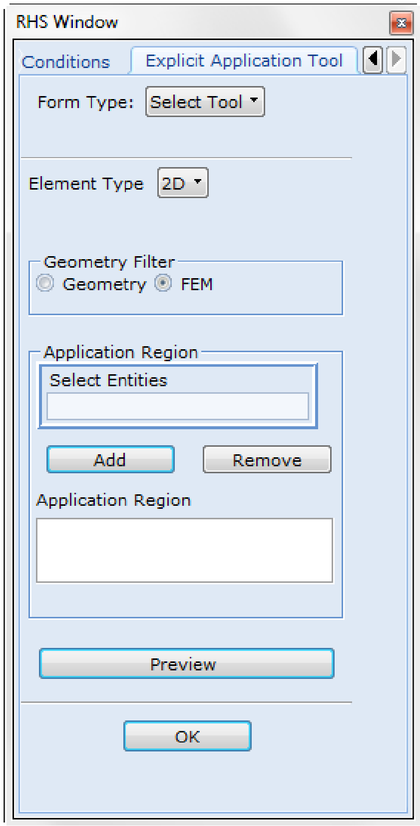




Parameter	Description
Option	Choose between: -Subsurface -Surface
Fluid Volume	Fluid Volume in the container.
Atmospheric Pressure	Atmospheric Pressure.

Note:	There is no Input Data form for the option “Subsurface”. Only for the option “Surface”.

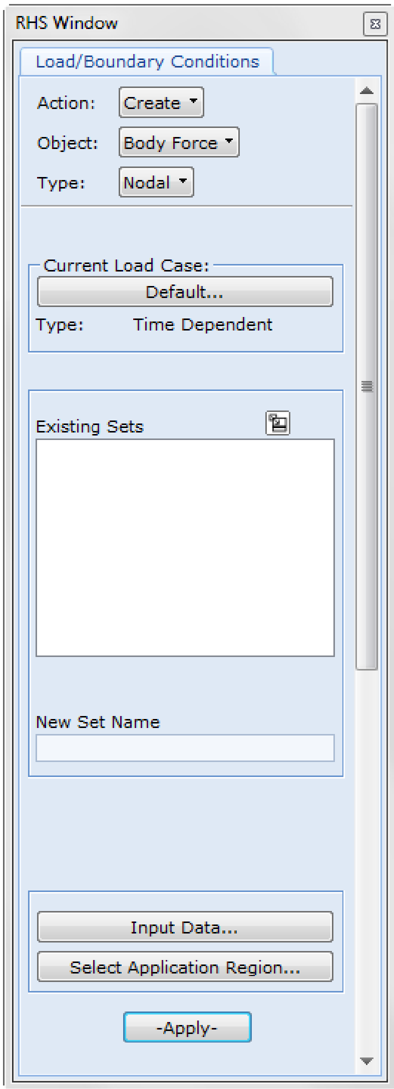
Application Region form for Fluid Filled Containers for both surface and subsurface options.



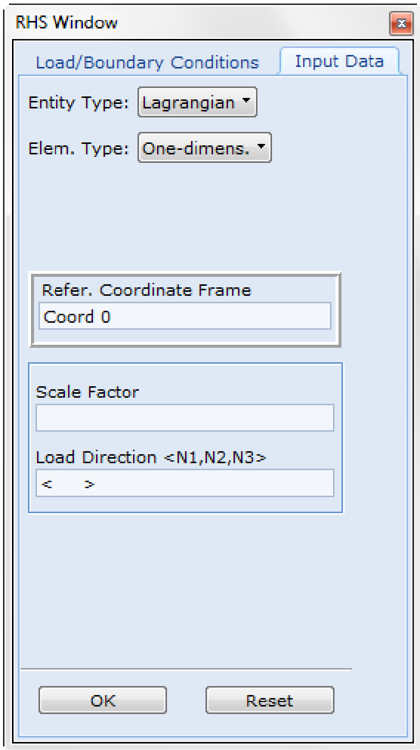
Parameter	Description
Form Type	Choose between: -Select Tool -Group -Subsurface (only for surface) Note: The other options on this form varies depending on the selection above. Described below are the options for type Select Tool.
Element Type	Choose between: -2D -3D
Preview	Preview Fluid Filled Container surface graphically.

Body Force

Defines a body force loading.



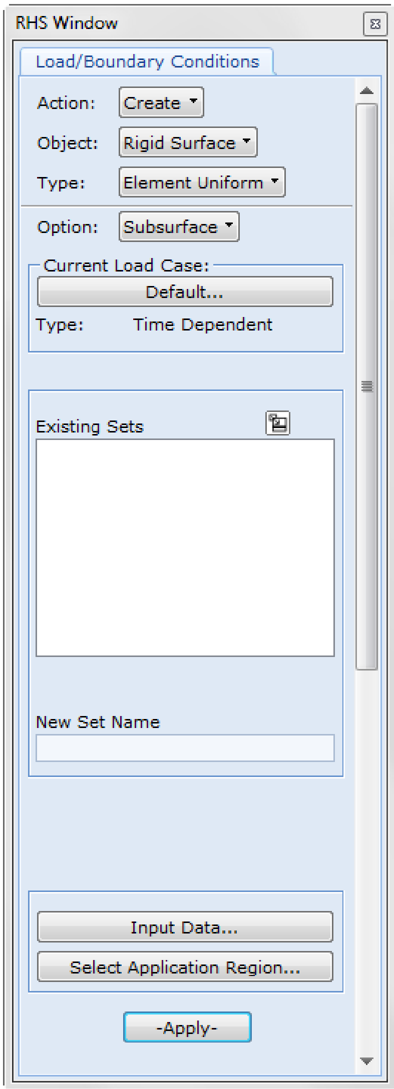
Note:	Standard Application Region form for Nodal type is used only if the option Grid is selected in the Entity Type. Otherwise, no application region is used.



Parameter	Description
Entity Type	Choose between: -Lagrangian -Eulerian -Ellipsoid -Grid Note: The other options on this form varies depending on the selection above. Described below are the options in Entity Type.
Scale Factor	Scale Factor can be either a constant value or a tabular field.
Load Direction	At least one component of the Load Direction should be non-zero.

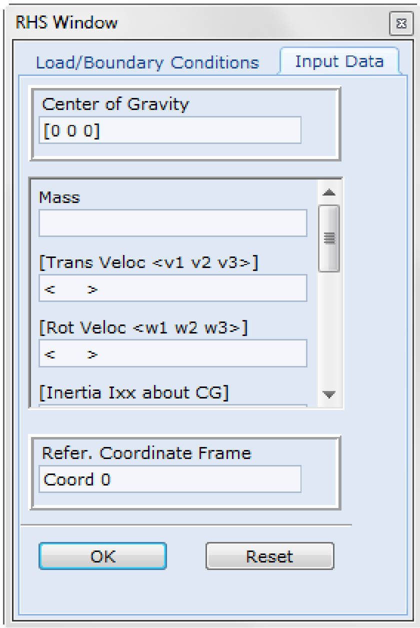
Rigid Surface

Defines a rigid surface.



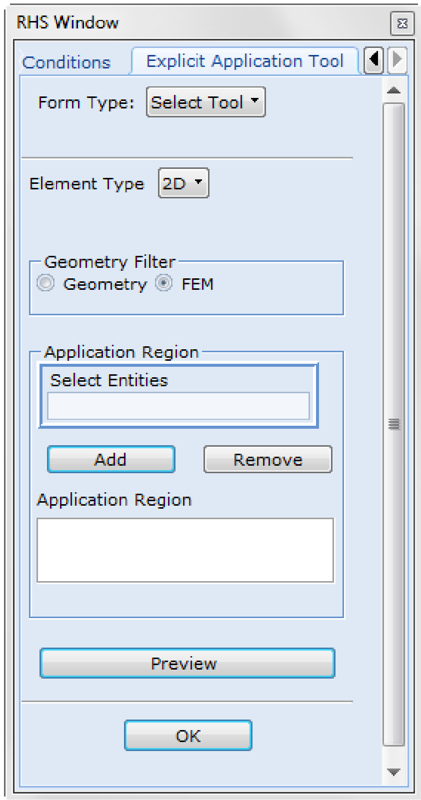
Parameter	Description
Option	Choose between: -Subsurface -Surface

Note: There is no Input Data form for the option "Subsurface". Only for the option "Surface".



Parameter	Description
Input Data: Additional Parameters	-Inertia Ixx about CG -Inertia Ixy about CG -Inertia Ixz about CG -Inertia Iyy about CG -Inertia Iyz about CG -Inertia Izz about CG

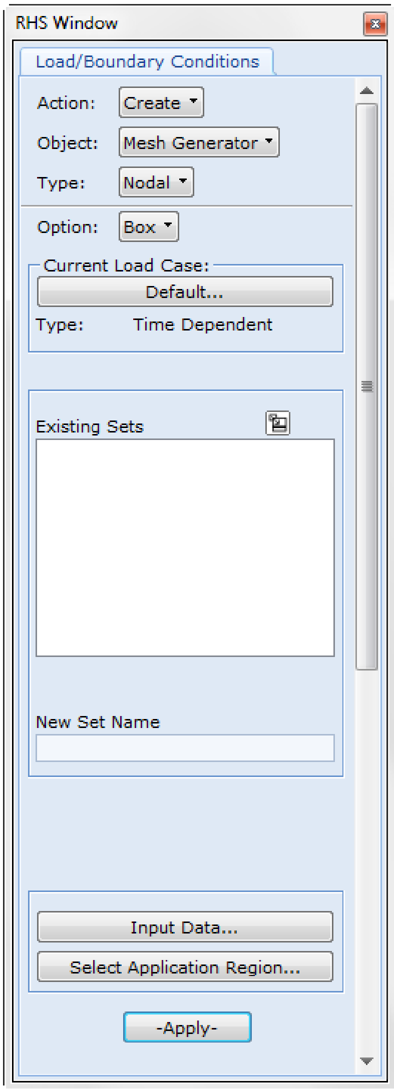
Application Region form for Rigid Surface for both surface and subsurface options.



Parameter	Description
Form Type	Choose between: -Select Tool -Group -Subsurface (only for surface) Note: The other options on this form varies depending on the selection above. Described below are the options for type Select Tool.
Element Type	Choose between: -2D -3D
Preview	Preview Rigid Surface graphically.

Mesh Generator

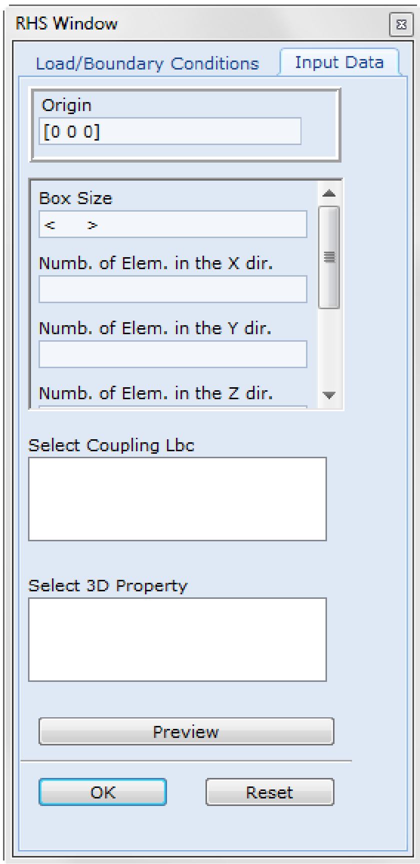
Defines a rigid surface.



Parameter	Description
Option	Choose between: -Box -Adaptive

Note:	There is no Application region for Mesh Generator.

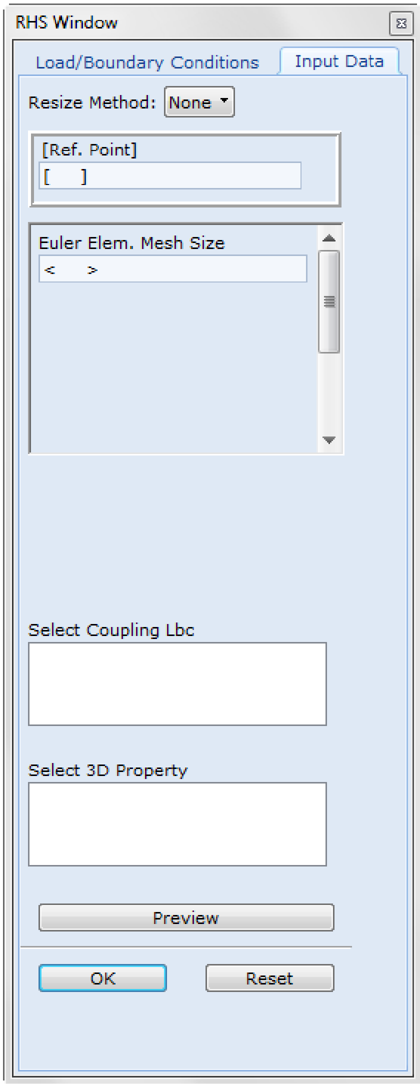
The following form is used for the input data of Mesh Box.



Parameter	Description
Box Size	Choose between: -Box -Adaptive
Select Coupling Lbc	Single selection of General Coupling lbc or Airbag lbc.
Select 3D Property	Single selection of Lagrangian or Eulerian property.

Parameter	Description
Preview	Preview the box mesh graphically.
Additional Parameters	-Numb. of Elem. in the Z dir. -Starting Node Id -Starting Elem. Id

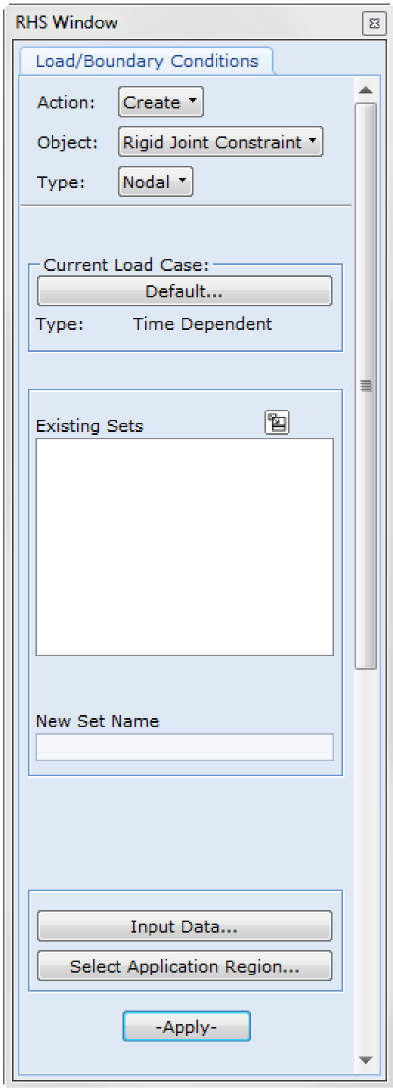
The following form is used for the input data of Mesh Adaptive.



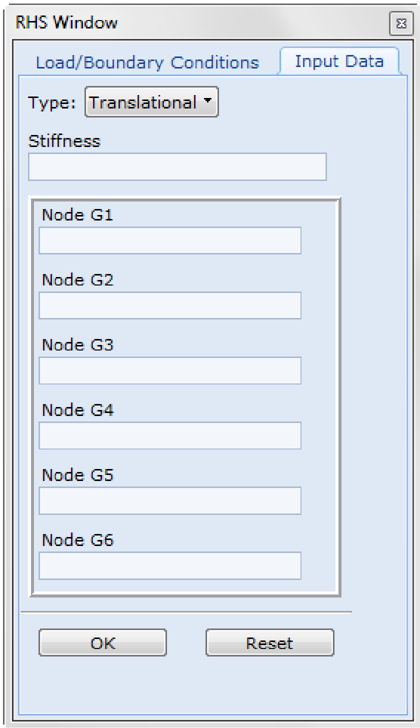
Parameter	Description
Resize Method	Choose between: -None -Scale -Length Note: For Scale or Length options, other parameters are: -Resize in the X direction. -Resize in the Y direction. -Resize in the Z direction.
Select Coupling Lbc	Single selection of General Coupling lbc or Euler/Switch Airbag lbc.
Select 3D Property	Single selection of Eulerian property.

Rigid Joint Constraint

Defines a rigid joint constraint



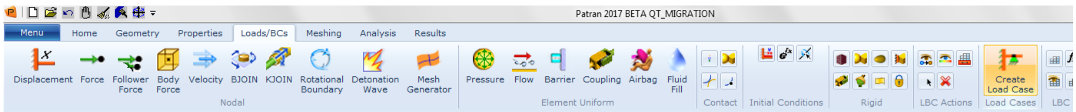
Note: There is no Application region for Rigid Joint Constraint.



Parameter	Description
Type	Choose between: -Cylindrical (default) -Planar -Revolute -Ellipsoid -Spherical -Translational -Universal
Node G3 **Node G4**	Available for all the options but Spherical.
Node G5 **Node G6**	Available only for the option Translational.

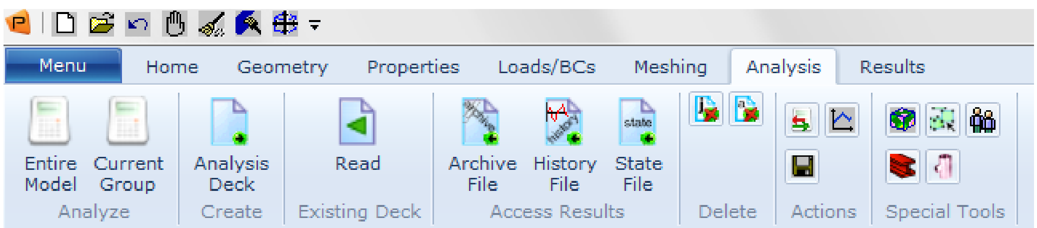
Load Cases

Load cases in Patran are used to group a series of load sets into one load environment for the model. Load cases are selected when preparing an analysis, not load sets. The usage for Dytran is consistent, however only one loadcase can be selected for translation. For information on how to define static and/or transient load cases, see Overview of the Load Cases Application (Ch. 5) in the *Patran Reference Manual*.



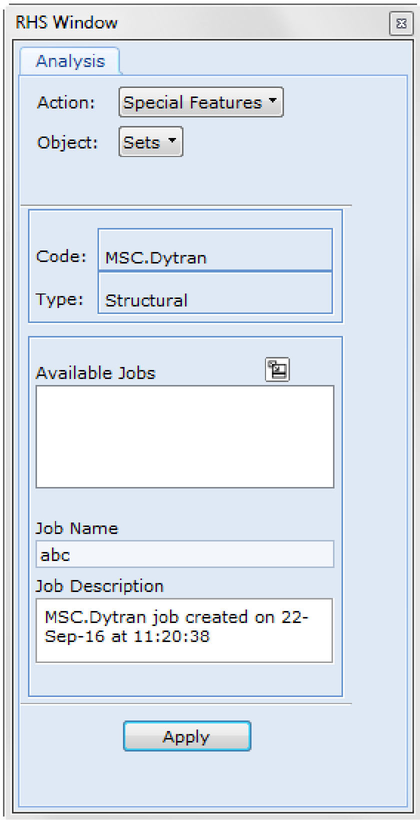
Special Features

Two special features are supplied to facilitate the use of Patran in conjunction with Dytran. These allow the user to create sets of nodes or elements to be written out in the Dytran input file as SET1 entries and a dummy positioner. These special features are accessed via the Analysis form which appears when the Analysis toggle, located on the Patran main form, is chosen. These forms supporting this functionality are described on the following pages.



Analysis Form

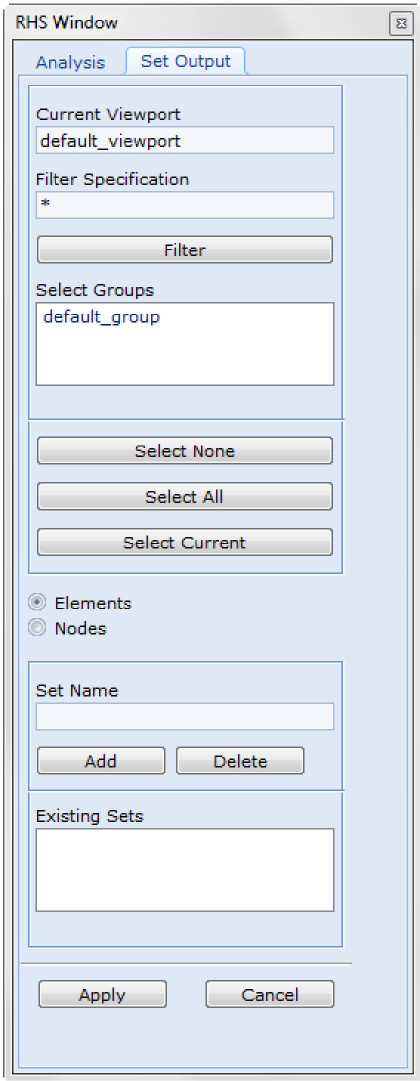
This form appears when the Analysis toggle is chosen on the main form. To utilize the special features, select Special Features as the action on the Analysis form. Select either “Sets”, “Dummy Positioner”, “Beam Post Processing”, or “Spotweld/Stiffener Tool” as the “Object.”



Parameter	Description
Action	Select the action “Special Features”.
Object	Options for Object are: -Sets -Dummy Positioner -Beam Post Processing -Spotweld/Stiffener Tool
Apply	Subordinate forms associated with the “Sets”, “Dummy Positioner”, “Beam Post Processing”, and “Spotweld/Stiffener Tool” are accessed by pressing “Apply”.

Set Creation

The subordinate form illustrated below appears if the “Object” is “Sets.” It is used to create a list of elements or nodes that is subsequently to be written to the Dytran input file as a SET1 Bulk Data entry. The “Set” is assembled from existing Patran groups. New groups are not created.



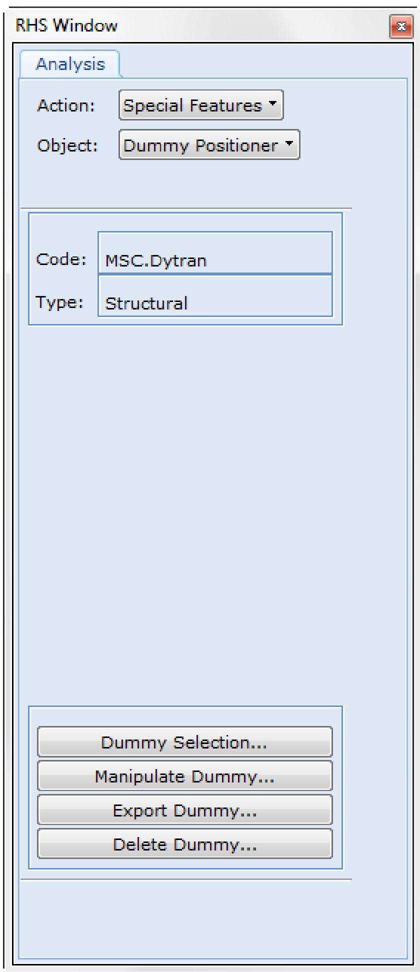
Parameter	Description
Current Viewport	These facilitate selection of the groups for which data is to be output. Set the viewport so it contains the groups of entities of interest. The filter is useful if a consistent group naming convention is used.
Filter Specification	

Parameter	Description
Select Groups	Lists the groups when the filter * is used so all groups in the current viewport are listed. Click on those required.
Select None	These exist to aid selection of the required groups of entities.
Select All	
Select Current	
Elements	Either element or node sets may be created.
Nodes	
Existing Sets	A list of existing sets. The set named in the “Set Name” box will be added to the list, replacing one of the same name. To delete an existing set pick the name from the existing set list.

Dummy Positioning

The purpose of the dummy positioner is to allow a user to import a standard finite element representation of a dummy so it can be correctly positioned within the Patran model. The dummy is not part of the finite element model as it is exported prior to creation of the Dytran input file. The exported file defining the dummy can be used in an ATB calculation run in parallel to Dytran simulation. Note that the positioner only works with the standard ATB dummy that is part of the standard Patran deliverable. ATB hybridII and hybrid III dummy files are included in the <installation_directory>/mscdytran_files directory.

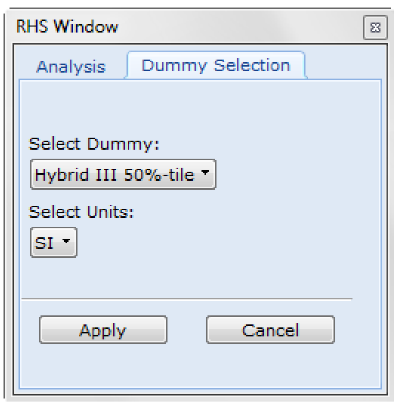
In v2002, several modification have been carried out in the dummy positioner to make it more user friendly.The Main "Dummy Positioning" form provides access to subordinate forms that provide for import, creation, manipulation, and export of an input file containing a finite element representation of a standard dummy.



Parameter	Description
Dummy Selection	Selects the standard ATB dummy model.
Manipulate Dummy	Manipulates the ATB dummy model. All the logical sequence of activities involved in positioning the dummy are stored in the Patran database and recorded in the session file.
Export Dummy	The dummy data set will be written out to a .dat file by clicking on OK on the Export Dummy form. The dummy should then be deleted.
Delete Dummy	Deletes the dummy from the database.

Dummy Selection

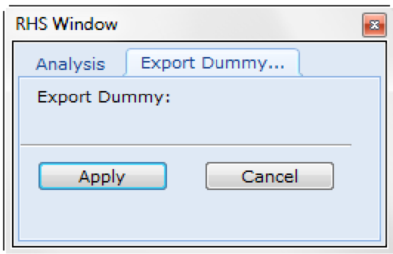
This form appears when "Dummy Selection..." is selected.



Parameter	Description
Select Dummy	Choose between: -Hybrid III 50% - tile -Hybrid III 5% - tile -Hybrid III 95% - tile -Hybrid II 5% - tile
Select Units	Choose between: -SI -English (only for 50% - tile options)

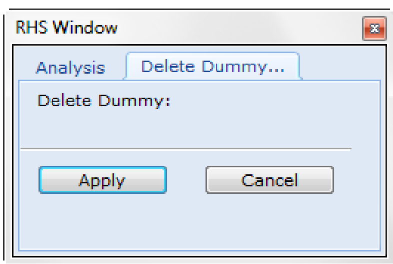
Export Dummy

This form appears when "Export Dummy..." is selected. It is under control of this form that a new input file is written in the working directory. This new file will be exactly the same as the "Master" model selected from the Dummy Selection form but with new grid points coordinates.



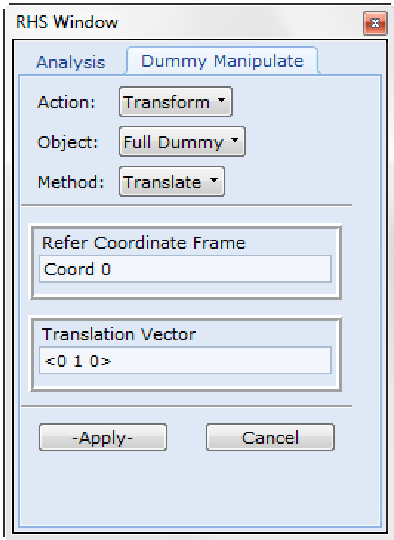
Delete Dummy

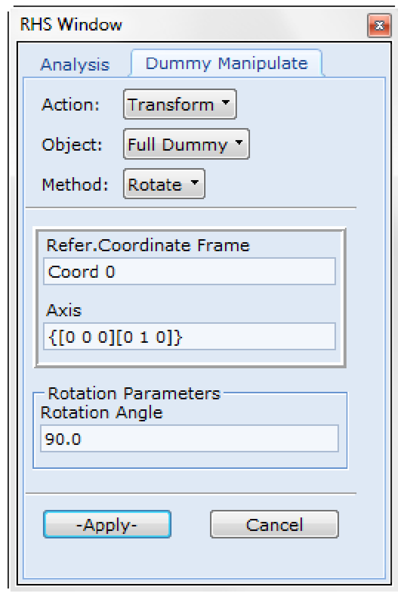
This form appears when "Delete Dummy ..." is selected. It is under control of this form that the dummy model is removed from the database.



Manipulate Dummy

These subordinate forms appear when "Manipulate Dummy..." and "Full Dummy" are selected

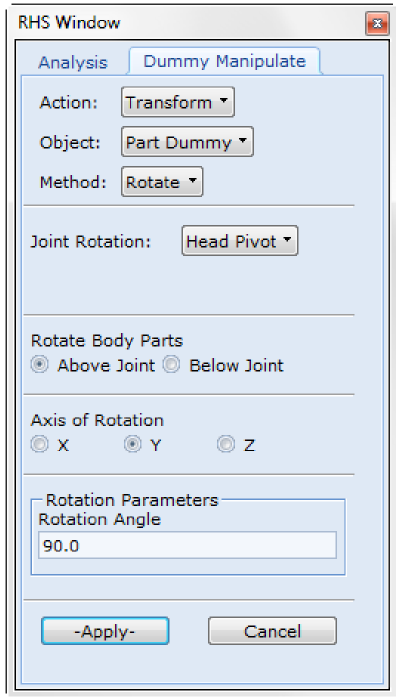




Parameter	Description
Action	Only option: Transform.
Object	Choose between: -Full Dummy -Part Dummy
Method	Choose between: -Translate (only for Full Dummy option) -Rotate
Refer. Coordinate Frame: Translate	Select the coordinate system for the translation. The default is the global system.

Parameter	Description
Translation Vector	Define the translation in the selected coordinate system, using global model units.
Refer. Coordinate Frame: Rotate	Select the coordinate system for the rotation. The default is the global system.
Axis	Select the axis of rotation. The default is {[0 0 0] [0 1 0]}.
Rotation Angle	Define the angle of rotation. The default is 90.0

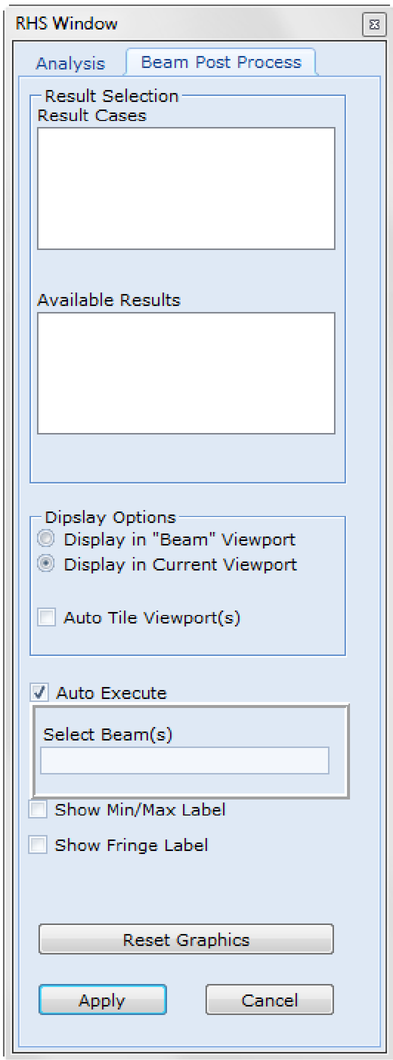
This form appears when "Manipulate Dummy..." and "Part Dummy" are selected.



Parameter	Description
Joint Rotation: Hip	Choose between: -Head Pivot -Neck Pivot -Waist -Pelvis -Hip -Knee -Ankle -Shoulder -Elbow -Wrist
Joint Rotation: Both	Only for Hip Knee, Ankle, Shoulder, Elbow, and Wrist. Choose between: -Both (Default) -Left -Right
Rotate Body Parts	Only one can be active. Default depends on the joint selected.
Axis of Rotation	Only one can be active. Y is the default.
Rotation Parameters	Define the angle of rotation. The default is 90.0.

Beam Postprocessing

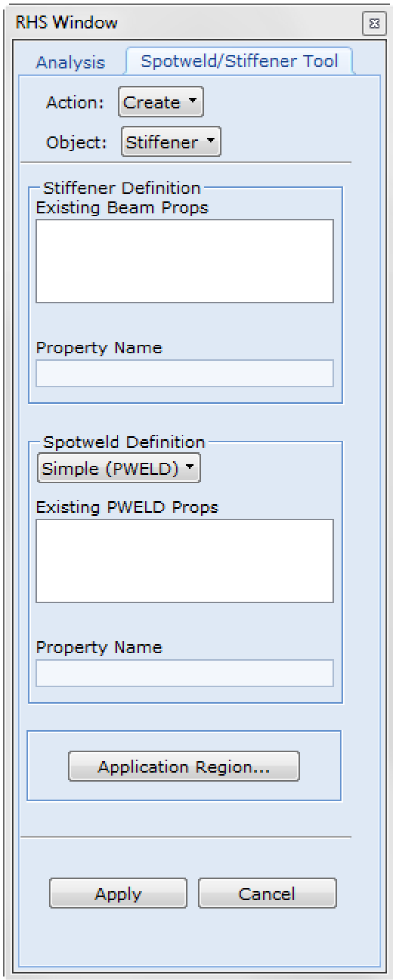
The subordinate form illustrated below appears if the “Object” on the Analysis form is set to “Beam Postprocessing”.



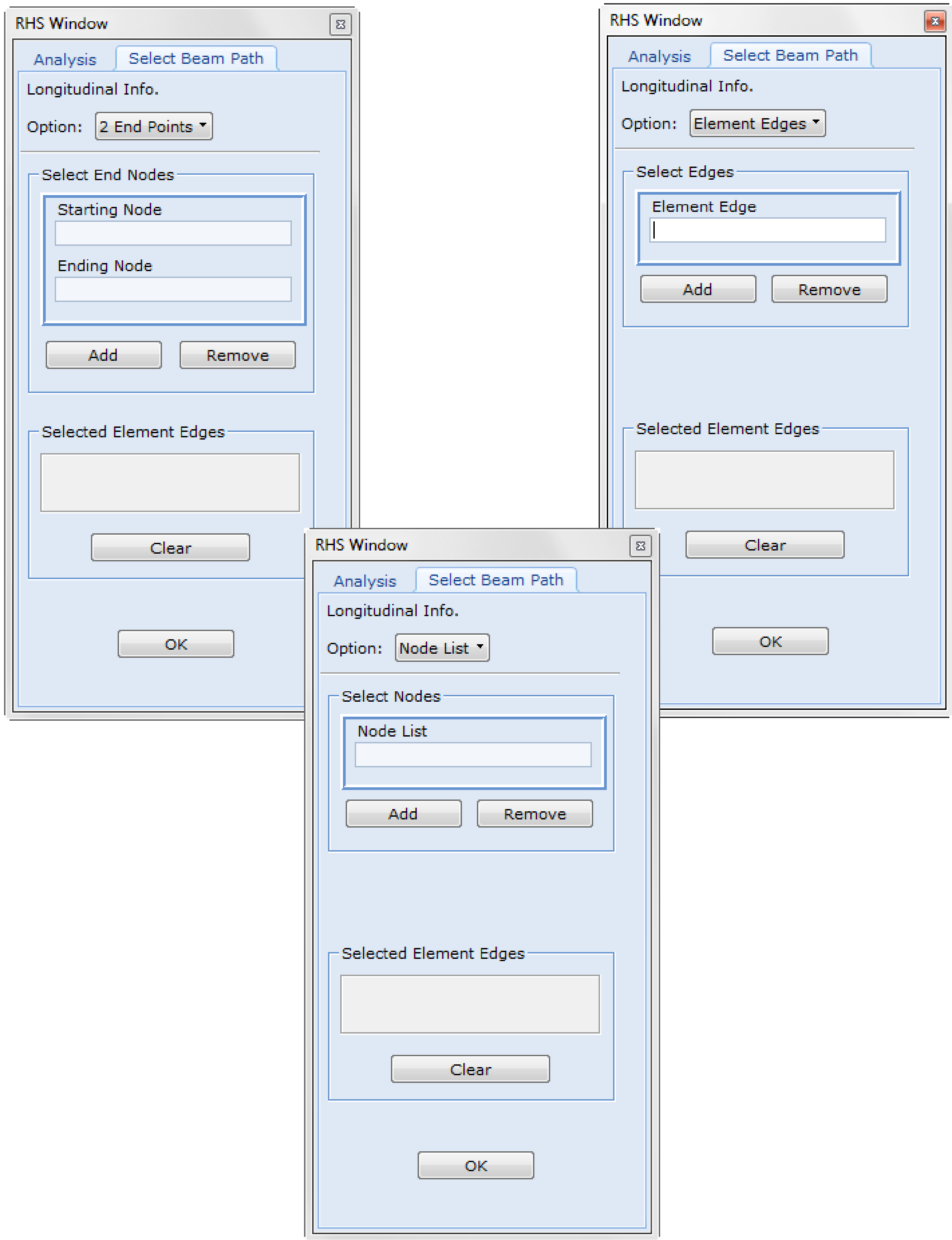
Parameter	Description
Result Cases	List of Result Cases User can select only one result case.
Available Results	List of qualified variables Only certain variables can be post-processed. See Dytran Users Manual under Sublayer Variables.
Display Options	Fringe can be plotted in a current viewport or a new viewport.
Auto Tile Viewport(s)	If the Beam Viewport was selected, the user has the option to automatically tile all opened viewports.
Select Beam(s)	Select beams to be post processed.

Spotweld/Stiffener Tool

The subordinate form illustrated below appears if the “Object” on the Analysis form is set to “Spotweld/Stiffener Tool” with options Create and Stiffener selected.

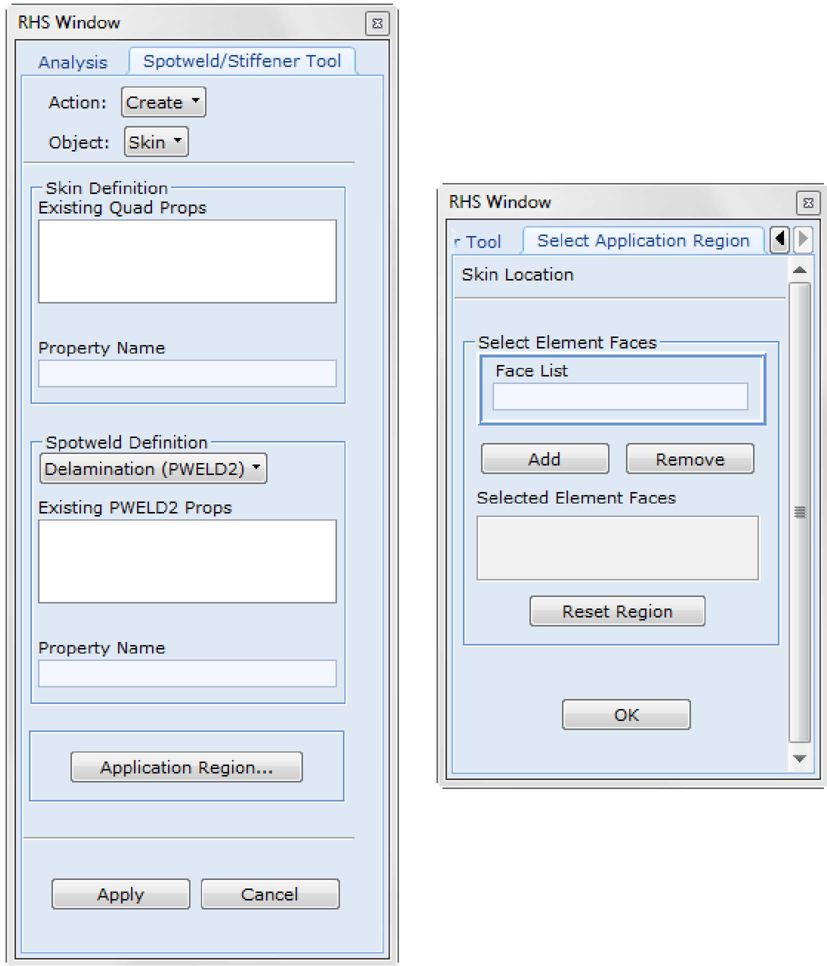


Parameter	Description
Existing Beam Props	List of existing beam property definitions.
Property Name	Selected name of new beam property from beam property list.
Spotweld Definition	Options: Simple (PWELD) Rupture (PWELD1)
Existing PWELD Props	List of existing spotweld property definitions.
Property Name	Selected name from spotweld property list or new spotweld property.


RHS Window
Analysis
Select Beam Path
Longitudinal Info.
Option: 2 End Points
Select End Nodes
Starting Node
Ending Node
Add
Remove
Selected Element Edges
Clear
OK
RHS Window
Analysis
Select Beam Path
Longitudinal Info.
Option: Element Edges
Select Edges
Element Edge
Add
Remove
Selected Element Edges
Clear
OK
RHS Window
Analysis
Select Beam Path
Longitudinal Info.
Option: Node List
Select Nodes
Node List
Add
Remove
Selected Element Edges
Clear
OK

Parameter	Description
Application Region Type 1: 2 End Points	
Select End Nodes	All shell element edges will be selected between the two points. Shortest distance along the shells will be calculated.
Select Element Edges	List of shell edges.
Application Region Type 2: Element Edges	
Element Edge	Shell element edges have to be selected directly.
Select Element Edges	List of shell edges
Application Region Type 3: Node List	
Node List	All element edges between the nodes will be selected.
Selected Element Edges	List of shell edges.

The subordinate form illustrated below appears if the “Object” on the Analysis form is set to “Spotweld/Stiffener Tool” with options Create and Skin selected.



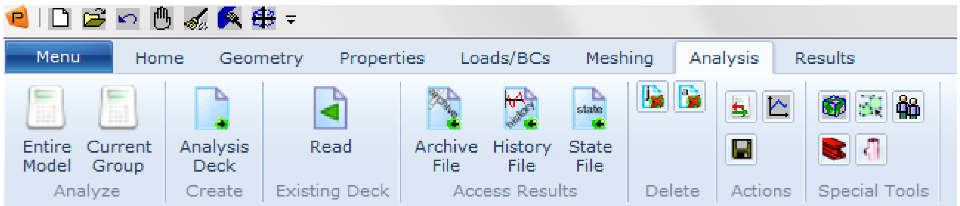
Parameter	Description
Existing Quad Props	List of: Existing Quad Props Property Definitions
Property Name	Selected name from quad property.
Delamination	Option: Delamination spotweld (PWELD2)
Existing PWELD2 Props	List of existing spotweld property definitions.
Property Name	Selected name from spotweld property list or new spotweld property.
Selected Elements Faces	Faces of solid elements may be selected.

Note:	After pressing Apply the following will happen: 1) Quads will be created on the solid faces from the application region. When a new name was typed in on Property Name, a new Default Pshell with dummy values will be created (an ACK message will appear). Otherwise the quads will be added to the existing quad property name. 2)The quads will be connected with the solid faces by spotwelds (CROD's) with zero length. When a new name was typed in on Property Name, and new spotweld with zero values will be created (an ACK message will appear). Otherwise the spotwelds will be added to the existing quad property name.

3 Running an Analysis

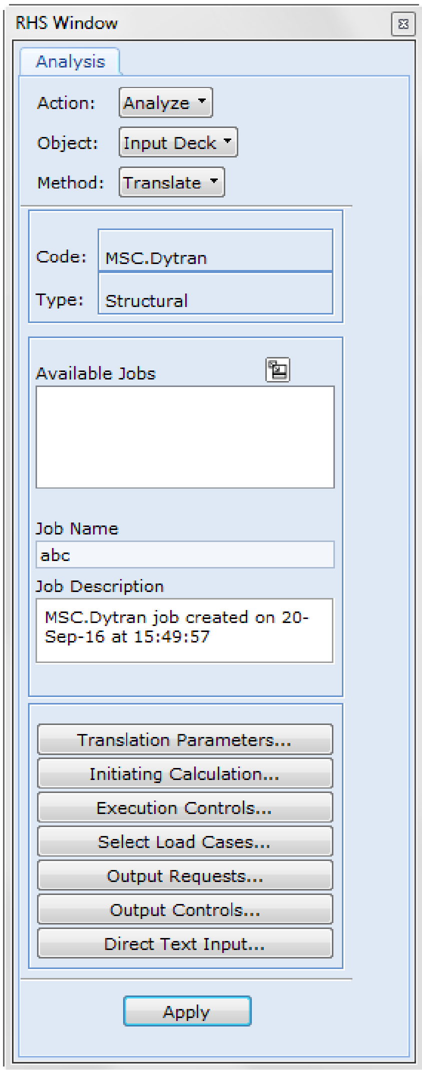
Review of the Analysis Form

The Analysis form appears when the Analysis toggle, located on the Patran switch, is chosen. To create an Dytran input file, select Analyze as the Action on the Analysis form. Other forms brought up by the Analysis form are used to define and control the analysis to be conducted and to set global defaults, where appropriate. These forms are described on the following pages. For further information see The Analysis Form (p. 6) in the *Reference Manual - Part V*.



Analysis Form

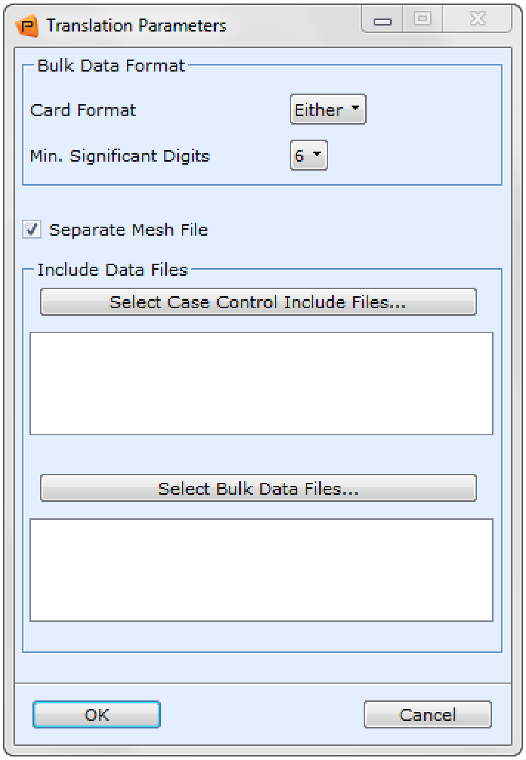
This form appears when the Analysis toggle is chosen on the main form. When preparing for an analysis run, select Analyze as the Action.



Parameter	Description
Action	Analysis Options for Action are: Analyze, Read Archive File, Read History File, Read Input File, Result Tools, Time History, Special Features, Save, and Delete.
Object	Options for Object depend on the Action selected. For "Analyze" these are: Input Deck, Bulk Data File Only, Restart, and Current Group.
Method	If the Action is "Analyze" then the options are: Translate and Full Run. Translate produces a Dytran input file. Full Run also initiates a Dytran analysis. If the Action is "Read Archive File" then the options are Attach and Translate.
Translation Parameters	These selections apply only when the action is "Analyze" and the object "Input Deck" or "Current Group." If the Object is "Restart" then there is one selection, "Restart Control." If the Object is "Bulk Data File Only" there is one selection, "Translation Parameters."
Initiating Calculation	
Execution Controls	
Select Load Cases	
Output Requests	
Output Controls	
Direct Text Input	

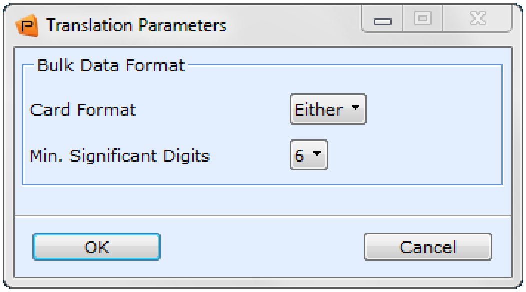
Translation Parameters

The translation parameters form allows the user to control the manner in which the Dytran input file is generated.



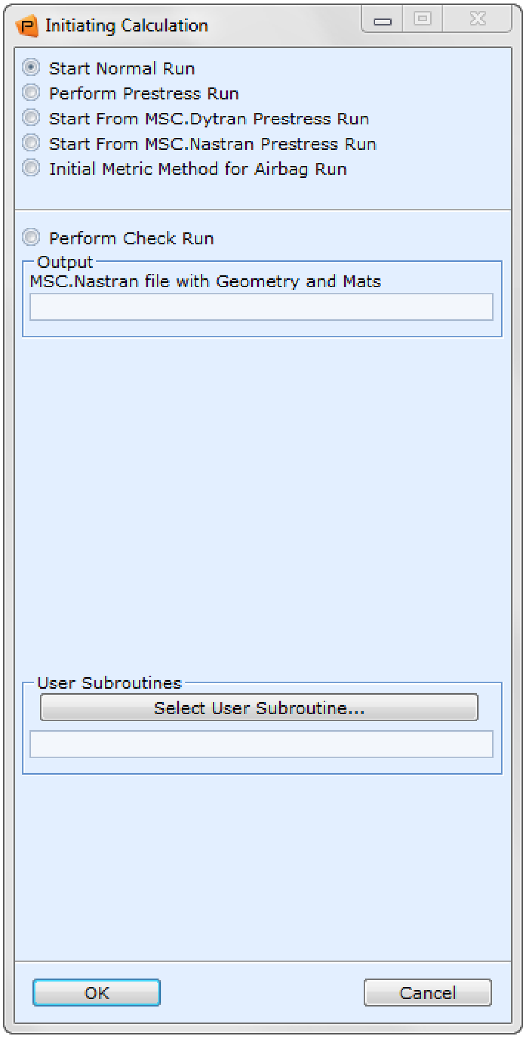
Parameter	Description
Card Format	The entry format may be Small, Large, Either, or Free.
Min. Significant Digits	Toggle between 4 and 7. The default is 6.
Separate Mesh File	The user may prefer that the mesh file is separate from the rest of the data deck. Set to On as default.
Select Case Control Include Files	Select files to be included in the Case Control section of the input file.
Select Bulk Data Files	Select files to be included in the Bulk Data section of the input file.

For the options Analyze/Bulk Data File Only, the Translation Parameters form is simplified to:



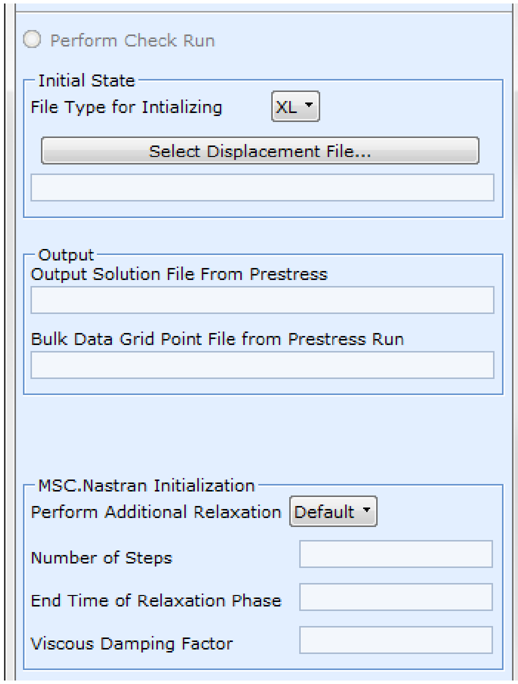
Initiating Calculation

The Initiating Calculation form allows the user to determine what type of Dytran analysis is to be conducted. This determines which options might be available on subsequent forms


Initiating Calculation
Start Normal Run
Perform Prestress Run
Start From MSC.Dytran Prestress Run
Start From MSC.Nastran Prestress Run
Initial Metric Method for Airbag Run
Perform Check Run
Output
MSC.Nastran file with Geometry and Mats
User Subroutines
Select User Subroutine...
OK
Cancel

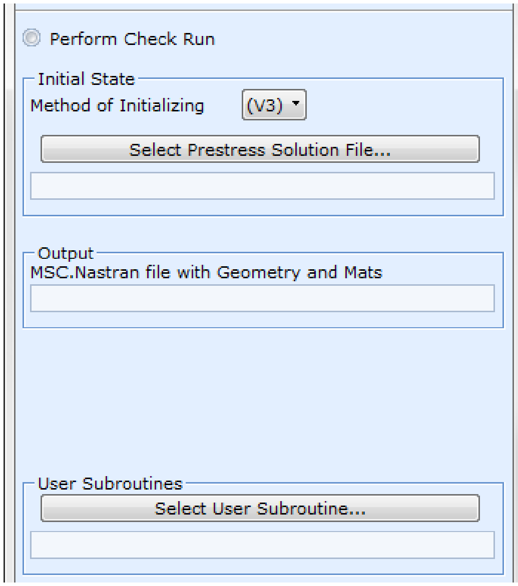
Parameter	Description
Start normal Run	Start Normal Run is the default. The corresponding FMS controls are: 1: START 2: PRESTRESS 3: START 4: START 5: START
Perform Check Run	Defines Case Control CHECK which allows the user to create an input file for a check run. The analysis will terminate after 2 time steps, after completing a full data check.
Output	Enter the name of the file referenced in the NASTOUT FMS statement.
Select User Subordinate	Defines USERCODE FMS. Select the file in which any user subroutines are to be located.

For Perform Prestress Run the variable part of the Initiating Calculation form is



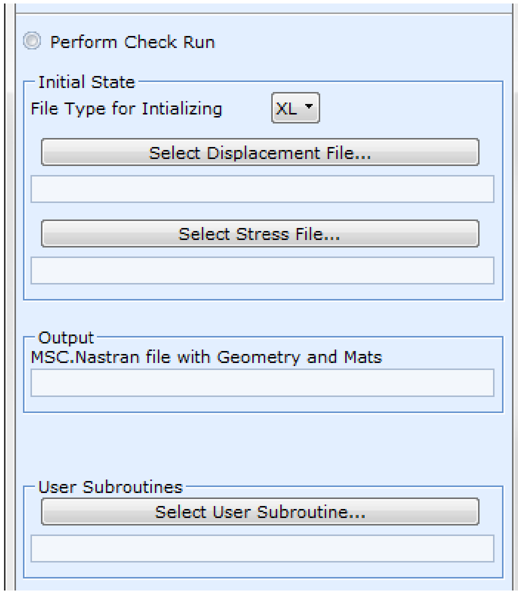
Parameter	Description
File Type for Initializing	Defines PARAM INITNAS, Select XL (Default), PUNCH or PATRAN.
Select Displacement File	Defines NASTDISP FMS.
Output Solution File From Prestress	Defines SOLUOUT FMS.
Bulk Data Grid Point File from Prestress Run	Defines BULKOUT FMS.
MSC. Nastran Initialization	Defines NASINIT Bulk Data.
Perform Additional Relaxation	Select Default, YES or NO.

For Start From Dytran Prestress Run the variable part of the form is.



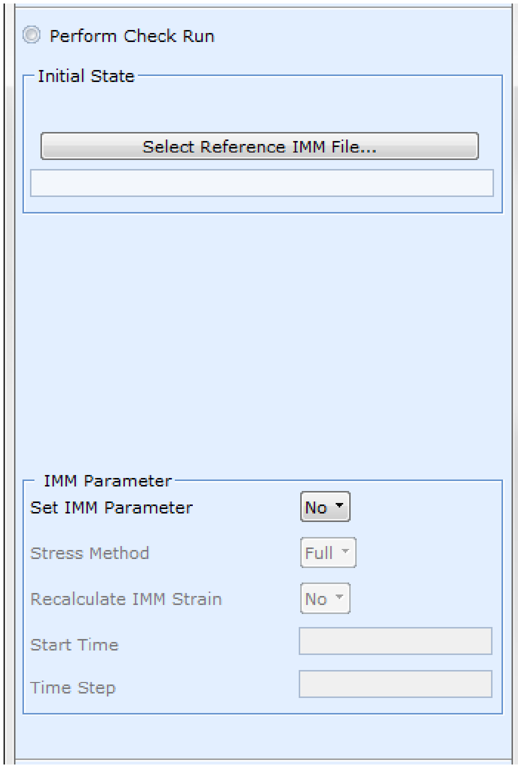
Parameter	Description
Perform Check Run	Defines Case Control CHECK.
Method of Initializing	Defines PARAM, INITFILE, Select Strict (V1), Flexible (V2), or (V3) (Default)
Select Prestress Solution File	Defines SOLINT FMS.
MSC. Nastran File with Geometry and Mats	Defines NASTOUT FMS.
Select User Subroutines	Defines USERCODE FMS.

For Start From MSC Nastran Prestress Run the variable part of the form is



Parameter	Description
Perform Check Run	Defines Case Control CHECK.
File Type for Initializing	Defines PARAM, INITNAS, Select XL (Default), PUNCH or PATRAN
Select Displacement File	Defines NASTINP FMS.
MSC. Nastran file with Geometry and Mats	Defines NASTOUT FMS.
Select User Subroutine	Defines USERCODE FMS.

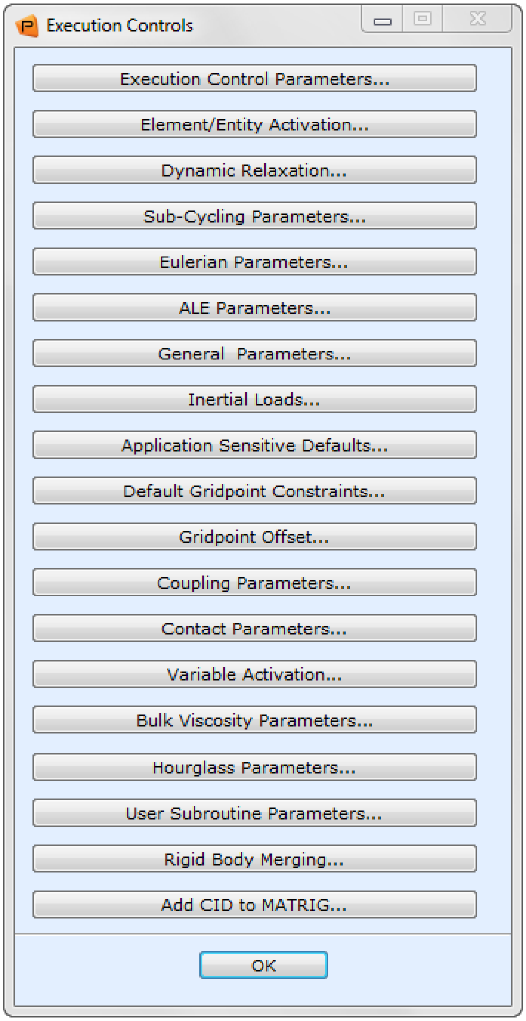
For Initial Metric Method for Airbag Run the variable part of the form is



Parameter	Description
Perform Check Run	Defines Case Control CHECK.
Select Reference IMM File	Defines IMMFILE FMS.
IMM Parameter	Defines PARAM, IMM.
Set IMM Parameter	Select No (Default) or Yes.
Recalculate IMM Strain	
Stress Method	Select Full (Default), Reduced, or Zero.

Execution Controls

The Execution Controls form provides access to subordinate forms upon which are defined the parameters controlling execution of an Dytran analysis.

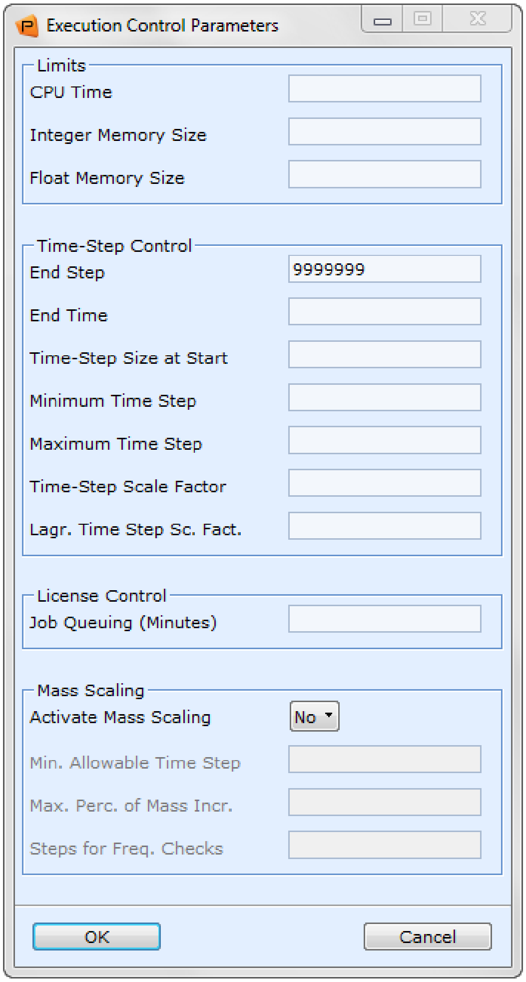


Parameter	Description
Inertial Load	Defines gravitational and rotational inertial loads to be applied to the whole model. (TLOAD1, GRAV, FORCE).
Default Gridpoint Constraints	Defines the default constraints for "Single Point Constraints." (GRDSET) and the parameter NZEROVEL.
Gridpoint Offset	Defines the offsets for selected groups of nodes. (GRDOFFS).

Note:	Use only those selections relevant to the analysis to be performed. The subordinate forms are illustrated and described in the following pages. In the subordinate forms, if the databox is left blank or the option "Default" is selected in the option menu, the corresponding parameter will not be written to the input file. The Dytran solver will use the parameter's default value.

Execution Control Parameters

The Execution Controls subordinate form defines data to be written to the Executive Control and Case Control sections of the input file.

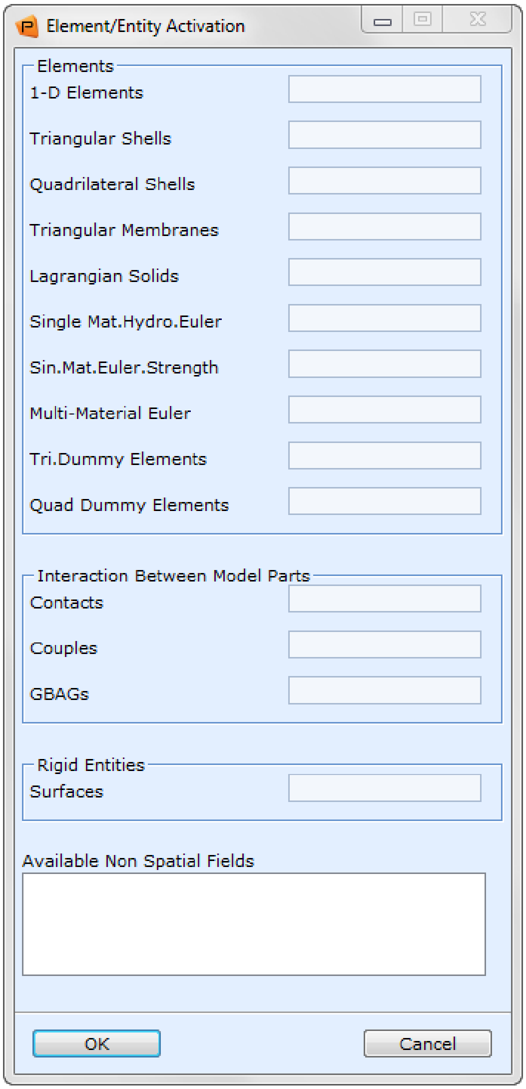


Parameter	Description
Limits	Defines TIME and MEMORY SIZE in the Executive Control Section of the input data file.
CPU Time	Total CPU time limit in minutes.
Integer Memory Size	Defines the size of integer memory in words.
Float Memory Size	Defines the size of float memory in words.
Time Step Control	These parameters control the start and end of the analysis and place constraints on time, step, size, and scaling between successive steps.
End Step End Time	Defines ENDSTEP and ENDTIME in the Case Control Section of the input data file.

Parameter	Description
Time Size Step at Start	Defines the parameters INISTEP, MINSTEP, MAXSTEP, STEPFCT and STEPFCTL in the input data file.
Minimum Time Step	
Maximum Time Step	
Time-Step Scale Factor	
Lagr. Time Step Sc. Factor	
Job Queuing	Defines PARAM, AUTHQUEUE.
Mass Scaling	Defines PARAM, SCALEMAS.

Element/Entity Activation

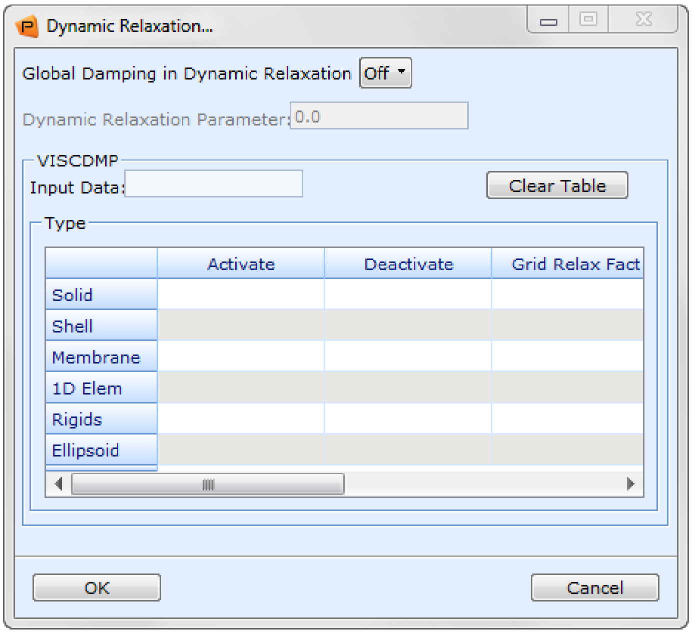
This form defines the parameters that control activation of elements for only part of an analysis. The data is all entered via the ACTIVE entry in the Bulk Data section of the input file. This means they cannot be reset on RESTART. The defaults for all these entries are on at all times.



Parameter	Description
Elements	Field Name (Non Spatial)
Interaction Between Model Parts	
Rigid Entities	
Available Non Spatial Field	List of existing Non-Spatial-Fields

Dynamic Relaxation Parameters

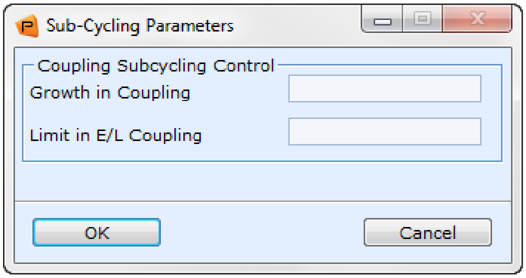
This form allows the user to define the data for the VISCDMP Bulk Data entry or the parameter VDAMP.



Parameter	Description
Global Damping in Dynamic Relaxation	Choose ON or OFF. The relaxation parameter can be defined if Dynamic Relaxation is set ON. Otherwise, the VISCDMP entry can be defined.
Dynamic Relaxation Parameter	Defines PARAM, VDAMP.
VISCDMP	Defines the VISCDMP entry. **Note:** The 4th column Stiff Relax Fact can be used only for Membrane.

Sub-Cycling Parameters

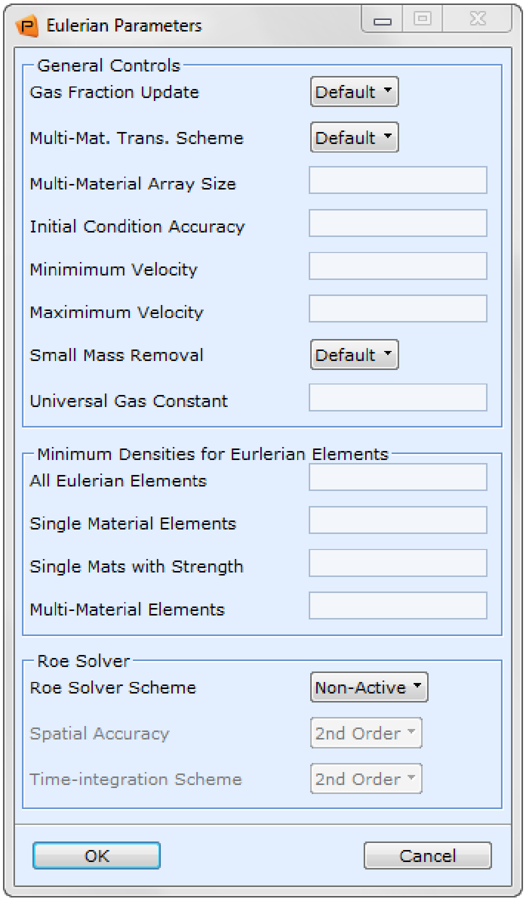
This form is used to define the sub-cycling parameters for Euler-Lagrange coupling.



Parameter	Description
Growth in Coupling	Defines PARAM, COSUBCYC: growth of subcycling interval in coupling.
Limit in E\L Coupling	Defines PARAM, COSUBMAX: subcycle limit in coupling.

Eulerian Parameters

This form is used to define the coefficients for eulerian elements.

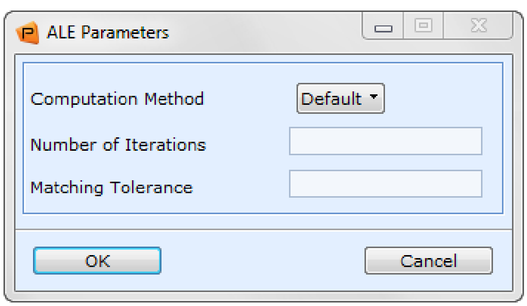


Parameter	Description
Gas Fraction Update	Defines PARAM, MIXGAS. Select Default, Yes, or No.
Multi. Mat. Trans. Scheme	Defines PARAM, EULTRAN. Select Default, Impulse, or Average.
Multi-Material Array Size	Defines PARAM, FMULTI. Dimensioning of the multi-material overflow array.
Initial Condition Accuracy	Defines PARAM, MICRO.
Minimum Velocity	Velocity controls for Eulerian elements, VELCUT and VELMAX.
Maximum Velocity	
Small Mass Removal	Select Default, Yes, or No
Universal Gas Constant	Defines PARAM, UGASC.

Parameter	Description
Minimum Densities for Eulerian Element	Density controls for Eulerian Elements: RHOCUT, ROHYDRO, ROSTR, and RHOMULTI.
Roe Solver Scheme	Choose between: Non-Active (Default) Active
Spatial Accuracy	Defines PARAM, LIMITER, ROE Choose between: -2nd Order (Default) -1st Order
Time Integration Scheme	Defines PARAM, RKSCHEME Choose between: -2nd Order (Default) -1st Order

ALE Parameters

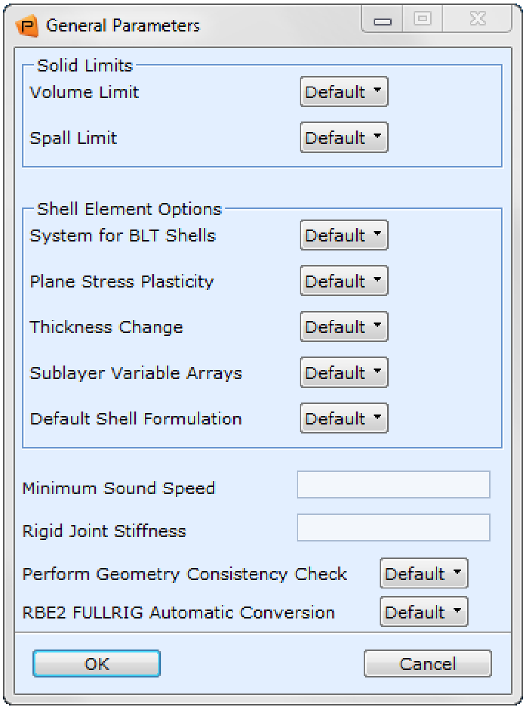
This form allows the user to define the data for the ALE Parameter Options on the PARAM entry of the input file.



Parameter	Description
Computation Method	Defines PARAM, ALEVER. Select Default, Fast (V2.1) or Exact (V2.2).
Number of Iterations	Defines PARAM, ALEITR. Number of ALE grid iterations.
Matching Tolerance	Defines PARAM, ALETOL. The tolerance at the ALE interface.

General Parameters

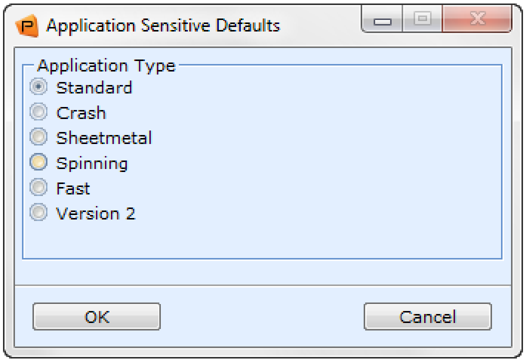
This form defines failure limits and shell options. All are PARAM entries in the Bulk Data Section of the input file.



Parameter	Description
Volume Limit	Defines PARAM, HVLFAIL. Select Default, No Failure, or Failure.
Spall Limit	Defines PARAM, PMINFAIL. Select Default, No Failure, or Failure.
System for BLT Shells	Defines PARAM, SHELMSYS. Select Default, Midsides, or Side21.
Plane Stress Plasticity	Defines PARAM, SHPLAST. Select Default, Radial Return, Vect. Iterative, or Non Vect. Iterative.
Thickness Change	Defines PARAM, SHTHICK. Select Default, Not Modifies, or Modified.
Sublayer Variable Arrays	Defines PARAM, SLELM. Select Default, Store, or Do Not Store.
Default Shell Formulation	Defines PARAM, SHELLFORM. Select Default, BELY, BLT, or KEYHOF.
Minimum Sound Speed	Defines PARAM, SNDLIM. The minimum sound speed for fractured elements.
Rigid Joint Stiffness	Defines PARAM, RJSTIFF.
Perform Geometry Consistency Check	Defines PARAM, GEOCHECK. Select Default, Yes, or No.
RBE2 FULLRIG Automatic Conversion	Defines PARAM, CFULLRIG. Select Default, Yes, or No.

Application Sensitive Defaults

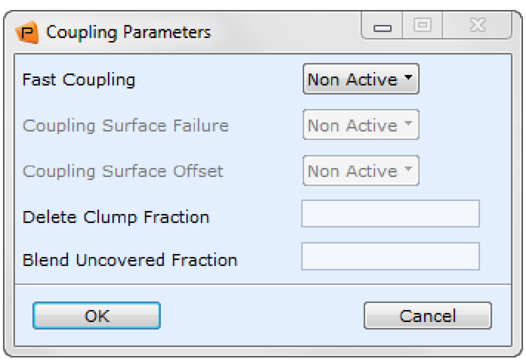
This form causes the defaults to be customized for a particular application of Dytran.



Parameter	Description
Application Type	The default is “Standard”.

Coupling Parameters

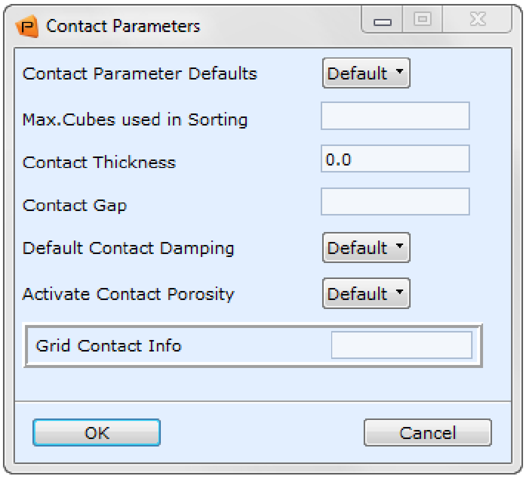
This form allows you to define the data for the FASTCOUP Parameter Options on the PARAM entry of the input file.



Parameter	Description
Fast Coupling	Defines PARAM, FASTCOUP. Choose between: -Non-Active (Default) -Active
Coupling Surface Failure	
Coupling Surface Offset	
Delete Clump Fraction	Defines PARAM, DELCLUMP.
Blend Uncovered Fraction	Defines PARAM.

Contact Parameters

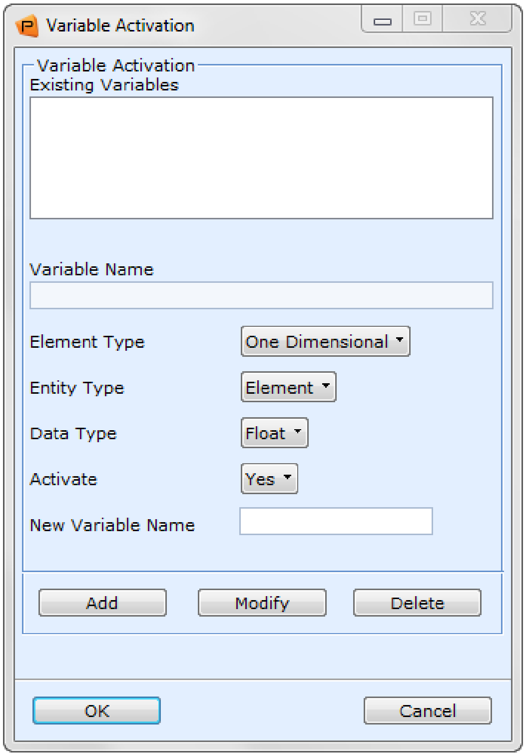
This form is used to define the contact control parameters.



Parameter	Description
Contact Parameter Defaults	Defines PARAM, CONTACT, DYNA. Select Default or MSC/Dyna.
Max Cubes Using Sorting	Defines PARAM, LIMCUB.
Contact Thickness	Defines PARAM, CONTACT, THICK.
Contact Gap	Defines PARAM, CONTACT, GAP.
Default Contact Damping	Defines PARAM, CONTACT, DAMPING. Select Default, Off or On.
Activate Contact Porosity	Defines PARAM, CONTACT, COPOR. Select Default, Yes, or No.
Grid Contact Info	Defines PARAM CONTACT, INFO.

Variable Activation

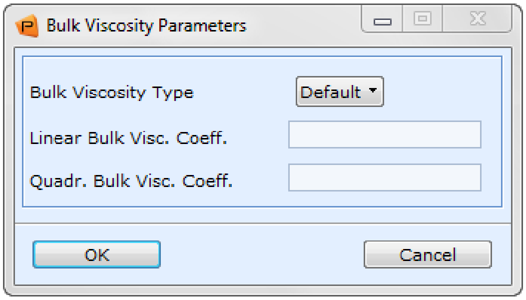
This form is used to define the data for the VARACTIV parameter.



Parameter	Description
Element Type	Element type options: -One Dimensional -Triangular Shell -Quadrilateral Shell -Membrane -Triangular Dummy -Quadrilateral Dummy -Lagrangian Solids -Eulerian Hydro Solid -Multimat. Eulerian Solid -Activate All Variables -Activate All and Print
Entity Type	Entity Type options: -Element -Grid Point -Face
Data Type	Data Type options: -Float -Integer -Character
Activate	Activate options -Yes -No
New Variable Name	If the form is left blank, the existing variable name will be used.

Bulk Viscosity Parameters

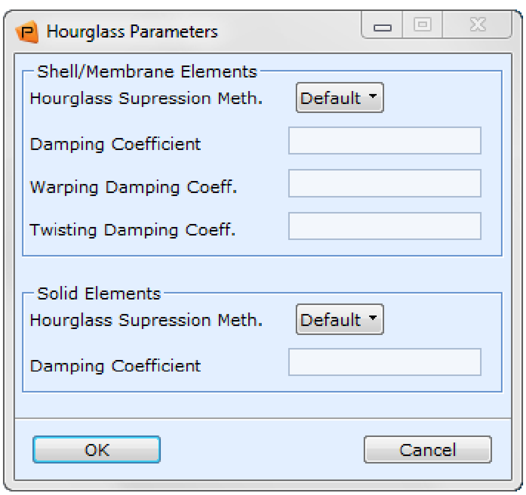
This form is used to define the data for the bulk viscosity control parameters.



Parameter	Description
Bulk Viscosity Type	Defines PARAM, BULKTYP. Select Default, Dyna, or Dytran.
Linear Bulk Visc. Coeff.	Defines PARAM, BULKL.
Quadr. Bulk Visc. Coeff	Defines PARAM, BULKQ.

Hourglass Parameters

This form is used to define the data for the hourglass control parameters.

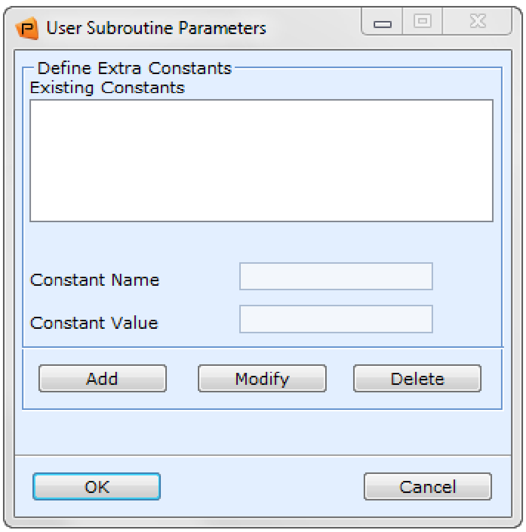


Parameter	Description
Hourglass Suppression Method	Defines PARAM, HGSHELL. Select Default, F-B Viscous, or Dyna.
Damping Coefficient	Defines PARAM, HGCMEM.
Warping Damping Coeff.	Defines PARAM, HGCWRP.
Twisting Damping Coeff.	Defines PARAM, HGCTWS.

Parameter	Description
Hourglass Suppression Method	Defines PARAM, HGSOLID. Select Default, F-B Stiffness or Dyna.
Damping Coefficient	Defines PARAM, HGCSOL.

User Subroutine Parameters

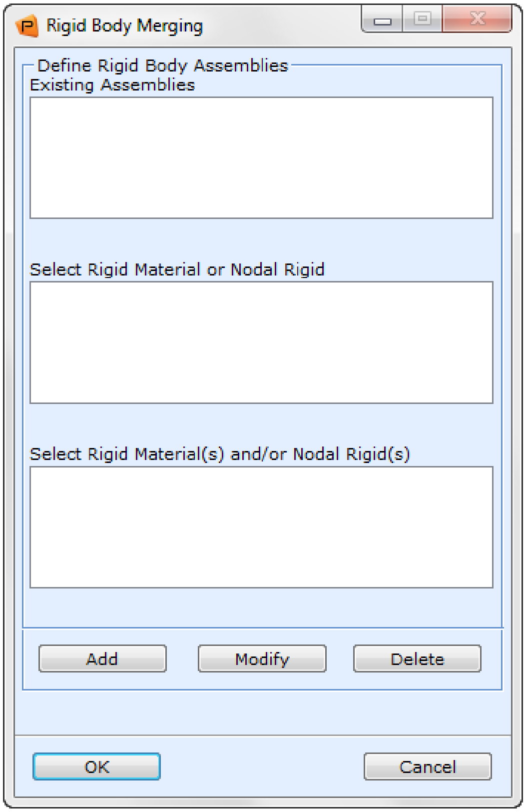
This form is used to define the data for the EXTRAS parameter.



Parameter	Description
Existing Constants	This is a list of existing constants. Clicking on an existing name brings up the Constant Name and its Value in the boxes below.
Constant Name	This is the name of a new or existing constant.
Constant Value	This is the value of a new or existing constant.
Add	Select Add, Modify, or Delete to create a new constant or modify or delete an existing constant.
Modify	
Delete	

Rigid Body Merging

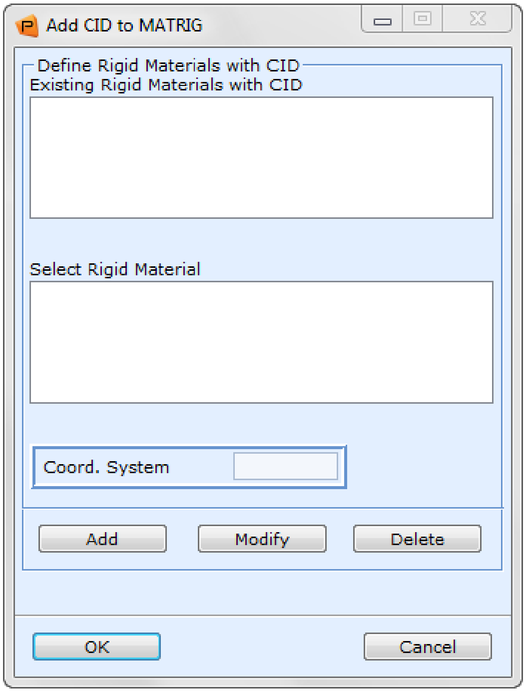
This form is used to define the data for the MATRMRG1 parameter.



Parameter	Description
Existing Assemblies	This is a list of existing assemblies. Clicking on an existing name highlights the selected items in the listboxes below.
Select Rigid Material or Nodal Rigid	This is a list of existing rigid materials and Nodal Rigid lbcs. Only one item may be selected. The selected item will be the name of the assembly.
Select Rigid Material and/or Nodal Rigid	This is a list of existing rigid materials and Nodal Rigid lbcs. Multiple items may be selected. The selected items will be merged into a new assembly.
Add	Select Add, Modify, or Delete to create a new assembly or modify or delete an existing assembly.
Modify	
Delete	

Add CID to MATRIG

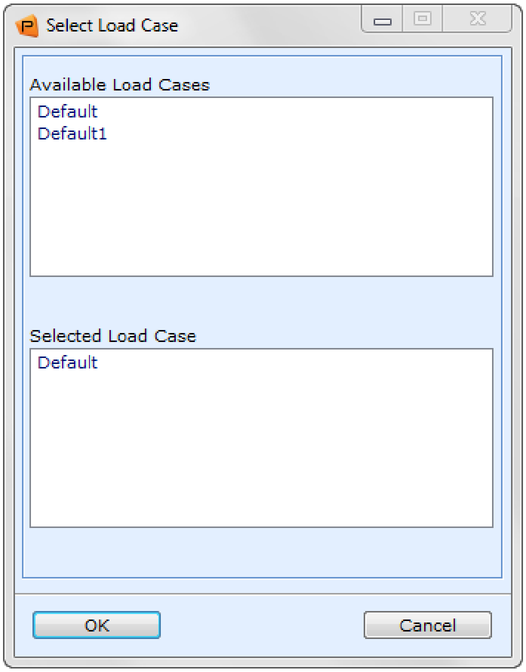
This form is used to define local coordinate system (CID) of centre of gravity in the MATRIG entry.



Parameter	Description
Existing Rigid Materials with CID	This is a list of existing MATRIG with CID. Clicking on an existing name highlights the selected item in the listbox and shows the CID in the databox below.
Select Rigid Material	This is a list of existing rigid materials. Only one item may be selected. The selected item will be the name of the MATRIG with CID.
Coord. System	This is the local coordinate system (CID) of a new or existing MATRIG with CID.
Add	Select Add, Modify, or Delete to create a new assembly or modify or delete an existing assembly.
Modify	
Delete	

Select Load Cases

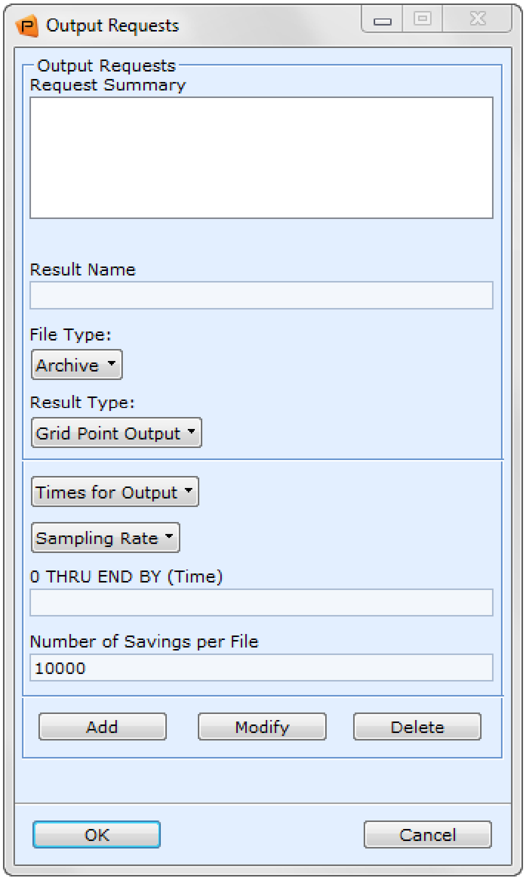
This form appears when the Select Load Case button is selected on the Analysis form. Use this form to select the load case to be included in this run.



Parameter	Description
Available Load Cases	Displays the list of all load cases currently in the database. The desired load cases may be selected from this area.
Selected Load Cases	Only one load case can be selected. The default is the current load case.

Output Requests

This form allows the definition of what results data is desired from the analysis code in the form of results. The settings can be accepted, as altered, by selecting the OK button on the bottom of the form. If the Cancel button is selected instead, the form will be closed without any of the changes being accepted. Selecting the Defaults button resets the form to the initial default settings.



Parameter	Description
Request Summary	This is a list of results requests. Clicking on an existing name brings up the "Result Name" and its description in the boxes below.
Result Name	This is the name of a new or existing "Result".
File Type	File types may be: Archive, Time History, Restart File, Step Summary, Material Summary, Eulerian Boundary Summary, User Defined Output and Rigid Body Summary.

Parameter	Description
Result Type	Depending on the selection in the file type menu, the results types may be: -Grid Point Output (ARC, THS) -Element Output (ARC, THS) -Rigid Surface-MATRIG Output (THS) -Gas Bag Output (THS -Rigid Ellipsoid Output (ARC, THS) -Material Output (ARC, THS) -Contact Surface Output (ARC, THS) -Cross Section Output (ARC, THS) -Coupling Surface Output (ARC, THS) -Surface Output (THS) -Subsurface Output (THS) -Eulerian Boundary (THS) -Center of Gravity (THS) -Accelerometer Output (THS) -Head Injury Criteria (THS) -User Defined Grid Point Output (UDO) -User Defined Element Output (UDO)
Times for Output	Choose between: -Times for Output (Default) -Steps for Output
Sampling Rate	Choose between: -Sampling Rate (Default) -User Specified
0 Thru End By Time	Title changes depending on selection.
Add	Select Add, Modify, or Delete to create a new assembly or modify or delete an existing assembly.
Modify	
Delete	

This form is used to define the following entries:

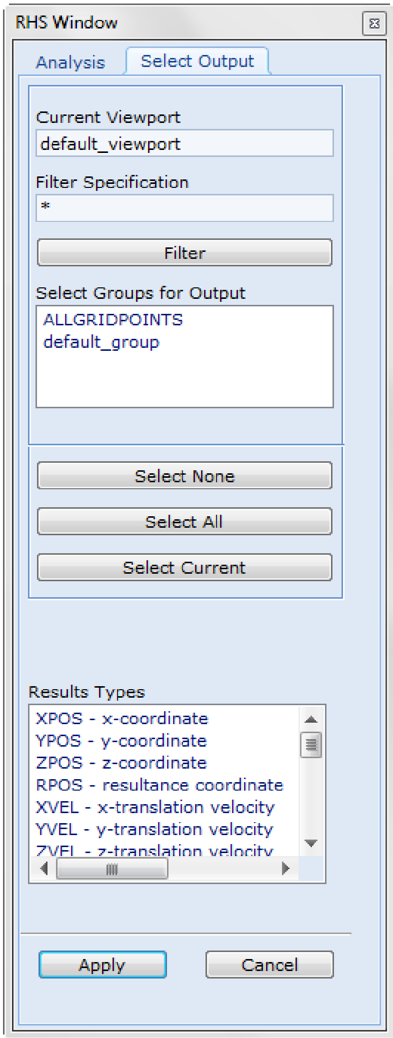
TYPE (*Result Name*) = {File Type option}

TIMES (*Result Name*) = {Value} (Note: if option is Times for Output)

STEPS (*Result Name*) = {Value} (Note: if option is Steps for Output)

SAVE (*Result Name*) = {Number of Saving per File}

To define the data written to the requested file the following subordinate form is used.



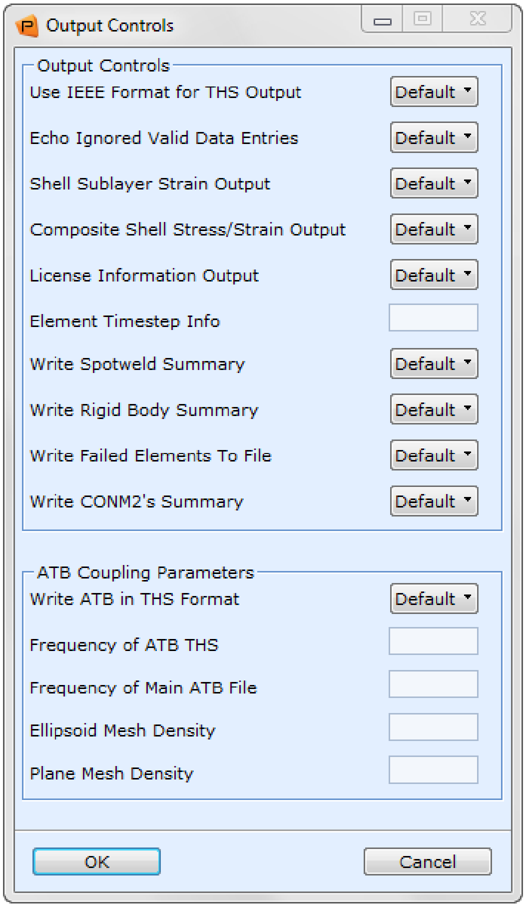
Parameter	Description
Current View	These facilitate selection of the groups for which data is to be output. Set the viewport so it contains the groups of entities of interest. The filter is useful if a consistent group naming convention is used.
Filter Specification	
Select Groups for Output	Groups for grid point and element. Material Names for material output. List the groups for which output might be requested. Here the filter * was used so all groups in the current viewport are listed. Click on those required.

Parameter	Description
Select None	These exist to aid selection of the required groups of entities.
Select All	
Select Current	
Result Types	This acts as a filter on the groups selected. Only those passing the filter will be requested.

Note:	The options except Result Type are disabled for Rigid Surface, Gas Bag, Contact, and Cross Section.

Output Controls

This form allows the user to control output options during an analysis. The defaults will normally be acceptable.

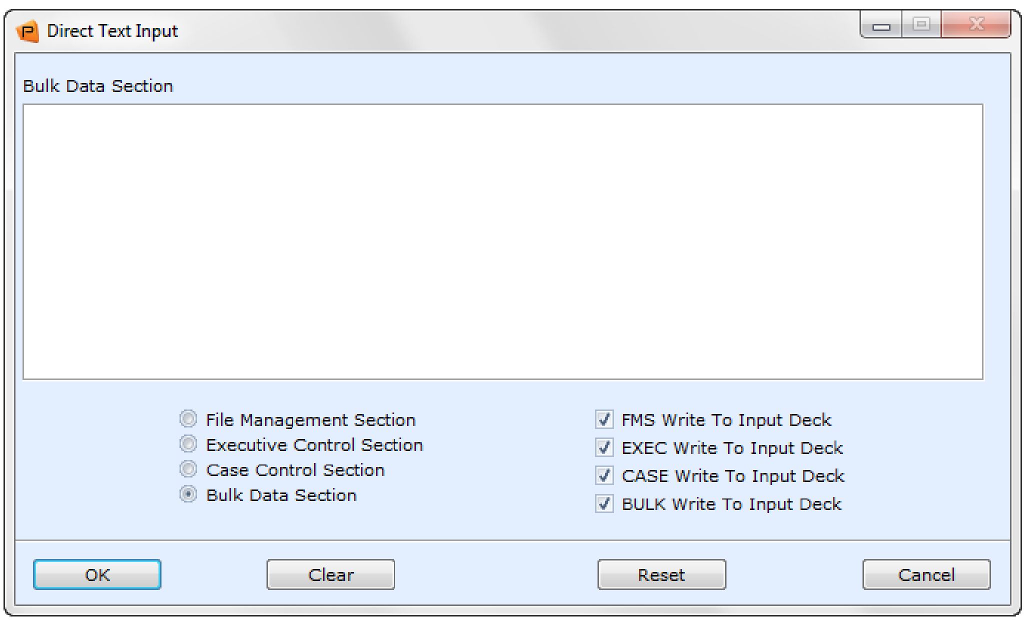


Parameter	Description
Use IEEE Format for THS Output	Defines PARAM, IEEE. Select Default, OFF, or ON.
Echo Ignored Valid Data Entries	Defines PARAM, NASIGN. Select Default, No, or Yes.
Shell Sublayer Data Output	Defines PARAM, STRNOUT. Select Default, No, or Yes.
Composite Shell Stress/Strain Output	Defines PARAM, SHSTRDEF. Select Default, Fiber/Matrix, or Element.
Licence Information Output	Defines PARAM, AUTHINFO. Select Default, Minimum, Medium, or Maximum.
Element Timestep Info	Defines PARAM, ELDTH.
Write Spotweld Summary	Defines PARAM, INFO-BJOIN. Select Default, Yes, or No.

Parameter	Description
Write Rigid Body Summary	Defines PARAM, RBE2INFO. Select Default, Yes, or No.
Write Failed Elements to File	Defines PARAM, FAILOUT. Select Default, Yes, or No.
Write CONM2's Summary	Defines PARAM, CONM2OUT. Select Default, Yes, or No.
Write ATB in THS Format	Defines PARAM, ATB-H-OUTPUT. Select Default, Yes, or No.
Frequency of ATB THS	Defines PARAM, ATBTOUT.
Frequency of Main ATB File	Defines PARAM, ATBAOUT.
Ellipsoid Mesh Density	Defines PARAM, MESHELL.
Plane Mesh Density	Defines PARAM, MESHPLN.

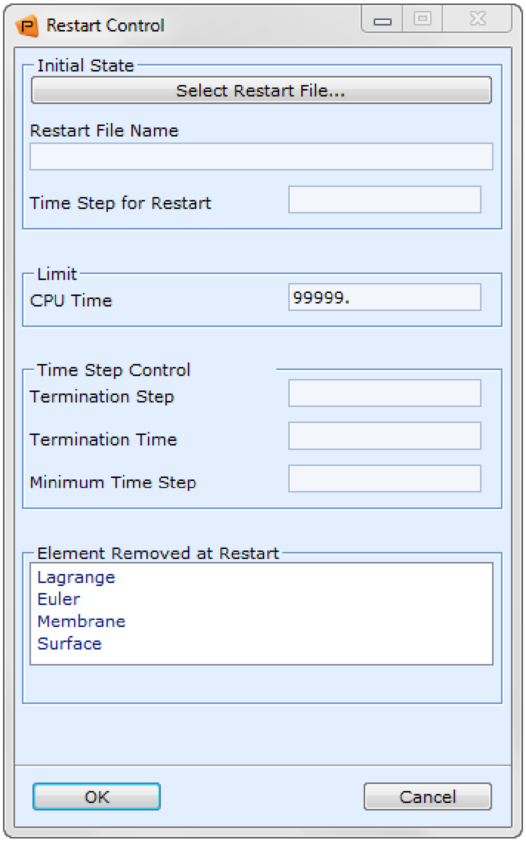
Direct Text Input

The Direct Text Input form allows you to add text directly to the Dytran input file.



Restart Control

This form allows the user to define parameters controlling the restart of a Dytran job.

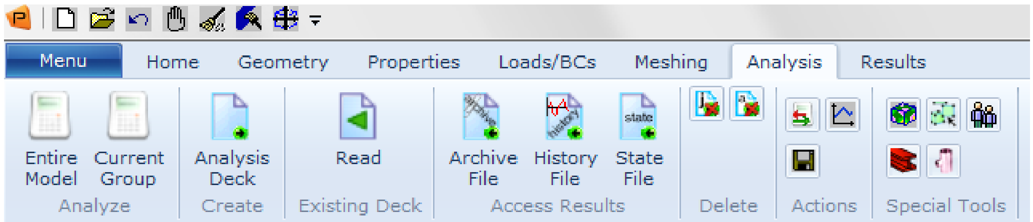


Parameter	Description
Select Restart File	Click on this to access a standard file selection form named “Select Restart File.” The default filter is .RST.

4 Read Results

Review of the Read Results Form

The Analysis form will appear when the Analysis toggle, located on the Patran control panel, is chosen.



Creation of MPEGs

In recognition of the value of being able to store compactly and quickly replay results animations, utilities are provided within the context of results recovery, to enable a user to save animations in MPEG format. These make use of a public domain program that can be obtained by anonymous ftp from: havefun.stanford.edu/pub/mpeg/MPEGv1.2.2.tar.Z. The terms under which MSC Software makes use of this program are documented in the following statement.

Copyright (C) 1990, 1991, 1993 Andy C. Hung, all rights reserved. PUBLIC DOMAIN LICENSE: Stanford University Portable Video Research Group. If you use this software, you agree to the following: This program package is purely experimental, and is licensed "as is". Permission is granted to use, modify, and distribute this program without charge for any purpose, provided this license/ disclaimer notice appears in the copies. No warranty or maintenance is given, either expressed or implied. In no event shall the author(s) be liable to you or a third party for any special, incidental, consequential, or other damages, arising out of the use or inability to use the program for any purpose (or the loss of data), even if we have been advised of such possibilities. Any public reference or advertisement of this source code should refer to it as the Portable Video Research Group (PVRG) code, and not by any author(s) (or Stanford University) name.

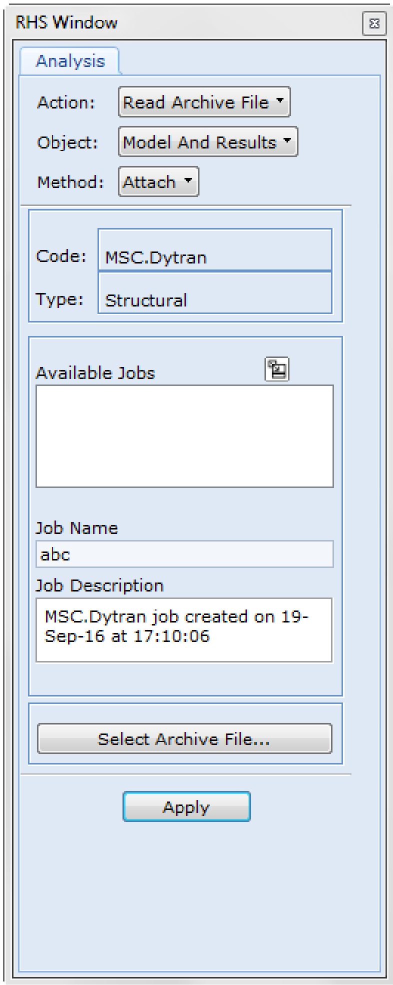
MSC Software has integrated the PVRG code within the Patran Dytran preference, "as is," for the convenience of the users of that preference. No warranty or maintenance of PVRG code is given by MSC Software, either expressed or implied.

Group Creating/Posting

New groups will be automatically posted to the viewport only when model data are imported. The new groups are still being created and populated during the results import process, but just not posted to the current viewport. However, you still have the option to post these new groups manually under Group/Post menu.

Read Results Form

The Action option menu provides several methods to import and process results.



Parameter	Description
Action	Options for Actions related to results recovery are: Read Archive File; Read History File; Results Tools; Time History; Read State File.
Object	Options for Object depend upon the Action selected. For “Read Archive File” these are: Model, Results, or Model and Results. For “Read History File” the Object is Results. For “Result Tools” and “Time History” there is no object or method. For “Read State File” the options are Results Entities, Model Data, and Both.

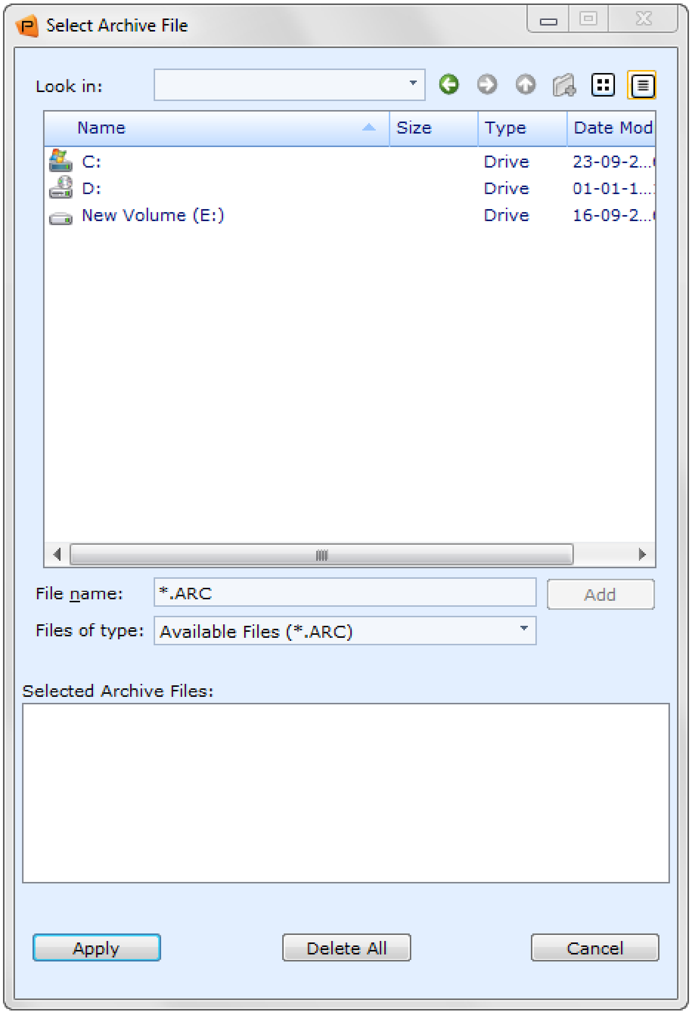
Parameter	Description
Method	Options for Method depend upon the Action selected. For "Read Archive file" these are Attach and Translate. For "Read History File" the Method is Translate. For "Read State File" the method is Attach. The Attach method uses the Direct Results Access (DRA) mechanism.
Job Name	p3dytran uses the Jobname as a title for the current job.
Select Archive File	The selection here, if any, depends upon the "Action" selected.

Subordinate Forms

The subordinate forms accessed from the Read Results form will depend upon the Action and Object selected. The various possibilities are described in this subsection. The first of these allows the user to select the archive file from which results are to be recovered. The remainder supports specialized functionality that is intended to enable visualization of the transient results produced by Dytran, by facilitating the creation time history plots.

Select Results File Subsidiary Form

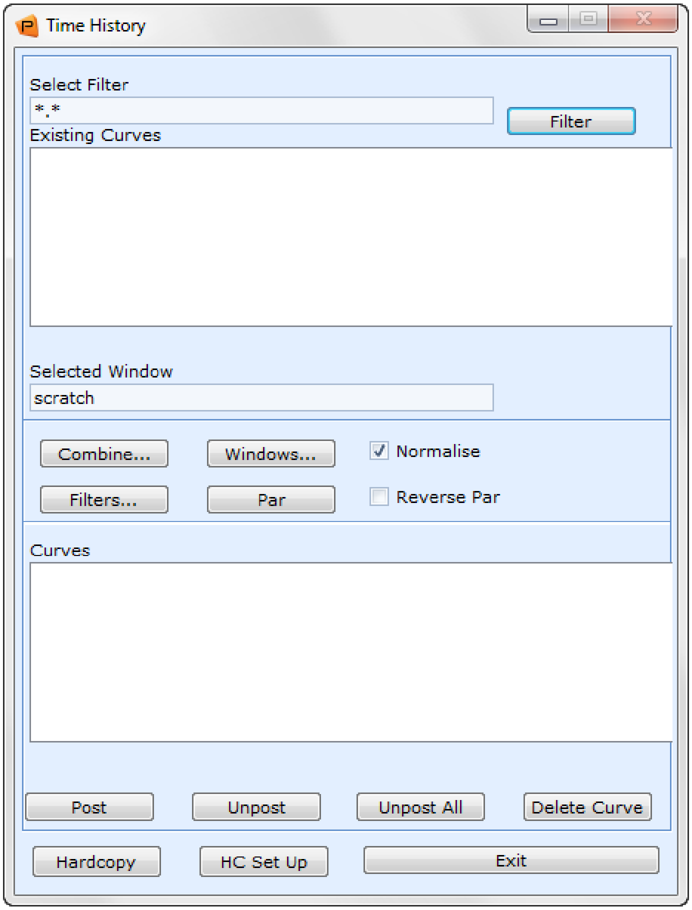
The subordinate archive file selection form allows the user to select either an Dytran archive file or an Dytran history file from which data is to be extracted. The name of this form will be either "Select Archive File" or "Select History File." These differ only in the names on the form and the default filter. The results reader dytranp3 is set up to read in Model data first from Archive files. This is specified by selecting Model as object on the main analysis form. Next, the object has to be changed to Results. Upon apply dytranp3 reads results from the selected Archive Files. If multiple Archive files exist for different timesteps but for the same elements/nodes, only one Archive file has to be read with the object Model on the main analysis form.



Parameter	Description
Look In	Select the location of *.ARC file.
File Name	Type or Select the name of file.
Files of Type	Shows the list of acceptable file extension.

Time History Subsidiary Form

The time history tool facilitates generation of time history plots. Advantages of using this tool are the possibilities of combining and parameterizing curves, filtering and a hard copy facility.

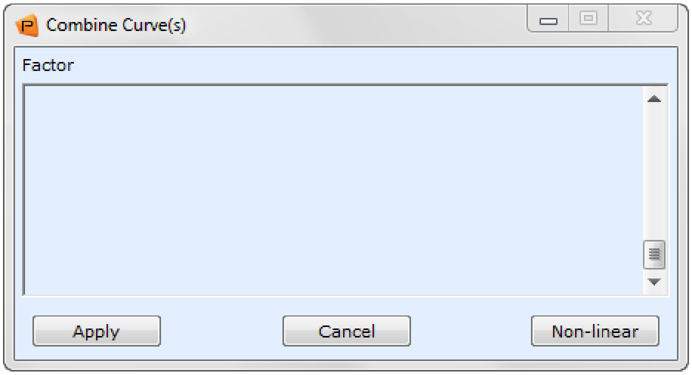


Parameter	Description
Filter	Filter updates the existing curve list by filtering with the indicated filter.
Existing Curves	Scratch is the default window. If this window is posted, a single click to the curve will post the curve to the scratch window. Any other window will not allow you this feature.
Combine	Brings up the form on the next page which helps to combine curves in the curves list box.
Windows	Brings up a window which can be used to create, post, unpost, and delete windows.
Normalize	Normalizes the axis of the plot, scaling to 1 or -1.
Filters	Brings up the form on the next page which helps to filter the curve data by using a SAE filter or a least square fit.

Parameter	Description
Par	Creates a parametric curve by combination of two separate curves into one curve. The Y-axis data of each curve is used for X-and Y-data respectively. This is only valid if two curves are placed in the curves listbox.
Reverse Par	Reverses the axes of a plot. Only valid for PAR.
Hardcopy	Pressing hardcopy writes the current window plot to a postscript file.
HC Setup	Brings up the print standard hardcopy setup form.

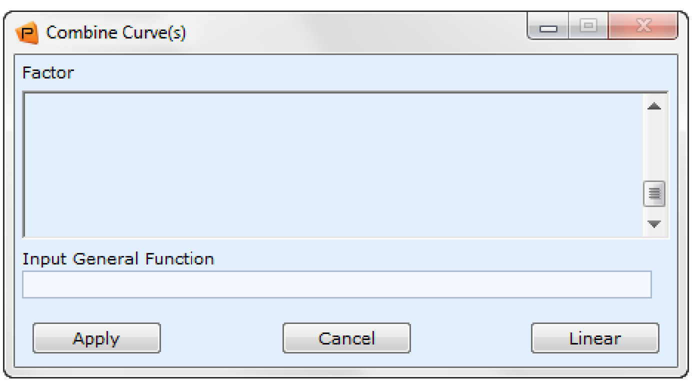
Combine Curve(s) Window

This form appears when the "Combine" button on the Time History form is depressed. It is used to define the scaling factors for the curves to be combined.



Parameter	Description
Linear	Toggles between linear and non-linear curve combination.

Upon selecting "Apply" a new curve, consisting of the linear combination of the selected curves, is created. The example above simply adds two separate curves. The "Non Linear" button invokes a databox in which a pcl function can be created to allow for non linear data manipulation. For non-linear combinations PCL-expressions may be used to create virtually any kind of combination. The individual components of the combined curve are indicated by %#%, where # represents a number. The following example illustrates how to create a combined curve from the square root of the sum of the squared components.



Parameter	Description
Linear	Toggles between linear and non-linear curve combination.

Curve Naming Convention for Contact

From v2001, the new curve naming convention enables you to find the results for the curves they are interested in quickly and without having to guess or refer back to the original input deck.

Old Curve Name: th_DMIN_co_1.curve1

“th_”	=	a constant and always present
“DMIN”	=	the variable being plotted
“co”	=	an abbreviation for “Contact”
“1”	=	a monotonically increasing integer assigned in the order in which the contacts are encountered
“curve1”	=	an arbitrary string assigned when the curves are read from the “.ths” file. The curve number is a monotonically increasing integer assigned in the order in which the curves are encountered.

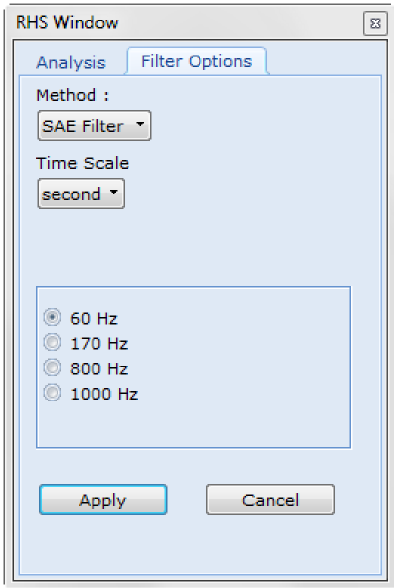
Because of the arbitrarily assigned integers in the above generated names, it was very difficult to correlate an output curve with the input data that it represents. The user had no control over the assignment of these arbitrary numbers.

New Curve Name: DMIN_CONTACT_5_CONT_DIS_3PLATE.curve1

“DMIN”	=	the variable being plotted
“CONTACT”	=	a master contact curve. Other possibilities here include “COSLAVE” for a slave contact, or “COTOTAL” for the sum of the master and slave contacts.
“5”	=	the contact number from the input deck that the user has assigned to this contact.
“CONT_DIS”	=	the user assigned case name.
“3PLATE”	=	the name job name of this run. Note that only that part of the job name up to the first underscore character will be used here. Any additional characters in the job name will be ignored.
“curve1”	=	an arbitrary identifier assigned by Patran in the order in which the curves are read from the archive file. This number will make the curve names unique if the same archive file is read in more than once.

Filter Option

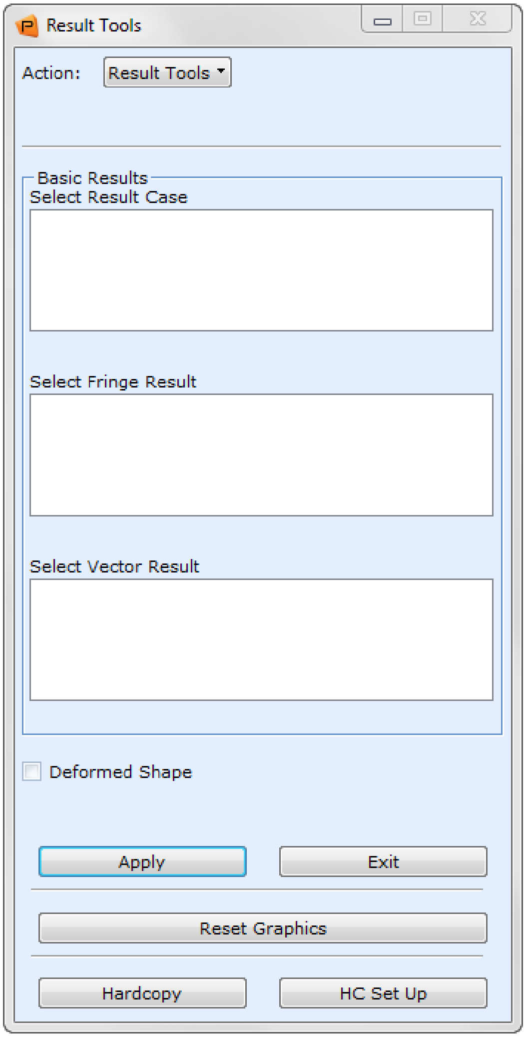
This form appears when the “Filter” button on the Time History form is depressed. It is used to the filtering to be applied to the time history data recovered from the archive files.



Parameter	Description
Method	Two Methods can be chosen: The SAE filter filters the data based on a specified frequency option. The least square fit option invokes the standard Patran least square fit options.
Time Scale	Sets the label for the time axis.

Mesh Plot Subsidiary Form

The mesh plot form allows the user to conveniently process the model results. Important advantages of using this tool are the possibilities of creating slide shows and animations.



Parameter	Description
Action	Options for Method are Results and Slide Show. These options are described in the following pages.
Select Result Case	Lists available result cases.
Select Fringe Result	Fringe results related to result case.
Select Vector Result	Vector results related to result case.
Deformed Shape	Switches Deformed Shape ON and OFF.
Reset Graphics	Cleans the graphics screen.

Parameter	Description
Hardcopy	Pressing Hardcopy makes a postscript file of the graphics window.
HC Set Up	Invokes standard Patran hardcopy setup.

Assembling an Animation from Separate Frames

If an animation is built from separate frames, the frames must have all the same size. The files must also be of the same type. If they are not, the program *dytranp3*, described in the next section, may provide a solution. If the files of the separate frames are used as input for the *dytranp3* to create an MPEG file, the numbering of the files will define the order in which the frames are processed. In some cases the numbering of the files may not be as desired. In such situation a simple shell script can help to renumber a large number of files.

DYTRANP3 Functions for Window Grabbing and MPEG Generation

DYTRANP3 is primarily the interface between Dytran and Patran. In addition it is the home of several utilities needed for the conversion of series of Patran generated image files into MPEG animations. Which utility is used depends on the first argument:

-grab	grab a window from the display and convert it into .ppm or .YUV format.
-ppm2yuv	convert a ppm file into a set of YUV files
-img2ppm	convert an .img file into .ppm format
-img2yuv	convert an .img file into YUV format
-ppm2mpg	create an MPEG animation from a series of .ppm files
-yuv2mpg	create an MPEG animation from a series of .yuv files
-img2mpg	create an MPEG animation from a series of .img files

The creation of an MPEG animation requires the executable of the "MPEG-codec" to be present in:

- Any directory included in your "search path"
- A directory specified by the environment variable MPEGDIR
- The directory $DYTRANDIR/patran, with DYTRANDIR an environment variable used to specify the root of the Dytran installation directory.

GRAB

The grab utility is used to grab the contents of an X11 window and write it out in .YUV or ppm format. Outside Patran it can be started with the command

dytranp3 -grab -id <window name> -o <outfile>

[-compr|-gzip] [-ppm] [-YUV] [-noborder] [-half]

With the arguments:

-grab	keyword used by dytranp3 to select this routine.
-id <window name>	specifies the name of the window to be dumped
-o <outfile>	specifies the name of the output file
-ppm	keyword to force output to be in “Portable PixMap” format.
-YUV	keyword to force output in “YUV” format (three separate files).
-noborder	strips 6 pixels from each border to get rid of the red border Patran generates around the current viewport.
-half	forces all frames to be halved on both directions for higher performance when using the MPEG-player.
-compr	forces output to be compressed by standard “compress”
-gzip	forces output to be compressed by GNU gzip

Using the “-compr” or “-gzip” option will save a lot of disk space. Since the ppm2mpg option will automatically detect compressed files, there is no need to uncompress .ppm files before they are used to create an MPEG file. The -half option is useful since typical output windows are of the order of magnitude of 640 x 480, while the size used for MPEG’s is usually in the order of 320 x 240. In theory MPEG-player can handle any size but large frames will decrease the performance of the player.

PPM2YUV

The -ppm2yuv utility is used to convert one single ppm file into a set of YUV files (the YUV format uses three separate files for one image).

Outside Patran it can be started with:

```
dytranp3 -ppm2yuv   file.ppm   [-half]
```

with

file.ppm	the name of the ppm file to be converted into YUV format
-half	forces all frames to be halved on both directions for a better performance when using the MPEG-player.

Remarks:

- Compressed files (with .Z or .gz) will be recognized, and decompressed automatically during read.

The extension .ppm will be replaced by the appropriate .Y, .U and .V extensions for the result files, provided no “postfix sequence” was applied. For example, example.10.ppm will result into example.10.Y etc., but example.ppm.10   will result into example.ppm.10.Y.

IMG2PPM

The -img2ppm utility is used to convert a single img file into ppm format.

Usage:

dytranp3 -img2ppm file.img [-half] [-compr | -gzip]

With arguments

file.img	file name of the existing .img file. Compressed files with .Z or .gz extensions will be recognized and decompressed during read. The current version does not recognize postfix sequence numbers. In that case “img” will not be stripped and “.ppm” will be appended to the full name.
-half	forces all frames to be halved in both directions for a better performance when using the MPEG-player.
-compr	forces the output ppm file to be compressed (only if the Linux compress program is in your search path).
-gzip	forces the output ppm file to be compressed by GNU zip, provided the executable is in your search path.

IMG2YUV

The -img2yuv utility is used to convert a single img file into a set of YUV files.

Usage:

dytranp3 -img2yuv file.yuv [-half] [-compr | -gzip]

With arguments

file.img	file name of the existing .img file. Compressed files with .Z or .gz extensions will be recognized and decompressed during read. The current version does not recognize postfix sequence numbers. In that case “img” will not be stripped and the “.Y”, “.U” and “.V” will be appended to the full name.
-half	forces all frames to be halved in both directions for better performance when using the MPEG-player.
-compr	forces the output YUV files to be compressed (only if the Linux compress program is in your search path).
-gzip	forces the output YUV file to be compressed by GNU gzip, provided the executable is in your search path.

Remarks:

- The MPEG codec does not support reading of compressed files, so when the compress options are used, the files should be decompressed before “yuv2mpg” is started.
- Don’t forget to make a note about the actual size of the frames (height and width in pixels), because the dimensions are not stored on the YUV files and have to be passed to “yuv2mpeg” by arguments.

PPM2MPG

Create an MPEG animation from a series of .ppm files.

Usage:

dytranp3 -ppm2mpg prefix ifirst ilast [-half] [-pad <N>] [-postnum]

With arguments

prefix	prefix of the files to be converted. The program dytranp3 expects the files to be present as: prefix.N.ppm or prefix.ppm.N (when “-postnum” was selected). with N being a sequence number in the range ifirst <= N <= ilast. Also compressed files with .Z and .gz extensions will be recognized and decompressed automatically, provided the compress or gzip executables are present in the users search path.
ifirst	sequence number of the first frame to be used.
ilast	sequence number of the last frame to be used.
-half	forces all frames to be halved in both directions for a better performance when using the MPEG-player.
-pad <N>	pad sequence numbers with leading zeros up to N digits. For example “-pad 3” will expect file names like example.007.ppm.
-postnum	Sequence numbers are expected to be appended to the file names: example.ppm.007 (if also “pad -3” was used)

The program will create a subdirectory named prefix_PID, with <PID> the process id of the current process used to force a unique name. In this directory it will create the YUV files needed by the “MPEG codec.” The “MPEG codec” is executed in a child process. When finished, the resulting MPEG file is moved to the directory from which the program was started and the “prefix_PID” scratch directory is removed.

IMG2MPG

Create an MPEG animation from a series of .img files.

Usage:

dytranp3 -img2mpg prefix ifirst ilast [-half] [-pad <N>] [-postnum]

With arguments

prefix	prefix of the files to be converted. The program dytranp3 expects the files to be present as: prefix.N.img or prefix.img.N (when “-postnum” was selected). with N being a sequence number in the range ifirst <= N <= ilast. Also compressed files with .Z and .gz extensions will be recognized and decompressed automatically, provided the compress or gzip executables are present in the users search path.
ifirst	sequence number of the first frame to be used.
ilast	sequence number of the last frame to be used.

-half	forces all frames to be halved on both directions for a better performance when using the MPEG-player.
-pad <N>	pad sequence numbers with leading zeros up to N digits. For example "-pad 3" will expect file names like example.007.img.
-postnum	Sequence numbers are expected to be appended to the file names: example.img.007 (if also "pad -3" was used)

The program will create a subdirectory named prefix_PID, with <PID> the process id of the current process, to force a unique name. In this directory it will create the YUV files needed by the "MPEG codec." The "MPEG codec" is executed in a child process, and when finished, the resulting MPEG file is moved to the directory from which the program was started and the "prefix_PID" scratch directory is removed.

YUV2MPG

Create an MPEG animation from a series of .yuv files

Usage:

dytranp3 -yuv2mpg prefix ifirst ilast hsize vsize [-pad <N>] [-postnum]

With arguments

prefix	prefix of the files to be converted. The program dytranp3 expects the files to be present as: prefix.N.[YUV] or prefix.[YUV].N (when "-postnum" was selected) with N being a sequence number in the range ifirst <= N <= ilast. Note that compressed files (.gz and .Z) cannot be handled here.
hsize	the horizontal size of the frames in pixels
vsize	the vertical size of the frames in pixels
-pad <N>	pad sequence numbers with leading zeros up to N digits. For example "-pad 3" will expect file names like example.007.Y
-postnum	Sequence numbers are expected to be appended to the file names: example.Y.007 (if also "pad -3" was used)

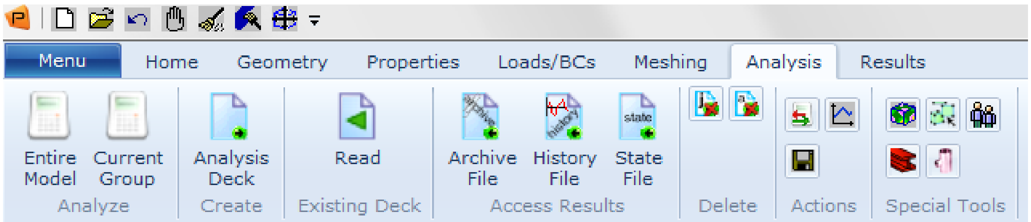
Results Created in Patran

All data available in the Dytran Archive and Time History files can be imported into Patran, where it is stored in the database and is accessible for processing using the full range of postprocessing tools.

5 Read Input File

Review of Read Input File Form

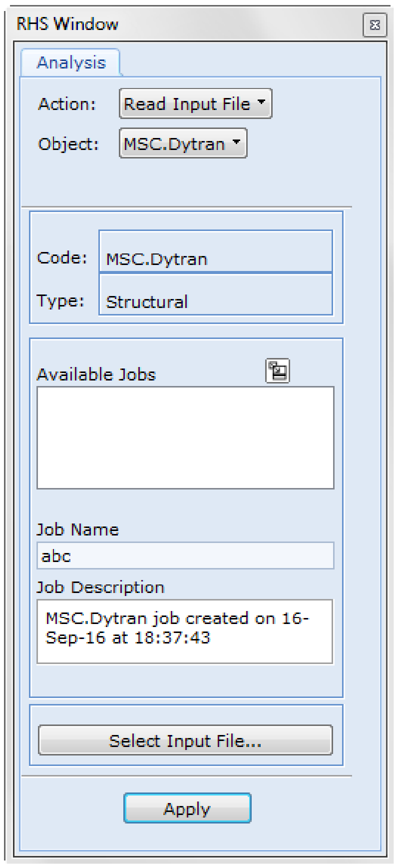
The Analysis form will appear when the Analysis toggle, located on the Patran Main form, is chosen.



Read Input File as the selected Action on the Analysis form allows some of the model data from Dytran, LS-DYNA3D and PAMCRASH input files to be translated into the Patran database. A subordinate File Selection form allows the user to specify the input file to translate. This form is described on the following pages.

Read Input File Form

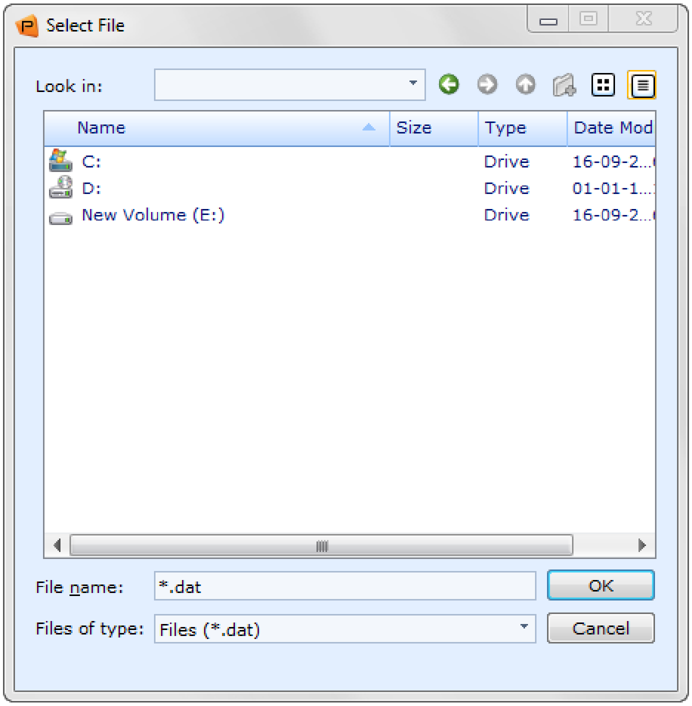
This form appears when the Analysis toggle is selected on the main form. Read Input File, as the selected Action, specifies that model data is to be translated from the specified Dytran input file into the Patran database.



Parameter	Description
Object	The Object can be Dytran, LS-DYNA3D, or PAMCRASH.
Code **Type**	Indicates the selected Analysis Code and Analysis Type, as defined in the Preferences>Analysis (p. 367) in the Patran Reference Manual, Part 1: Basic Functions.
Available Jobs	List of already existing jobs.
Job Name	Name assigned to current translation job. This job name will be used as the base file name for the message file.
Select Input File	Activates a subordinate File Select form which allows the user to specify the Dytran input file to be translated.

Selection of Input File

This subordinate form appears when the Select Input File button is selected on the Analysis form when Read Input File is the selected Action. It allows the user to specify which Dytran input file to translate.



Data Translated from the Dytran Input File

The Patran Dytran preference translator currently translates most of the bulk data entries from an input file. The following is a list of Dytran Bulk Data entries supported by the reader.

Entity Type	Dytran Bulk Data Entries
Nodes	GRID
Elements	CBAR, CBEAM, CDAMP1, CELAS1, CHEXA, CONM2, CPENTA, CQUAD4, CROD, CSPR, CTETRA, CTRIA3, CVISC
Material Models	DMAT, DMATEL, DMATEP, DMATOR, DYMAT14, DYMAT24, DYMAT25, DYMAT26, FABRIC, FOAM1, FOAM2, MAT1, MAT2, MAT8, MAT8A, MATRIG, RUBBER1, SHEETMAT
Yield Models	YLDEX, YLDJC, YLDMC, YLDMSS, YLDPOL, YLDRPL, YLDTM, YLDVM, YLDZA
Shear Models	SHREL, SHREX, SHRLVE, SHRPOL
Failure Models	FAILEST, FAILEX, FAILEX1, FAILMES, FAILMPS, FAILPRS, FAILSDT
Spallation Models	PMINC

Entity Type	Dytran Bulk Data Entries
Equation of State	EOSEX, EOSGAM, EOSIG, EOSJWL, EOSPOL, EOSTAIT
Element Properties	HGSUPPR, PBAR, PBCOMP, PBEAM, PBEAM1, PBEAML, PBELT, PCOMP, PCOMPA, PDAMP, PELAS, PELAS1, PELASEX, PEULER, PEULER1, PROD, PSHELL, PSHELL1, PSOLID, PSPR, PSPR1, PSPREX, PVISC, PVISC1, PVISCEX, PWELD, PWELD1, PWELD2
Coordinate Frames	CORD2C, CORD2R, CORD2S
Loads and Boundary Conditions	BJOIN, CFACE, CONTACT, CONTREL, COUHTR, COUINFL, COUOPT, COUP1FL, COUP1INT, COUPLE, COUPLE1, COUPOR, CYLINDER, DETSHP, FLOW, FORCE, FORCE1, FORCE2, GBAG, GBAGCOU, GBAGPOR, GBAGHTR, GBAGINFL, HTRCONV, HTRRAD, INFLATR, KJOIN, MOMENT, MOMENT1, MOMENT2, PERMEAB, PLOAD, PORFLOW, PORHOLE, RBE2, RBHINGE, RCONN, RELLIPS, RFORCE, SET1, SETC, SPC, SPC1, SPC2, SPC3, SPHERE, SUBSURF, SURFACE, TABLED1, TIC3, TICEL, TICEUL, TICGP, TICVAL, TLOAD1, WALL, WALLET
MPC Data	RBE2

Reject File

During import of the Dytran input file, some entries might not be understood by Patran. Those entries are written in the reject file *filename.dat.rej*.

Limitations

- **Reader for Analysis form entries.** All the entries written by the Analysis forms cannot be read back to Patran (Entries before BEGIN BULK).
- **Entries not supported by the reader.** The following entries which are supported by the writer, are not supported by the reader in the current version:
    - Late v2003 lbcs implementation: ALE, ALEGRID, ALEGRID1, FFCONTR, MATINI.
    - v2004 lbcs implementation: BODYFOR, RIGID, MESH, RJCYL, RJPLA, RJREV, RJSPH, RJTRA, RJUNI, CONTFORC, INFLATR1, INFLHYB, INFLHYB1, INFLTANK, INFLFRAC, INFLGAS, INITGAS, PORLHOLE, PERMGBG, PORFCPL, PORFGBG, PORFLCPL, PORFLGBG.
- **Entries not supported by the preference.** The following entries are not supported by the current version of Dytran Preference:

Section	Dytran Entries
Case Control	CORDDEF, PLANES, PLNOUT, SGAUGES, USASOUT, USASURFS
Bulk Data	ATBACC, ATBJNT, ATBSEG, BOX, CDAMP2, CELAS2, CFACE1, CONTINI, CORD1C, CORD1R, CORD1S, CORD3R, CORD4R, CORDROT, CSEG, DAREA, FLOWDEF, FLOWEX, FORCE3, FORCEEX, GBAGC, IGNORE, JOIN, MADGRP, PLOAD4, PLOADEX, POREX, RBC3, RCONREL, RELEX, RPLEX, SECTION, SGAUGE, TABLEEX, TIC, TIC1, TIC2, TICEEX, TICGEX, TLOAD2, USA, YLDHY
Parameters	ATBSEGCREATE, CLUMPENER, ENTROPY-FIX, ERRUSR, FAILDT, FLOW-METHOD, HGCOEFF, HGTYPE, HICGRAV, HYDROBOD, IGNFRCER, MATRMERG, OLDLAGTET, PARALLEL, PLCOVCUT, TOLCHK, USA_CAV

6 Files

Files

The Patran Dytran Preference uses or creates several files.The following table outlines each file and its uses. In the filename definition, **jobname** will be replaced with the jobname assigned by the user.

File Name	Description
∗.db	This is the Patran database. During an Analyze pass, model data is read from, and during a Read Results pass, model and/or results data is written into. This file typically resides in the current directory.
jobname.dat	This is the Dytran input file created by the interface. This file typically resides in the current directory.
jobname.dat.rej	This file contains any keywords and data not recognized by the translator that reads in the Dytran input files. This file typically resides in the current directory.
jobname.arc	This is the Dytran archive file which is read by the *Read Results* pass. This file typically resides in the current directory.
jobname.ths	This is the Dytran time history file which is read by the *Read Results* pass. This file typically resides in the current directory.
jobname.flat	This file may be generated during a *Read Results* pass. If the results translation cannot write data directly into the specified Patran database it will create this jobname flat file. This file typically resides in the current directory.

File Name	Description
MscDytranExecute	This is a Linux script file which is called on to submit the analysis file to Dytran after translation is complete. This file might need customizing with site specific data, such as, host machine name and Dytran executable commands. This file contains many comments and should be easy to edit. Patran searches its file path to find this file, but it typically resides in the <installation_directory>/bin/exe directory. Either use the general copy in <installation_directory>/bin/exe, or place a local copy in a directory on the file path which takes precedence over the <installation_directory>/bin/exe directory.
p3dytran	This is the actual translation program, translating between the Patran database and an Dytran input file. It is typically run within Patran, **transparent to the user,** but can also be run independently. For example: **`<installation_directory>/bin/exe/p3dytran -j my_job -d my_database.db > my_job.msg &`** Patran searches its file path for this file, but it typically resides in the <installation_directory>/bin/exe directory. Note that `i` is the option for translating from an Dytran input file to an Patran database.
dytranp3	This is the actual reverse translation program. It is typically run within Patran, transparent to the user, but can also be run independently with the following command, (executable name) (jobfile name) (optional redirection of output) (optional backgrounding of process). For example, **`<installation_directory>/bin/exe/dytranp3 my_job.jbr > my_job.msg &`** Patran searches its file path for this file, but it typically resides in the `<installation_directory>/bin/exe` directory.

Index

Patran Interface to Dytran Preference Guide