



Command Reference

## Copyright Notice

The Chestnut Pens 'CP' logo, this document and the Chestnut Pens Modelling Studio software are all copyright Chestnut Pens. [www.chestnutpens.co.uk](http://www.chestnutpens.co.uk)

'NISA' is a trademark of EMRC, Big Beaver, Troy, Michigan, USA. [www.emrc.com](http://www.emrc.com)

'MYSTRAN' is copyright Dr W Case, [www.mystran.com](http://www.mystran.com)

'Poser' was a trademark of e-frontier America Inc. 5615 Scotts Valley Drive, Suite 210, Scotts Valley, California, USA. [www.e-frontier.com](http://www.e-frontier.com) and is now owned by [www.renderosity.com](http://www.renderosity.com).

'DAZ| Studio' and 'Bryce' are trademarks of DAZ Productions, Inc. 12637 South 265 West #300, Draper, Utah USA. [www.daz3d.com](http://www.daz3d.com)

## Contents

1.	Introduction .....	4
2.	Command Line Reference .....	5
	2.2.1 Plotting Commands.....	8
	2.2.2 Erase Commands .....	20
	2.2.3 Colours.....	27
	2.2.4 Views, Lights, Snaps & Labels.....	28
	2.2.5 Selection Commands .....	31
	2.2.6 Deletion Commands.....	43
	2.2.7 Points .....	46
	2.2.8 Lines.....	49
	2.2.9 Surfaces .....	53
	2.2.10 Meshing Commands .....	57
	2.2.11 Executive Commands .....	59
	2.2.12 Nodes.....	61
	2.2.13 Elements .....	65
	2.2.14 Rigid Links.....	82
	2.2.15 Analysis Titles .....	84
	2.2.16 Materials.....	85
	2.2.17 Laminations.....	86
	2.2.18 Lamination Angles.....	87
	2.2.19 Properties.....	88
	2.2.20 Load Case Commands .....	89
	2.2.21 Nodal Displacements .....	90
	2.2.22 Pressures .....	93
	2.2.23 Nodal Pressures.....	96
	2.2.24 Forces .....	99
	2.2.25 Accelerations.....	102

2.2.26	Texture Mapping .....	103
2.2.27	Miscellaneous .....	104
2.2.28	Commands to fix bugs .....	106
3.	Menu Command Reference .....	107
3.2.1	File Menu .....	109
3.2.2	Edit Menu .....	111
3.2.3	View Menu .....	114
	View  Plot Menu .....	114
	View  Erase Menu .....	124
	View  Zoom Menu .....	134
	View  Angle Menu .....	134
	View  Perspective Menu .....	135
	View  Display Options Menu .....	136
	View  Colours Menu .....	139
	View  Labels Menu .....	144
	View  Light Source Menu .....	145
	View  Layers Menu .....	146
	View  Groups Menu .....	149
	View  Status .....	152
3.2.4	Selection Menu .....	153
3.2.5	Geometry Menu .....	164
3.2.6	FE Model Menu .....	174
	FE Model  Executive Commands .....	174
	FE Model  Nodes .....	177
	FE Model  Elements .....	180
	FE Model  Rigid Links .....	194
	FE Model  Analysis Title .....	196
	FE Model  Material .....	197
	FE Model  Laminations .....	200
	FE Model  Lamination Angles .....	203
	FE Model  Properties .....	205
	FE Model  Load Case .....	214
	FE Model  Displacement Constraints .....	217
	FE Model  Pressures .....	219
	FE Model  Nodal Pressures .....	221
	FE Model  Forces .....	223
	FE Model  Accelerations .....	225
	FE Model  Load Combiner .....	227
	FE Model  Analysis Readiness Check .....	229
3.2.7	Mapping Menu .....	231
3.2.8	Window Menu .....	236
3.2.9	Help Menu .....	237
3.2.10	Pop-up menu over model Viewport .....	238
4.	Interface Reference .....	240

## 1. Introduction

This document is the command reference intended to accompany the Chestnut Pens Modelling Studio program.

There are two principal sections to the document and a minor section. The principal sections are the command line command reference (section 2) and the menu command reference (section 3). The final section details the use of each button and active area on the Modelling Studio interface (section 4). Section 4 is, inevitably, a partial repeat of information contained in the Quick Start Guide.

Section 3 is probably the most useful as it covers the menu commands and all the dialogue windows that will be encountered. Also, there are some pictures, so it isn't quite so heavy going!

## 2. Command Line Reference

### 2.1 General Information

This section details every command that may be entered in the Modelling Studio command line. These commands are also applicable in the session file should it need to be edited.

**Note:** If all goes well, there should never be any need to resort to command line commands. So, don't worry if it all seems too much to learn and remember.

Throughout this section the minimum typed command is in **BOLD CAPITALS**. The command line is case insensitive, so there is no need to have any capitals anywhere.

Optional statements are in square brackets [ ].

Typed data entry in examples etc is in `Courier New` font.

As an example:-

```
NODE,ADD,nodenumber,[cs,x,y,z[,selected,visible,colour,layernumber]
```

This means that the command could be typed as

```
nod,add,1,1.5,1.1,0
```

Or as:-

```
Node,Add,1,1.5,1.1,0,0,1,16728345,1
```

Where a range is specified it's possible to use various delimiters. More than one integer may be specified separated by commas or slashes. Also a consecutive range may be indicated by a '-' or a 't'. In consecutive ranges make sure that the first number is smaller than the second. For example:-

```
Node,Delete,2/4,1,7t9,15-19
```

Indicates that nodes 1, 2, 4, 7, 8, 9, 15, 16, 17, 18 & 19 are to be deleted, whereas the following:-

```
Node,Delete,19-15
```

Is not acceptable.

Where a colour is specified, the number representing the colour can be calculated from the following RGB calculation. Assuming each channel can be represented by an integer in the range 0-255:-

$$\text{Colournumber} = \text{red} + \text{green} * 256 + \text{blue} * 256^2.$$

The following are the principal abbreviations that may be used:-

Abbreviation	Un-abbreviated form
Plo	Plot
Era	Erase
Sel	Select
Unsel	Unselect (for Deselect)
Rev	Reverse (for Invert)
As	Association
Del	Delete
Col	Colour
Boun	Boundary Conditions
FE	FE Model
Geo	Geometry
Lab	Label
Poi	Point
Lin	Line
Sur	Surface
Nod	Node
Ele	Element
Rig	Rigid Link
NDisp	NDisplacement for Nodal Displacement
Pres	Pressure
NPres	NPressure for Nodal Pressure
NFor	Force
BodyF	BodyForce for Acceleration

## 2.2 Command List

The commands are broken down into the categories listed below. Where a command falls into more than one category, it is repeated in all applicable categories.

2.2.1	Plotting Commands.....	8
2.2.2	Erase Commands .....	20
2.2.3	Colours.....	27
2.2.4	Views, Lights, Snaps & Labels.....	28
2.2.5	Selection Commands .....	31
2.2.6	Deletion Commands.....	43
2.2.7	Points .....	46
2.2.8	Lines.....	49
2.2.9	Surfaces .....	53
2.2.10	Meshing Commands .....	57
2.2.11	Executive Commands .....	59
2.2.12	Nodes .....	61
2.2.13	Elements .....	65
2.2.14	Rigid Links.....	82
2.2.15	Analysis Titles .....	84
2.2.16	Materials.....	85
2.2.17	Laminations .....	86
2.2.18	Lamination Angles.....	87
2.2.19	Properties .....	88
2.2.20	Load Case Commands .....	89
2.2.21	Nodal Displacements .....	90
2.2.22	Pressures .....	93
2.2.23	Nodal Pressures.....	96
2.2.24	Forces .....	99
2.2.25	Accelerations.....	102
2.2.26	Texture Mapping .....	103
2.2.27	Miscellaneous .....	104
2.2.28	Commands to cure bugs .....	106

### 2.2.1 Plotting Commands

The following plotting commands may be entered.

**PLOt**

Plot currently visible model at a zoom factor of 1.0

**PLOt,ALL**

Plot all entities.

**PLOt,BOUNDaryconditions,ALL**

Plot All Nodal Displacements, Pressures, Nodal Pressures and Forces.

**PLOt,BOUNDaryconditions,CASe,range**

Plot All Nodal Displacements, Pressures, Nodal Pressures and Forces with load case within 'range'.

**PLOt,FEmodel,ALL**

Plot all FE Model Entities.

**PLOt,GEOmetry,ALL**

Plot all Points, Lines, Surfaces and Solids

**PLOt,REFresh**

Refresh the screen.

**ERase,BOUNDaryconditions,NONE**

Plot All Nodal Displacements, Pressures, Nodal Pressures and Forces.

**ERase,ELEments,NONE**

Plot all elements. Non-preferred alternative to 'ELEMENT,PLOt,ALL'

**ERase,FEmodel,NONE**

Plot all FE Model Entities.

**ERase,GEOmetry,NONE**

Plot all Points, Lines, Surfaces and Solids

**ERase,NODE,NONE**

Plot all nodes. Non-preferred alternative to 'NODE,PLOt,ALL'

**POInt,ERase,NONE**

Plot all points

**POInt,PLOt,ALL**

Plot all points

**POInt,PLOt,ASsociation,LINe**

Plot points by association with currently selected lines.

**POInt,PLOt,ASsociation,SURface**

Plot points by association with currently selected surfaces.

**POInt,PLOt,SELected**

Plot selected points. Should be redundant as all selected points should be visible.

**POInt,PLOt,range**

Plot points within 'range'.

**POInt,VISible,ALL**

Plot all points



**POInt,VISible,ASsociation,LINE**

Plot points by association with currently selected lines.

**POInt,VISible,ASsociation,SURface**

Plot points by association with currently selected surfaces.

**POInt,VISible,SELEcted**

Plot selected points. Should be redundant as all selected points should be visible.

**POInt,VISible,range**

Plot points within 'range'.

**LINE,ERAsE,NONE**

Plot all lines.

**LINE,PLOt,ALL**

Plot all lines

**LINE,PLOt,ASsociation,POInt**

Plot all lines associated with currently selected points.

**LINE,PLOt,ASsociation,SURface**

Plot all lines associated with currently selected surfaces.

**LINE,PLOt,SELEcted**

Plot all selected lines. Should be redundant as all lines should be plotted as they are selected.

**LINE,PLOt,range**

Plot a range of lines.

**LINE,VISible,ALL**

Plot all lines

**LINE,VISible,ASsociation,POInt**

Plot all lines associated with currently selected points.

**LINE,VISible,ASsociation,SURface**

Plot all lines associated with currently selected surfaces.

**LINE,VISible,SELEcted**

Plot all selected lines. Should be redundant as all lines should be plotted as they are selected.

**LINE,VISible,range**

Plot a range of lines.

**SURface,ERAsE,NONE**

Plot all surfaces.

**SURface,PLOt,ALL**

Plot all surfaces

**SURface,PLOt,ASsociation,LINE**

Plot surfaces by association with currently selected lines.

**SURface,PLOt,ASsociation,POInt**

Plot surfaces by association with currently selected points.

**SURface,PLOt,SELEcted**

Plot selected surfaces. This command should be redundant.

**SURface,PLOt,range**

Plot surfaces within 'range'.

**SURface,VISible,ALL**

Plot all surfaces

**SURface,VISible,ASsociation,LINE**

Plot surfaces by association with currently selected lines.

**SURface,VISible,ASsociation,POInt**

Plot surfaces by association with currently selected points.

**SURface,VISible,SElected**

Plot selected surfaces. This command should be redundant.

**SURface,VISible,range**

Plot surfaces within 'range'.

**NODE,ERASE,NONE**

Plot all nodes.

**NODE,PLOt,ALL**

Plot all nodes.

**NODE,PLOt,ASsociation,ELEment**

Plot all nodes associated with the currently selected Elements

**NODE,PLOt,ASsociation,RIGidLink**

Plot all nodes associated with the currently selected Rigid Links

**NODE,PLOt,ASsociation,NDISPlacement**

Plot all nodes associated with the currently selected Displacement Constraints

**NODE,PLOt,ASsociation,PRESSure**

Plot all nodes associated with the currently selected Pressures

**NODE,PLOt,ASsociation,NPRESSure**

Plot all nodes associated with the currently selected Nodal Pressures

**NODE,PLOt,ASsociation,NFORce**

Plot all nodes associated with the currently selected Forces

**NODE,PLOt,SElected**

Plots all selected nodes. This should be redundant as all nodes should be made visible as they are selected.

**NODE,PLOt,range**

Plots a range of nodes.

**NODE,VISible,ALL**

Plot all nodes.

**NODE,VISible,ASsociation,ELEment**

Plot all nodes associated with the currently selected Elements

**NODE,VISible,ASsociation,RIGidLink**

Plot all nodes associated with the currently selected Rigid Links

**NODE,VISible,ASsociation,NDISPlacement**

Plot all nodes associated with the currently selected Displacement Constraints

**NODE,VISible,ASsociation,PRESSure**

Plot all nodes associated with the currently selected Pressures

**NODE,VISible,ASsociation,NPRESSure**

Plot all nodes associated with the currently selected Nodal Pressures

**NODE,VISible,ASsociation,NFORCE**

Plot all nodes associated with the currently selected Forces

**NODE,VISible,SElected**

Plots all selected nodes. This should be redundant as all nodes should be made visible as they are selected.

**NODE,VISible,range**

Plots a range of nodes.

**ELEment,ERASE,NONE**

Plot all elements

**ELEment,PLOT,ALL**

Plot all elements

**ELEment,PLOT,ASsociation,NODE**

Plot elements by association with selected nodes

**ELEment,PLOT,ASsociation,NDISplacements**

Plot Elements by association with selected Nodal Displacements

**ELEment,PLOT,ASsociation,NFORCE**

Plot Elements by association with selected Forces

**ELEment,PLOT,ASsociation,NPRESSure**

Plot Elements by association with selected Nodal Pressures

**ELEment,PLOT,ASsociation,PRESSure**

Plot Elements by association with selected Pressures

**ELEment,PLOT,ASsociation,RIGidLink**

Plot Elements by association with selected Rigid Links

**ELEment,PLOT,FORMulation,range**

Plot the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment,PLOT,MATerial,range**

Plot the elements by material range.

**ELEment,PLOT,NKTP,range**

Plot the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid

10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,PLOt,NORDR,range**  
Plot the elements by Node Order range.

**ELEment,PLOt,NSRL,range**  
Plot the elements by NSRL range.

**ELEment,PLOt,PROPerty,range**  
Plot the elements by property range.

**ELEment,PLOt,TYPE,range**  
Plot the elements by element type range.  
The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid

9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,PLOt,NONE**

Erase all elements

**ELEment,PLOt,SELected**

Plot selected elements

**ELEment,PLOt,range**

Plot element range.

**ELEment,VISible,ALL**

Plot all elements

**ELEment,VISible,ASsociation,NODE**

Plot elements by association with selected nodes

**ELEment,VISible,ASsociation,NDISPlacements**

Plot Elements by association with selected Nodal Displacements

**ELEment, VISible, ASSociation, NFORce**  
Plot Elements by association with selected Forces

**ELEment, VISible, ASSociation, NPRESSure**  
Plot Elements by association with selected Nodal Pressures

**ELEment, VISible, ASSociation, PRESSure**  
Plot Elements by association with selected Pressures

**ELEment, VISible, ASSociation, RIGidLink**  
Plot Elements by association with selected Rigid Links

**ELEment, VISible, FORMulation, range**  
Plot the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment, VISible, MATerial, range**  
Plot the elements by material range.

**ELEment, VISible, NKTP, range**  
Plot the elements by element type range.  
The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap

43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment, VISible, NORDR, range**  
Plot the elements by Node Order range.

**ELEment, VISible, NSRL, range**  
Plot the elements by NSRL range.

**ELEment, VISible, PROPerty, range**  
Plot the elements by property range.

**ELEment, VISible, TYPE, range**  
Plot the elements by element type range.  
The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap

43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment, VISible, SElected**

Plot selected elements

**ELEment, VISible, range**

Plot element range.

**RIGidlink, ERAse, NONe**

Plot All Rigid Links

**RIGidlink, PLOt, ALL**

Plot all rigid links

**RIGidlink, PLOt, SElected**

Plot all selected rigid links

**RIGidlink, PLOt, range**

Plot all rigid links in 'range'

**RIGidlink, VISible, ALL**

Plot all rigid links

**RIGidlink, VISible, SElected**

Plot all selected rigid links

**RIGidlink, VISible, range**

Plot all rigid links in 'range'

**NDISp, ERAse, NONe**

Plot all nodal displacements

**NDISp, PLOt, ALL**

Plot all nodal displacements

**NDISp, PLOt, ASSociation, NODE**

Plot nodal displacements by association with currently selected nodes.

**NDISp, PLOt, ASSociation, ELEment**

Plot nodal displacements by association with currently selected elements.

**NDISp, PLOt, CASE, range**

Plot all nodal displacements with load case within 'range'.

**NDISp, PLOt, INDividual, nodenumber, casenumber**

Plot nodal displacement on node 'nodenumber' for load case 'casenumber'.

**NDISp, PLOt, NODE, range**

Plot nodal displacements with nodenumber within 'range'.



**NDISp,VISible,ALL**

Plot all nodal displacements

**NDISp,VISible,ASSociation,NODE**

Plot nodal displacements by association with currently selected nodes.

**NDISp,VISible,ASSociation, ELEment**

Plot nodal displacements by association with currently selected elements.

**NDISp,VISible,CASe,range**

Plot all nodal displacements with load case within 'range'.

**NDISp,VISible,INDividual,nodenumbe,r,casenumbe,r**

Plot nodal displacement on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NDISp,VISible,NODE,range**

Plot nodal displacements with nodenumbe,r within 'range'.

**PRESsure,ERASE,NONE**

Plot all nodal pressure forces

**PRESsure,PLOT,ALL**

Plot all pressure forces

**PRESsure,PLOT,ASSociation,NODE**

Plot pressure forces by association with currently selected nodes.

**PRESsure,PLOT,ASSociation, ELEment**

Plot pressure forces by association with currently selected elements.

**PRESsure,PLOT,CASe,range**

Plot all pressure forces with load case within 'range'.

**PRESsure,PLOT,INDividual,elementnumbe,r,facenumbe,r,casenumbe,r**

Plot nodal pressure force on element 'elementnumbe,r', face 'facenumbe,r' for load case 'casenumbe,r'.

**PRESsure,PLOT,ELEment,range**

Plot pressure forces with element numbe,r within 'range'.

**PRESsure,VISible,ALL**

Plot all pressure forces

**PRESsure,VISible,ASSociation,NODE**

Plot pressures by association with currently selected nodes.

**PRESsure,VISible,ASSociation, ELEment**

Plot pressure forces by association with currently selected elements.

**PRESsure,VISible,CASe,range**

Plot all pressure forces with load case within 'range'.

**PRESsure,VISible,INDividual,elementnumbe,r,facenumbe,r,casenumbe,r**

Plot nodal pressure force on element 'elementnumbe,r', face 'facenumbe,r' for load case 'casenumbe,r'.

**PRESsure,VISible,NODE,range**

Plot pressure forces with nodenumbe,r within 'range'.

**NPRESsure,ERASE,NONE**

Plot all nodal pressure forces

**NPRESsure,PLOt,ALL**

Plot all nodal pressure forces

**NPRESsure,PLOt,ASSociation,NODE**

Plot nodal pressure forces by association with currently selected nodes.

**NPRESsure,PLOt,ASSociation, ELEment**

Plot nodal pressure forces by association with currently selected elements.

**NPRESsure,PLOt,CASe,range**

Plot all nodal pressure forces with load case within 'range'.

**NPRESsure,PLOt,INDividual,nodenumber,casenumber**

Plot nodal pressure force on node 'nodenumber' for load case 'casenumber'.

**NPRESsure,PLOt,NODE,range**

Plot nodal pressure forces with nodenumber within 'range'.

**NPRESsure,VISible,ALL**

Plot all nodal pressure forces

**NPRESsure,VISible,ASSociation,NODE**

Plot nodal pressure forces by association with currently selected nodes.

**NPRESsure,VISible,ASSociation, ELEment**

Plot nodal pressure forces by association with currently selected elements.

**NPRESsure,VISible,CASe,range**

Plot all nodal pressure forces with load case within 'range'.

**NPRESsure,VISible,INDividual,nodenumber,casenumber**

Plot nodal pressure force on node 'nodenumber' for load case 'casenumber'.

**NPRESsure,VISible,NODE,range**

Plot nodal pressure forces with nodenumber within 'range'.

**NFORce,ERASE,NONE**

Plot all forces

**NFORce,PLOt,ALL**

Plot all forces

**NFORce,PLOt,ASSociation,NODE**

Plot forces by association with currently selected nodes.

**NFORce,PLOt,ASSociation, ELEment**

Plot forces by association with currently selected elements.

**NFORce,PLOt,CASe,range**

Plot all forces with load case within 'range'.

**NFORce,PLOt,INDividual,nodenumber,casenumber**

Plot force on node 'nodenumber' for load case 'casenumber'.

**NFORce,PLOt,NODE,range**

Plot forces with nodenumber within 'range'.

**NFORce,VISible,ALL**

Plot all forces

**NFOR**ce,**VIS**ible,**ASS**ociation,**NODE**

Plot forces by association with currently selected nodes.

**NFOR**ce,**VIS**ible,**ASS**ociation, **ELE**ment

Plot forces by association with currently selected elements.

**NFOR**ce,**VIS**ible,**CAS**e,range

Plot all forces with load case within 'range'.

**NFOR**ce,**VIS**ible,**IND**ividual,nodenumbe,r,casenumbe,r

Plot force on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NFOR**ce,**VIS**ible,**NODE**,range

Plot forces with nodenumbe,r within 'range'.

**GRO**up,**PLO**t,groupname

Plot elements in group 'groupname'.

**GRO**up,**VIS**ible,groupname

Plot elements in group 'groupname'.

**LAY**er,**PLO**t,layername

Plot all entities on layer 'layername'.

**LAY**er,**VIS**ible,layername

Plot all entities on layer 'layername'.

### 2.2.2 Erase Commands

The following erase commands may be entered. When an entity is erased it is retained in the model (it is not deleted), the entity is not plotted. This has the effect of reducing extraneous clutter when examining a part of the model.

**ERase,ALL**

Erase all visible entities.

**ERase,BOUNDaryconditions,ALL**

Erase all Nodal Displacements, Pressures, Nodal Pressures and Forces.

**ERase,BOUNDaryconditions,CASe,range**

Erase all Nodal Displacements, Pressures, Nodal Pressures and Forces with load case number in 'range'.

**ERase,ELEments,ALL**

Erase all elements. Non-preferred alternative to 'ELEment,ERase,ALL'

**ERase,ELEments,SELected**

Select all elements. Non-preferred alternative to 'ELEment,ERase,SELected'

**ERase,FEmodel,ALL**

Erase all FE model Entities, leaving only points, lines, surfaces and solids visible.

**ERase,FEmodel,NONe**

Plot all FE Model Entities.

**ERase,GEOmetry,ALL**

Erase all Points, Lines, Surfaces and Solids

**ERase,NODe,ALL**

Erase all nodes. Non-preferred alternative to 'NODe,ERase,ALL'

**ERase,NODe,SELected**

Erase selected nodes. Non-preferred alternative to 'NODe,ERase,SELected'

**ERase,SELected**

Erase selected entities.

**PLOt,BOUNDaryconditions,NONe**

Erase all Nodal Displacements, Pressures, Nodal Pressures and Forces.

**PLOt,FEmodel,NONe**

Erase all FE model Entities, leaving only points, lines, surfaces and solids visible.

**PLOt,GEOmetry,NONe**

Erase all Points, Lines, Surfaces and Solids

**POInt,ERase,ALL**

Erase all points

**POInt,ERase,ASsociation,LINe**

Erase points by association with currently selected lines.

**POInt,ERase,ASsociation,SURface**

Erase points by association with currently selected surfaces.

**POInt,ERase,SELected**

Erase selected points.

**POInt,ERAsE,range**  
Erase all points within 'range'.

**POInt,PLOt,NONe**  
Erase all points.

**POInt,VISible,NONe**  
Erase all points.

**LINE,ERA,ALL**  
Erase all lines.

**LINE,ERAsE,ASsociation,POInt**  
Erase lines associate with currently selected points.

**LINE,ERAsE,ASsociation,SURface**  
Erase lines associated with currently selected surfaces.

**LINE,ERAsE,SELected**  
Erase selected lines.

**LINE,ERAsE,range**  
Erase a range of lines.

**LINE,PLOt,NONe**  
Erase all lines

**LINE,VISible,NONe**  
Erase all lines

**SURface,ERAsE,ALL**  
Erase all surfaces.

**SURface,ERAsE,ASsociation,LINE**  
Erase surfaces associated with currently selected lines.

**SURface,ERAsE,ASsociation,POInt**  
Erase surfaces associated with currently selected points.

**SURface,ERAsE,SELected**  
Erase selected surfaces.

**SURface,ERAsE,range**  
Erase all surfaces within 'range'.

**SURface,PLOt,NONe**  
Erase all surfaces.

**SURface,VISible,NONe**  
Erase all surfaces.

**NODE,ERAsE,ALL**  
Erase all nodes.

**NODE,ERAsE,ASsociation,ELEment**  
Erase all nodes associated with the currently selected Elements.

**NODE,ERAsE,ASsociation,RIGidLink**  
Erase all nodes associated with the currently selected Rigid Links

**NODE,ERASE,ASSOCIATION,NDISPLACEMENT**

Erase all nodes associated with the currently selected Displacement Constraints

**NODE,ERASE,ASSOCIATION,PRESSURE**

Erase all nodes associated with the currently selected Pressures

**NODE,ERASE,ASSOCIATION,NPRESSURE**

Erase all nodes associated with the currently selected Nodal Pressures

**NODE,ERASE,ASSOCIATION,NFORCE**

Erase all nodes associated with the currently selected Forces

**NODE,ERASE,SELECTED**

Erase selected nodes.

**NODE,ERASE,RANGE**

Erases a range of node numbers.

**NODE,PLOT,NONE**

Erase all nodes

**NODE,VISIBLE,NONE**

Erase all nodes

**ELEMENT,ERASE,ALL**

Erase all elements

**ELEMENT,ERASE,ASSOCIATION,NODE**

Erase elements associated with selected nodes.

**ELEMENT,ERASE,ASSOCIATION,NDISPLACEMENTS**

Erase Elements by association with selected Nodal Displacements

**ELEMENT,ERASE,ASSOCIATION,NFORCE**

Erase Elements by association with selected Forces

**ELEMENT,ERASE,ASSOCIATION,NPRESSURE**

Erase t Elements by association with selected Nodal Pressures

**ELEMENT,ERASE,ASSOCIATION,PRESSURE**

Erase Elements by association with selected Pressures

**ELEMENT,ERASE,ASSOCIATION,RIGIDLINK**

Erase elements by association with selected Rigid Links

**ELEMENT,ERASE,SELECTED**

Erase selected elements

**ELEMENT,ERASE,FORMULATION,RANGE**

Erase the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEMENT,ERASE,MATERIAL,RANGE**

Erase all elements with material number specified in 'range'

**ELEMENT,ERASE,NORDR,RANGE**

Erase all elements with Node Order number specified in 'range'

**ELEment,ERASE,NSRL,range**

Erase all elements with NSRL number specified in 'range'

**ELEment,ERASE,PROPERTY,range**

Erase all elements with property number specified in 'range'

**ELEment,ERASE,TYPE,range**

Erase all elements with element type number specified in 'range'

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,ERASE,range**

Erase all elements within 'range'

**ELEment,VISible,NONe**  
Erase all elements

**RIGidlink,ERASE,ALL**  
Erase all Rigid Links

**RIGidlink,ERASE,SElected**  
Erase selected Rigid Links

**RIGidlink,ERASE,..range..**  
Erase all Rigid Links in 'range'

**RIGidlink,PLOt,NONe**  
Erase all rigid links

**RIGidlink,VISible,NONe**  
Erase all rigid links

**NDISp,ERASE,ALL**  
Erase all nodal displacements

**NDISp,ERASE,ASSociation,NODE**  
Erase all nodal displacements associated with currently selected nodes. Any selected nodal displacements are deselected as they are erased.

**NDISp,ERASE,ASSociation, ELEment**  
Erase all nodal displacements associated with currently selected elements. Any selected nodal displacements are deselected as they are erased.

**NDISp,ERASE,CASE,range**  
Erase all nodal displacements with load case numbers within 'range'

**NDISp,ERASE,INDividual,nodenumbe,r,casenumbe,r**  
Erase nodal displacement on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NDISp,ERASE,NODE,range**  
Erase all nodal displacements on nodes with nodenumbe,r within 'range'.

**NDISp,ERASE,SElected**  
Erase selected nodal displacements

**NDISp,PLOt,NONe**  
Erase all nodal displacements

**NDISp,VISible,NONe**  
Erase all nodal displacements

**PRESsure,ERASE,ALL**  
Erase all pressure forces

**PRESsure,ERASE,ASSociation,NODE**  
Erase all pressures associated with currently selected nodes. Any selected pressures are deselected as they are erased.

**PRESsure,ERASE,ASSociation, ELEment**  
Erase all pressure forces associated with currently selected elements. Any selected forces are deselected as they are erased.



**PRESsure,ERAsE,CASe,range**

Erase all pressure forces with load case numbers within 'range'

**PRESsure,ERAsE,INDividual,elementnumber,face,case,range**

Erase pressure force on element 'elementnumber', face 'face' for load case 'case'.

**PRESsure,ERAsE,ELEment,range**

Erase all pressures on elements with element numbers within 'range'.

**PRESsure,ERAsE,SELEcted**

Erase selected pressure forces

**PRESsure,PLOt,NONE**

Erase all pressure forces

**PRESsure,VISible,NONE**

Erase all pressure forces

**NPRESsure,ERAsE,ALL**

Erase all nodal pressure forces

**NPRESsure,ERAsE,ASSociation,NODE**

Erase all nodal pressure forces associated with currently selected nodes. Any selected forces are deselected as they are erased.

**NPRESsure,ERAsE,ASSociation, ELEment**

Erase all nodal pressure forces associated with currently selected elements. Any selected forces are deselected as they are erased.

**NPRESsure,ERAsE,CASe,range**

Erase all nodal pressure forces with load case numbers within 'range'

**NPRESsure,ERAsE,INDividual,nodenum,case,range**

Erase nodal pressure force on node 'nodenum' for load case 'case'.

**NPRESsure,ERAsE,NODE,range**

Erase all forces on nodes with nodenumbers within 'range'.

**NPRESsure,ERAsE,SELEcted**

Erase selected nodal pressure forces

**NPRESsure,PLOt,NONE**

Erase all nodal pressure forces

**NPRESsure,VISible,NONE**

Erase all nodal pressure forces

**NFORce,ERAsE,ALL**

Erase all forces

**NFORce,ERAsE,ASSociation,NODE**

Erase all forces associated with currently selected nodes. Any selected forces are deselected as they are erased.

**NFORce,ERAsE,ASSociation, ELEment**

Erase all forces associated with currently selected elements. Any selected forces are deselected as they are erased.

**NFORce,ERAsE,CASe,range**

Erase all forces with load case numbers within 'range'

**NFOR**ce,**ER**Ase,**IND**ividual,nodenumbe,r,casenumbe,r  
Erase force on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NFOR**ce,**ER**Ase,**NOD**e,range  
Erase all forces on nodes with nodenumbe,r within 'range'.

**NFOR**ce,**ER**Ase,**SEL**ected  
Erase selected forces

**NFOR**ce,**PLO**t,**NON**e  
Erase all forces

**NFOR**ce,**VIS**ible,**NON**e  
Erase all forces

**GRO**up,**ER**Ase,groupname  
Erase elements in 'groupname'

**LAY**er,**ER**Ase,layername  
Erase all entities on layer 'layername'.

### 2.2.3 Colours

The following commands relate to the colours of the model.

**SNAP,COLOUR,number**  
Sets the colour of the snaps to 'number'

**POINT,COLOUR**  
Opens a dialogue box to specify the colours of points.

**POINT,COLOUR,pointcolour,pointselectedcolour**  
Sets the colour of points when not selected and when selected.

**LINE,COLOUR**  
Opens a dialogue box to set the colours of lines.

**LINE,COLOUR,linecolour,lineselectedcolour**  
Sets the colour of lines and the colour of lines when they are selected.

**SURFACE,COLOUR**  
Opens a dialogue box to set the colours of surfaces.

**SURFACE,COLOUR,surfacecolour,surfaceoutlinecolour**  
Sets the colours of surfaces and selected surfaces.

**SURFACE,FILLtype,DEPTH**  
Change the surface fill colour to depth shaded.

**SURFACE,FILLtype,SHaded**  
Change the surface fill colour to infinite light shaded.

**SURFACE,FILLtype,ID**  
Set the surface fill colour to be by ID number.

**NODE,COLOUR**  
Set node colour by opening dialogue box.

**NODE,COLOUR,ALL,colour**  
Change the colour of all nodes to 'colour'.

**ELEMENT,COLOUR**  
Set the colour of the selected elements

**ELEMENT,COLOURLIST**  
Opens a dialogue window to set the sixteen colours used to differentiate between materials, groups, layers etc.

**ELEMENT,COLOURLIST,listnumber,colour**  
Sets colour list 'listnumber' to 'colour'.

**ELEMENT,COLOUR,SElected**  
Set the colour of the selected elements

**ELEMENT,COLOUR,elementnumber,colour**  
Set the colour of element number 'elementnumber' to 'colour'

**ELEMENT,FILLcolour**  
Set the element fill colour to be by face number. Seemed a good idea at the time.

**RIGIDLINK,COLOUR**  
Opens a dialogue box to specify the rigid link colour.

## 2.2.4 Views, Lights, Snaps and Labels

The following commands relate to changing the view, lights, snaps or labels.

**LABels,state**

Sets all labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,ALL,state**

Sets all labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,POInt,state**

Sets point labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,LINE,state**

Sets line labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,SURfaces,state**

Sets surface labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,SOLids,state**

Sets solids labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,NODes,state**

Sets node labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,ELEments,state**

Sets element labels to 'state', where 'state' is 'ON' or 'OFF'.

**LABels,RIGidlinks,state**

Sets rigid link labels to 'state', where 'state' is 'ON' or 'OFF'.

**LIGHT,BOTTOMLEFT**

Sets the light position for infinite light shading of elements to the bottom left of the view.

**LIGHT,BOTTOMCENTRE**

Sets the light position for infinite light shading of elements to the bottom centre of the view.

**LIGHT,BOTTOMRIGHT**

Sets the light position for infinite light shading of elements to the bottom right of the view.

**LIGHT,CENTRELEFT**

Sets the light position for infinite light shading of elements to the centre left of the view.

**LIGHT,CENTRECENTRE**

Sets the light position for infinite light shading of elements to the centre of the view.

**LIGHT,CENTRERIGHT**

Sets the light position for infinite light shading of elements to the centre right of the view.

**LIGHT,TOPLEFT**

Sets the light position for infinite light shading of elements to the top left of the view.

**LIGHT,TOPCENTRE**

Sets the light position for infinite light shading of elements to the top centre of the view.

**LIGHT,TOPRIGHT**

Sets the light position for infinite light shading of elements to the top right of the view.

**PAN,x,y**

Pan the view x pixels to the right and y pixels down.

**PERspective,None**

View the model with no perspective

**PERspective,Wide**

View the model with perspective simulating a wide angle lens.

**PERspective,Standard**

View the model with perspective simulating a standard lens.

**PERspective,Telescopic**

View the model with perspective simulating a telescopic lens.

**SNAP,POint,state**

Set the screen snapping for points to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,LINe,End,state**

Set the screen snapping for line ends to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,LINe,Middle,state**

Set the screen snapping for line mid-points to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,LINe,Nearest,state**

Set the screen snapping for line nearest points to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,LINe,Perpendicular,state**

Set the screen snapping for line perpendicular points to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,SURface,CENTroid,state**

Set the screen snapping for surface centroids to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,NODE,state**

Set the screen snapping for nodes to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,ELEment,state**

Set the screen snapping for element centroids to 'state', where 'state' is 'ON' or 'OFF'.

**SNAP,COLour,number**

Sets the colour of the snaps to 'number'

**View,Front**

View the model from the front.

**View,BAck**

View the model from the back.

**View,Left**

View the model from the left.

**View,Right**

View the model from the right.

**View,Top**

View the model from the top.

**View,BOTTOM**

View the model from the bottom.

**View,ISO1**

View the model from the ISO1 view.

**View,ISO2**

View the model from the ISO2 view.

**View,ISO3**

View the model from the ISO3 view.

**View,ISO4**

View the model from the ISO4 view.

**View,Userdefined**

Open a dialogue window to enter user defined view.

**View,Userdefined, anglex,angley,anglez**

User defined view with rotations 'anglex,angley,anglez' about the X, Y and Z axes.

**Zoom,ALL**

View entire visible model.

**Zoom,ARea,left,bottom,right,top**

Zoom to rectangle defined by pixel co-ordinates 'left,bottom,right,top'

**Zoom,Extents**

View entire visible model.

**Zoom,In,x,y**

Zoom in to model, expanding all dimensions by 15% centred about x and y in pixel co-ordinates from the top left of the viewport. X is horizontal, Y is down.

**Zoom,Out,x,y**

Zoom out from model, shrinking all dimensions by 15% centred about x and y in pixel co-ordinates from the top left of the viewport. X is horizontal, Y is down.

**Zoom,Previous**

Zoom to previous view.

## 2.2.5 Selection Commands

The following commands relate to Selecting/Deselecting entities.

**SElect,ALL**

Select all entities.

**SElect,BOUNDaryconditions,ALL**

Select all Displacement constraints, Pressures, Nodal Pressures and Forces.

**SElect,BOUNDaryconditions,CASe,range**

Select all Displacement constraints, Pressures, Nodal Pressures and Forces with load case number within 'range'.

**SElect,BOUNDaryconditions,NONE**

Deselect all Displacement constraints, Pressures, Nodal Pressures and Forces.

**SElect,BOUNDaryconditions,REVerse**

Invert selection of all Displacement constraints, Pressures, Nodal Pressures and Forces.

**SElect,FEmodel,ALL**

Select all Nodes, Elements, Rigid Links, Displacement constraints, Pressures, Nodal Pressures and Forces.

**SElect,FEmodel,NONE**

Deselect all Nodes, Elements, Rigid Links, Displacement constraints, Pressures, Nodal Pressures and Forces.

**SElect,FLAG,state**

Set the 'DeSelect After Operation' flag to 'state', where 'state' is 'ON' or 'OFF'.

**SElect,GEOmetry,ALL**

Select all points, lines and surfaces.

**SElect,GEOmetry,NONE**

Deselect all points, lines and surfaces.

**SElection,GROup,groupname**

Select elements in group 'groupname'.

**SElection,LAYer,layername**

Select all entities in layer 'layername'

**SElect,NONE**

Deselect all entities.

**SElection,NOT**

Invert selection of all entities.

**SElection,REVerse**

Invert selection of all entities.

**POInt,SElect,ALL**

Select all points.

**POInt,SElect,ASsociation,LINe**

Select points by association with currently selected lines.

**POInt,SElect,ASsociation,SURface**

Select points by association with currently selected surfaces.

**POInt,SElect,NONE**

Deselect all points

**POInt,SELEct,NOT**

Invert point selection.

**POInt,SELEct,REVErse**

Invert point selection.

**POInt,SELEct,range**

Select all points within 'range'.

**POInt,UNSELEct,ALL**

Deselect all points.

**POInt,UNSELEct,ASsociation,LINE**

Deselect points by association with currently selected lines.

**POInt,UNSELEct,ASsociation,SURface**

Deselect points by association with currently selected surfaces.

**POInt,UNSELEct,NONE**

Select all points

**POInt,UNSELEct,NOT**

Invert point selection.

**POInt,UNSELEct,REVErse**

Invert point selection.

**POInt,UNSELEct,range**

Deselect all points within 'range'.

**LINE,SELEct,ALL**

Select all lines.

**LINE,SELEct,ASsociation,POInt**

Select lines by association with currently selected points.

**LINE,SELEct,ASsociation,SURface**

Select lines by association with currently selected surfaces.

**LINE,SELEct,NONE**

Deselect all lines.

**LINE,SELEct,NOT**

Invert line selection.

**LINE,SELEct,REVErse**

Invert line selection.

**LINE,SELEct,range**

Select lines that fall within 'range'.

**LINE,UNSELEct,ALL**

Deselect all lines

**LINE,UNSELEct,ASsociation,POInt**

Deselect lines by association with currently selected points.

**LINE,UNSELEct,ASsociation,SURface**

Deselect lines by association with currently selected surfaces.



**LINE,UNSElect,NONE**

Select all lines.

**LINE,UNSElect,NOT**

Invert line selection.

**LINE,UNSElect,REVerse**

Invert line selection.

**LINE,UNSElect,range**

Deselect a range of lines.

**SURface,SELEct,ALL**

Select all surfaces.

**SURface,SELEct,ASsociation,LINE**

Select surfaces by association with currently selected lines.

**SURface,SELEct,ASsociation,POInt**

Select surfaces by association with currently selected points.

**SURface,SELEct,NONE**

Deselect all surfaces.

**SURface,SELEct,NOT**

Invert surface selection.

**SURface,SELEct,REVerse**

Invert surface selection.

**SURface,SELEct,range**

Select surfaces within 'range'.

**SURface,UNSElect,ALL**

Deselect all surfaces.

**SURface,UNSElect,ASsociation,LINE**

Deselect surfaces by association with currently selected lines.

**SURface,UNSElect,ASsociation,POInt**

Deselect surfaces by association with currently selected points.

**SURface,UNSElect,NONE**

Select all surfaces.

**SURface,UNSElect,NOT**

Invert surface selection.

**SURface,UNSElect,REVerse**

Invert surface selection.

**SURface,UNSElect,range**

Deselect surfaces within 'range'.

**NODE,SELEct,ALL**

Select all nodes. If nodes are invisible prior to being selected, they are made visible.

**NODE,SELEct,ASsociation,ELEment**

Select all nodes associated with the currently selected Elements. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElect,ASsociation,RIGidLink**

Select all nodes associated with the currently selected Rigid Links. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElect,ASsociation,NDISPlacement**

Select all nodes associated with the currently selected Displacement Constraints

**NODE,SElect,ASsociation,PRESSure**

Select all nodes associated with the currently selected Pressures

**NODE,SElect,ASsociation,NPRESSure**

Select all nodes associated with the currently selected Nodal Pressures

**NODE,SElect,ASsociation,NFORce**

Select all nodes associated with the currently selected Forces

**NODE,SElected,FALse,nodenumber**

Deselect node 'nodenumber'

**NODE,SElect,NONe**

Deselect all nodes.

**NODE,SElect,NOT**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to '**NODE,SElect,REVerse**'

**NODE,SElect,REVerse**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to '**NODE,SElect,NOT**'

**NODE,SElect,range**

Selects a range of node numbers. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElected,TRUE,nodenumber**

Select node 'nodenumber'. If node is invisible prior to being selected, it is made visible.

**NODE,UNSElect,ALL**

Deselect all nodes.

**NODE,UNSElect,ASsociation,ELEment**

Deselect all nodes associated with the currently selected Elements. If nodes are invisible prior to being selected, they are made visible.

**NODE,UNSElect,ASsociation,RIGidLink**

Deselect all nodes associated with the currently selected Rigid Links. If nodes are invisible prior to being selected, they are made visible.

**NODE,UNSElect,ASsociation,NDISPlacement**

Deselect all nodes associated with the currently selected Displacement Constraints

**NODE,UNSElect,ASsociation,PRESSure**

Deselect all nodes associated with the currently selected Pressures

**NODE,UNSElect,ASsociation,NPRESSure**

Deselect all nodes associated with the currently selected Nodal Pressures

**NODE,UNSElect,ASsociation,NFORce**

Deselect all nodes associated with the currently selected Forces

**NODE,UNSElect,NONe**

Select all nodes.

**NODe,UNSElect,NOT**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODe,UNSElect,REVERSE'

**NODe,UNSElect,REVERSE**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODe,UNSElect,NOT'

**NODe,UNSElect,range**

Deselects a range of node numbers.

**ELEment,SElect,ALL**

Select all elements. Any selected elements initially invisible are plotted.

**ELEment,SElect,ASsociation,NODe**

Select elements associated with selected nodes. Any selected elements initially invisible are plotted.

**ELEment,SElect,ASsociation,NDISPlacements**

Select Elements by association with selected Nodal Displacements

**ELEment,SElect,ASsociation,NFORce**

Select Elements by association with selected Forces

**ELEment,SElect,ASsociation,NPRESSure**

Select Elements by association with selected Nodal Pressures

**ELEment,SElect,ASsociation,PRESSure**

Select Elements by association with selected Pressures

**ELEment,SElect,ASsociation,RIGidLink**

Select elements associated with selected Rigid Links. Any selected elements initially invisible are plotted.

**ELEment,SElect,FORMulation,range**

Select the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

Any selected elements initially invisible are plotted.

**ELEment, SElect, DUPLIcates**

Select all duplicated elements – whether forward or reverse nodal definitions.

**ELEment,SElect,MATerial,range**

Select all elements with material in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElect,NKTP,range**

Select all elements with element type in 'range'. Any selected elements initially invisible are plotted.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam

12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,SElect,NONe**

Deselect all elements

**ELEment,SElect,NORDR,range**

Select all elements with Node Order in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElection,NOT**

Invert the selection of the element. Any selected elements initially invisible are plotted.

**ELEment,SElect,NSRL,range**

Select all elements with NSRL in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElect,PROPerty,range**

Select all elements with property in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElection,REVerse**

Invert the selection of the element. Any selected elements initially invisible are plotted.

**ELEment,SElect,TYPE,range**

Select all elements with element type in 'range'. Any selected elements initially invisible are plotted.

The element type is the NISA NKTP number and is as below:-

Type	Description
------	-------------

1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,SElect,range**

Select all elements in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElection,SHApe, range**

Invert the selection of the element. Any selected elements initially invisible are plotted.

The element shape is as below:-

Shape	Description
1	Point Element
2	Line Element
3	Axisymmetric Shell Element
4	Axisymmetric Solid Element
5	Heat Transfer Element
6	Planar Element

- 7 Shell Element
- 8 Solid Element

**ELEment,UNSELEct,ALL**

Deselect all elements

**ELEment,UNSELEct,ASsociation,NODE**

Deselect elements associated with selected nodes. Any selected elements initially invisible are plotted.

**ELEment,UNSELEct,ASsociation,NDISPlacements**

Deselect Elements by association with selected Nodal Displacements

**ELEment,UNSELEct,ASsociation,NFORce**

Deselect Elements by association with selected Forces

**ELEment,UNSELEct,ASsociation,NPRESSure**

Deselect Elements by association with selected Nodal Pressures

**ELEment,UNSELEct,ASsociation,PRESSure**

Deselect Elements by association with selected Pressures

**ELEment,UNSELEct,ASsociation,RIGidLink**

Deselect elements associated with selected Rigid Links. Any selected elements initially invisible are plotted.

**ELEment,UNSELEct,FORMulation,range**

Deselect the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment,UNSELEct,MATERial,range**

Deselect the elements by material range.

**ELEment,UNSELEct,NONE**

Select all elements.

**ELEment,UNSELEct,REVERSE/NOT**

Invert element selection. Any selected elements initially invisible are plotted.

**ELEment,UNSELEct,NORDR,range**

Deselect the elements by Node Order range.

**ELEment,UNSELEct,NSRL,range**

Deselect the elements by nsrl range.

**ELEment,UNSELEct,PROPERty,range**

Deselect the elements by property range.

**ELEment,UNSELEct,TYPE/NKTP,range**

Deselect the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element

7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,UNSElect,range**  
Deselect the elements by range.

**RIGidlink,SElect,ALL**  
Select all rigid links.

**RIGidlink,SElect,NONe**  
Deselect all Rigid Links

**RIGidlink,SElect, NOT**  
Invert Rigid Link Selection

**RIGidlink,SElect,REVerse**  
Invert Rigid Link Selection

**RIGidlink,SElect,range**  
Select all rigid links within 'range'

**RIGidlink,UNSElect,ALL**

Deselect all rigid links

**RIGidlink,UNSElect,NONE**

Select all rigid links

**RIGidlink,UNSElect, NOT**

Invert rigid link selection

**RIGidlink,UNSElect,REVerse**

Invert rigid link selection

**RIGidlink,UNSElect,range**

Deselect all rigid links within 'range'

**NDISp,SElect,ALL**

Select all nodal displacements

**NDISp,SElect,ASSociation,NODE**

Select nodal displacements by association with currently selected nodes.

**NDISp,SElect,ASSociation,ELEment**

Select nodal displacements by association with currently selected elements.

**NDISp,SElect,CASe,range**

Select all nodal displacements with load case within 'range'.

**NDISp,SElect,INDividual,nodenumbe,r,casenumbe,r**

Select nodal displacement on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NDISp,SElect,NODE,range**

Select nodal displacements with nodenumbe,r within 'range'.

**NDISp,SElect,NONE**

Deselect all nodal displacements

**NDISp,SElect,NOT**

Invert selection of nodal displacements

**NDISp,SElect,REVerse**

Invert selection of nodal displacements

**NDISp,UNSElect,INDividual,nodenumbe,r,casenumbe,r**

Deselect nodal displacement on node 'nodenumbe,r' for load case 'casenumbe,r'.

**PRESsure,SElect,ALL**

Select all pressure forces

**PRESsure,SElect,ASSociation,NODE**

Select pressure forces by association with currently selected nodes.

**PRESsure,SElect,ASSociation,ELEment**

Select pressure forces by association with currently selected elements.

**PRESsure,SElect,CASe,range**

Select all pressure forces with load case within 'range'.

**PRESsure,SElect,INDividual,elementnumbe,r,face numbe,r,casenumbe,r**

Select pressure force on element 'elementnumbe,r', face 'face numbe,r' for load case 'casenumbe,r'.



**PRESsure,SELEct,ELEment,range**

Select pressure forces with element number within 'range'.

**PRESsure,SELEct,NONe**

Deselect all pressure forces

**PRESsure,SELEct,NOT**

Invert selection of pressure forces

**PRESsure,SELEct,REVerse**

Invert selection of pressure forces

**PRESsure,UNSELEct,INDividual,elementnumber,face,case**

Deselect nodal pressure force on element 'elementnumber', face 'face' for load case 'case'.

**NPRESsure,SELEct,ALL**

Select all nodal pressure forces

**NPRESsure,SELEct,ASSociation,NODE**

Select nodal pressure forces by association with currently selected nodes.

**NPRESsure,SELEct,ASSociation,ELEment**

Select nodal pressure forces by association with currently selected elements.

**NPRESsure,SELEct,CASe,range**

Select all nodal pressure forces with load case within 'range'.

**NPRESsure,SELEct,INDividual,nodenum,case**

Select nodal pressure force on node 'nodenum' for load case 'case'.

**NPRESsure,SELEct,NODE,range**

Select nodal pressure forces with nodenum within 'range'.

**NPRESsure,SELEct,NONe**

Deselect all nodal pressure forces

**NPRESsure,SELEct,NOT**

Invert selection of nodal pressure forces

**NPRESsure,SELEct,REVerse**

Invert selection of nodal pressure forces

**NPRESsure,UNSELEct,INDividual,nodenum,case**

Deselect nodal pressure force on node 'nodenum' for load case 'case'.

**NFORce,SELEct,ALL**

Select all forces

**NFORce,SELEct,ASSociation,NODE**

Select forces by association with currently selected nodes.

**NFORce,SELEct,ASSociation,ELEment**

Select forces by association with currently selected elements.

**NFORce,SELEct,CASe,range**

Select all forces with load case within 'range'.

**NFORce,SELEct,INDividual,nodenum,case**

Select force on node 'nodenum' for load case 'case'.

**NFORce,SELEct,NODE,range**  
Select forces with nodenumber within 'range'.

**NFORce,SELEct,NONE**  
Deselect all forces

**NFORce,SELEct,NOT**  
Invert selection of forces

**NFORce,SELEct,REVERSE**  
Invert selection of forces

**NFORce,UNSELEct,INDividual,nodenumber,casenumber**  
Deselect force on node 'nodenumber' for load case 'casenumber'.

**GROup,DESELEct,groupname**  
Deselect elements in group 'groupname'.

**GROup,SELEct,groupname**  
Select elements in group 'groupname'.

**GROup,UNSELEct,groupname**  
Deselect elements in group 'groupname'.

**LAYER,DESELEct,layername**  
Deselect all entities on layer 'layername'.

**LAYER,SELEct,layername**  
Select all entities on layer 'layername'

**LAYER,UNSELEct,layername**  
Deselect all entities on layer 'layername'.

## 2.2.6 Deletion Commands

The following commands relate to Deleting entities.

**DELEte,SELEcted**

Delete Selected Entities. Where one entity is referenced by another that is not deleted, then it will not be deleted. Eg if a point is to be deleted, but it is used by a line that will remain, the point is not deleted.

**POInt,DELEte,ALL**

Delete all points.

**POInt,DELEte,INACTive**

Delete all points not referenced by other entities.

**POInt,DELEte,SELEcted**

Delete selected points

**POInt,DELEte,range**

Delete range of points.

**LINE,DELEte,ALL**

Delete all lines.

**LINE,DELEte,INACTive**

Delete all lines not used to define surfaces.

**LINE,DELEte,SELEcted**

Delete selected lines

**LINE,DEL,range**

Delete range of lines.

**SURface,DELEte,ALL**

Delete all surfaces.

**SURface,DELEte,INACTive**

Delete all surfaces not referenced by solids.

**SURface,DELEte,SELEcted**

Delete selected surfaces.

**SURface,DELEte,range**

Delete range of surfaces.

**NODE,DELEte,ALL**

Delete all nodes.

**NODE,DELEte,INACTive**

Delete all inactive nodes.

**NODE,DELEte,SELEcted**

Deletes all selected nodes.

**NODE,DELEte,range**

Deletes a range of node numbers.

**ELEment,DELEte,ALL**

Delete all elements

**ELEment,DELEte,SELEcted**

Delete Selected Elements

**ELEment,DELEte,range**  
Delete a range of elements

**ELEment,TEXturemap,DELEte,ALL**  
Delete texture mapping on all elements

**ELEment,TEXturemap,DELEte,FACe,elementnumber,face,face,face**  
Delete the texture mapping on element 'elementnumber', face 'face'

**ELEment,TEXturemap,DELEte,SELEcted**  
Delete texture mapping on the selected elements.

**RIGidlink,DELEte,ALL**  
Delete all Rigid Links.

**RIGidlink,DELEte,SELEcted**  
Delete selected Rigid Links

**RIGidlink,DELEte,range**  
Delete a range of Rigid Links

**MATERial,DELEte,ALL**  
Delete all materials

**MATERial,DELEte,range**  
Delete all materials with material number within range.

**LAMInation,DELEte,ALL**  
Delete all laminates

**LAMInation,DELEte,range**  
Delete all laminates with laminate number within 'range'.

**LAMAngle,DELEte,ALL**  
Delete all lamination angles

**LAMAngle,DELEte,range**  
Delete all lamination angle sets within 'range'.

**PROPerty,DELEte,ALL**  
Delete all properties.

**PROPerty,DELEte,range**  
Delete all properties with property number within 'range'

**LCTitle,DELEte,ALL**  
Delete all load case titles.

**LCTitle,DELEte,range**  
Delete all load case titles with load case number within 'range'.

**LOADcase,DELEte,ALL**  
Delete all load cases

**LOADcase,DELEte,range**  
Delete all load cases with load case number within 'range'.

**NDISp,DELEte,ALL**  
Delete all nodal displacements

**NDISp,DELeTe,SELeCted**

Delete selected nodal displacements

**NDISp,DELeTe,nodenumber,casenumber**

Delete nodal displacement on node 'nodenumber' for load case 'casenumber'.

**PRESsure,DELeTe,ALL**

Delete all pressure forces

**PRESsure,DELeTe,SELeCted**

Delete selected pressure forces

**PRESsure,DELeTe,elementnumber,casenumber,facenumber**

Delete nodal pressure force on element 'elementnumber' for load case 'casenumber'. The pressure deleted is the one applied to face 'facenumber'.

**NPRESsure,DELeTe,ALL**

Delete all nodal pressure forces

**NPRESsure,DELeTe,SELeCted**

Delete selected nodal pressure forces

**NPRESsure,DELeTe,nodenumber,casenumber**

Delete nodal pressure force on node 'nodenumber' for load case 'casenumber'.

**NFORce,DELeTe,ALL**

Delete all forces

**NFORce,DELeTe,SELeCted**

Delete selected forces

**NFORce,DELeTe,nodenumber,casenumber**

Delete force on node 'nodenumber' for load case 'casenumber'.

**BODYForce,DELeTe,ALL**

Delete all accelerations.

**BODYForce,DELeTe,range**

Delete all accelerations with load case number within 'range'

**TEXTURENodes,DELeTe,nodenumber**

Delete texture node 'nodenumber'

## 2.2.7 Points

The following commands relate to points.

### **POInt,ADD**

Starts interactive addition of points

### **POInt,ADD,pointnumber,layernumber,x,y,z**

Adds point 'pointnumber' at x,y,z to layer number 'layernumber'.

### **POInt,COLour**

Opens a dialogue box to specify the colours of points.

### **POInt,COLour,pointcolour,pointselectedcolour**

Sets the colour of points when not selected and when selected.

### **POInt,DELeTe,ALL**

Delete all points.

### **POInt,DELeTe,INACTive**

Delete all points not referenced by other entities.

### **POInt,DELeTe,SELeCted**

Delete selected points

### **POInt,DELeTe,range**

Delete range of points.

### **POInt,ELIMinateduplicates**

Eliminate duplicated points from those points currently selected. Equivalent to 'Merge Points'.

### **POInt,ERAsE,ALL**

Erase all points

### **POInt,ERAsE,ASsociation,LINe**

Erase points by association with currently selected lines.

### **POInt,ERAsE,ASsociation,SURface**

Erase points by association with currently selected surfaces.

### **POInt,ERAsE,NONe**

Plot all points

### **POInt,ERAsE,SELeCted**

Erase selected points.

### **POInt,ERAsE,range**

Erase all points within 'range'.

### **POInt,FINd,range**

Plot all points within 'range' on top of the current screen & temporarily label them. If they are currently invisible, they are made visible.

### **POInt,INSPect**

Start interactive inspection of points.

### **POInt,MESh,ALL**

Mesh all points with point elements according to the current 'Point Element Mesh Settings'.

### **POInt,MESh,SELeCted**

Mesh selected points with point elements according to the current 'Point Element Mesh Settings'.

**POINT,MESH,SETup,elementtype,propertynumber**

Set up the 'Point Element Mesh Settings'. The property number is 'propertynumber', while the element type must be one of the following:-

25	=	2D Concentrated Mass
26	=	3D Concentrated Mass
27	=	2D General Mass
28	=	3D General Mass
29	=	2D General Mass with Inertia
30	=	3D General Mass with Inertia

**POINT,MESH,range**

Mesh all points within 'range' with point elements according to the current 'Point Element Mesh Settings'.

**POINT,MOVE,pointnumber,new x,new y,new z**

Change the co-ordinates of point 'pointnumber' to 'new x,new y,new z'.

**POINT,PLOT,ALL**

Plot all points

**POINT,PLOT,ASsociation,LINe**

Plot points by association with currently selected lines.

**POINT,PLOT,ASsociation,SURface**

Plot points by association with currently selected surfaces.

**POINT,PLOT,NONe**

Erase all points.

**POINT,PLOT,SELected**

Plot selected points. Should be redundant as all selected points should be visible.

**POINT,PLOT,range**

Plot points within 'range'.

**POINT,SELect,ALL**

Select all points.

**POINT,SELect,ASsociation,LINe**

Select points by association with currently selected lines.

**POINT,SELect,ASsociation,SURface**

Select points by association with currently selected surfaces.

**POINT,SELect,NONe**

Deselect all points

**POINT,SELect,NOT**

Invert point selection.

**POINT,SELect,REVerse**

Invert point selection.

**POINT,SELect,VISible**

Select all visible points.

**POINT,SELect,range**

Select all points within 'range'.

**POINT,UNSELect,ALL**

Deselect all points.

**POInt,UNSElect,ASsociation,LINe**

Deselect points by association with currently selected lines.

**POInt,UNSElect,ASsociation,SURface**

Deselect points by association with currently selected surfaces.

**POInt,UNSElect,NONE**

Select all points

**POInt,UNSElect,NOT**

Invert point selection.

**POInt,UNSElect,REVerse**

Invert point selection.

**POInt,UNSElect,range**

Deselect all points within 'range'.

**POInt,VISible,ALL**

Plot all points

**POInt,VISible,ASsociation,LINe**

Plot points by association with currently selected lines.

**POInt,VISible,ASsociation,SURface**

Plot points by association with currently selected surfaces.

**POInt,VISible,NONE**

Erase all points.

**POInt,VISible,SELEcted**

Plot selected points. Should be redundant as all selected points should be visible.

**POInt,VISible,range**

Plot points within 'range'.



## 2.2.8 Lines

The following commands relate to lines.

### **LINE,ADD**

Starts interactive addition of lines

### **LINE,ADD,linenumber,layernumber,startpoint,endpoint**

Add a new line with number 'linenumber' on 'layernumber' between point number 'startpoint' and point number 'endpoint'.

### **LINE,BREak,number of segments**

Breaks the selected lines into 'number of segments' and hides the original line. The original line is not deleted.

### **LINE,CALculate,MIDPOINTS**

Calculates midpoints on all lines. This is a remedial function to help, in the event of a bug, if lines have their mid points calculated in the wrong place.

### **LINE,COLOUR**

Opens a dialogue box to set the colours of lines.

### **LINE,COLOUR,linecolour,linesselectedcolour**

Sets the colour of lines and the colour of lines when they are selected.

### **LINE,DELeTe,ALL**

Delete all lines.

### **LINE,DELeTe,INACTive**

Delete all lines not used to define surfaces.

### **LINE,DELeTe,SELeCted**

Delete selected lines

### **LINE,DEL,range**

Delete range of lines.

### **LINE,ELIMinateduplicates**

Eliminate duplicated lines from those lines currently selected.

### **LINE,ERA,ALL**

Erase all lines.

### **LINE,ERAsE,ASsociation,POInt**

Erase lines associated with currently selected points.

### **LINE,ERAsE,ASsociation,SURface**

Erase lines associated with currently selected surfaces.

### **LINE,ERAsE,NONE**

Plot all lines.

### **LINE,ERAsE,SELeCted**

Erase selected lines.

### **LINE,ERAsE,range**

Erase a range of lines.

### **LINE,FINd,range**

Plot all lines within 'range' on top of the current screen & temporarily label them. If the lines are currently invisible, they are made visible.

**LINE,INSPEct**

Starts the interactive line inspection routine

**LINE,MESH,ALL**

Mesh all lines with line elements according to the current 'Line Element Mesh Settings'.

**LINE,MESH,SELEcted**

Mesh selected lines with line elements according to the current 'Line Element Mesh Settings'

**LINE,MESH,SETup,elementtype,propertynumber,materialnumber,numberofelements**

Specify the current 'Line Element Mesh Settings'. The new property number is 'propertynumber', the new material number is 'materialnumber'. The number of elements along each line is specified by 'numberofelements'.

The element type must be one of the following:-

11	=	3-D Tapered Beam
12	=	3-D Beam
13	=	2-D Beam
14	=	3-D Spar
15	=	2-D Spar
17	=	3-D Translational Spring
18	=	2-D Translational Spring
21	=	3-D Torsional Spring
22	=	2-D Torsional Spring
38	=	3-D General Spring
39	=	3-D General Beam
42	=	2-D or Axisymmetric Gap
43	=	3-D Gap
45	=	3-D Cable element
46	=	3-D Straight Pipe Element
47	=	3-D Elbow Element
48	=	3-D Damper
49	=	2-D Gap/Friction Element
112	=	3-D Bar Element for Heat Transfer
149	=	3-D Convection Link Element
150	=	3-D Radiation Link Element

**LINE,MESH,range**

Mesh lines within 'range' with elements specified by current 'Line Element Mesh Settings'.

**LINE,MOVE,linenumber,delta x,delta y,delta z**

Move Line 'linenumber' by 'delta x,delta y,delta z'. This function actually identifies the points marking the end of the line, and moves them.

**LINE,PLOT,ALL**

Plot all lines

**LINE,PLOT,ASsociation,POInt**

Plot all lines associated with currently selected points.

**LINE,PLOT,ASsociation,SURface**

Plot all lines associated with currently selected surfaces.

**LINE,PLOT,NONE**

Erase all lines

**LINE,PLOT,SELEcted**

Plot all selected lines. Should be redundant as all lines should be plotted as they are selected.

**LINE,PLOT,range**

Plot a range of lines.

**LINE,SElect,ALL**

Select all lines.

**LINE,SElect,ASsociation,POInt**

Select lines by association with currently selected points.

**LINE,SElect,ASsociation,SURface**

Select lines by association with currently selected surfaces.

**LINE,SElect,NONe**

Deselect all lines.

**LINE,SElect,NOT**

Invert line selection.

**LINE,SElect,REVerse**

Invert line selection.

**LINE,SElect,VISible**

Select all visible lines.

**LINE,SElect,range**

Select lines that fall within 'range'.

**LINE,UNSElect,ALL**

Deselect all lines

**LINE,UNSElect,ASsociation,POInt**

Deselect lines by association with currently selected points.

**LINE,UNSElect,ASsociation,SURface**

Deselect lines by association with currently selected surfaces.

**LINE,UNSElect,NONe**

Select all lines.

**LINE,UNSElect,NOT**

Invert line selection.

**LINE,UNSElect,REVerse**

Invert line selection.

**LINE,UNSElect,range**

Deselect a range of lines.

**LINE,VISible,ALL**

Plot all lines

**LINE,VISible,ASsociation,POInt**

Plot all lines associated with currently selected points.

**LINE,VISible,ASsociation,SURface**

Plot all lines associated with currently selected surfaces.

**LINE,VISible,NONe**

Erase all lines

**LINE,VISible,SElected**

Plot all selected lines. Should be redundant as all lines should be plotted as they are selected.

**LINE,ViSible**,range  
Plot a range of lines.

## 2.2.9 Surfaces

The following commands relate to surfaces.

**SURface,ADD,2Line,surfacenumber,layernumber,LIne,linenumber1,line1reversed,LIne,linenumber2,line2reversed**

Adds a surface with 2 Straight Lines. The new surface is 'surfacenumber', and is added to layer 'layernumber'. The first line is 'linenumber1', and the line is reversed if line1reversed = 1, and not reversed if line1reversed = 0. The second line defining the surface is 'linenumber2'. Line 2 is reversed if line2reversed = 1, and not reversed if line2reversed = 0. If both lines are parallel and start at the ends nearest to each other, then line2reversed should be '1' otherwise an hourglass shaped surface will be created.

It is not envisaged that this command will need to be used manually.

**SURface, AUTomesh, ALL, target edge length**

Automesh all surfaces, with the elements of type specified by the current surface mesh settings. The number of elements along each edge of a surface is the closest interger multiple of the target edge length on that surface edge.

**SURface, AUTomesh, SElected, target edge length**

Automesh all selected surfaces, with the elements of type specified by the current surface mesh settings. The number of elements along each edge of a surface is the closest interger multiple of the target edge length on that surface edge.

**SURface, AUTomesh, SET, target edge length**

Specify the target edge length for automeshing.

**SURface, AUTomesh, SET**

Open an interactive window to specify the automesh parameters.

**SURface,CALculate,NORMalcentroid**

Recalculates the surface normal and centroid in the event of a bug causing the information to be out of date.

**SURface,COLour**

Opens a dialogue box to set the colours of surfaces.

**SURface,COLour,surfacecolour,surfaceoutlinecolour**

Sets the colours of surfaces and selected surfaces.

**SURface,DELeTe,ALL**

Delete all surfaces.

**SURface,DELeTe,INACTive**

Delete all surfaces not referenced by solids.

**SURface,DELeTe,SElected**

Delete selected surfaces.

**SURface,DELeTe,range**

Delete range of surfaces.

**SURface,ERAsE,ALL**

Erase all surfaces.

**SURface,ERAsE,ASsociation,LINe**

Erase surfaces associated with currently selected lines.

**SURface,ERAsE,ASsociation,POInt**

Erase surfaces associated with currently selected points.

**SURface,ERAsE,NONe**

Plot all surfaces.

**SURface,ERASE,SELected**  
Erase selected surfaces.

**SURface,ERASE,range**  
Erase all surfaces within 'range'.

**SURface,FILLtype,DEPTH**  
Change the surface fill colour to depth shaded.

**SURface,FILLtype,SHaded**  
Change the surface fill colour to infinite light shaded.

**SURface,FILLtype,ID**  
Set the surface fill colour to be by ID number.

**SURface,FINd,range**  
Plot all surfaces within 'range' on top of the current screen & temporarily label them. If the surfaces are currently invisible, they are made visible.

**SURface,INSPEct**  
Starts the interactive surface inspection routine

**SURface,MESH,ALL**  
Mesh all surfaces with shell elements according to the current 'Shell Element Mesh Settings'.

**SURface,MESH,EDGE,edge1,edge2,edge3,edge4**  
Specify numbers of elements to be meshed along each surface edge. If surface is triangular, edge4 is ignored.

**SURface,MESH,SELected**  
Mesh selected surfaces with shell elements according to the current 'Shell Element Mesh Settings'.

**SURface,MESH,SETup,elementtype,propertynumber,materialnumber,elementorder,Unumber,Vnumber,erase\_**  
**surface\_after\_meshing**

Specify the surface mesh settings. The element type must be one of the following:-

20	=	3D General Shell
32	=	3D Laminated Composite Shell
33	=	3D Sandwich Shell
120	=	3D Thin Shell for Heat Transfer

The material number and property number are specified by 'materialnumber' and 'propertynumber'. The element order is as below:-

1	=	Linear
2	=	Parabolic

The 'Unumber' indicates the number of elements in the surface 'U' direction (along the first line in the surface) and the 'Vnumber' indicates the number of elements in the direction perpendicular to the first line in the surface.

If 'erase\_surface\_after\_meshing' = 1, then the surface that was meshed is erased, while if 'erase\_surface\_after\_meshing' = 0, it is not.

**SURface,MESH,range**  
Mesh surfaces within 'range' with shell elements according to the current 'Shell Element Mesh Settings'.

**SURface,PLOT,ALL**  
Plot all surfaces

**SURface,PLOT,ASsociation,LINE**  
Plot surfaces by association with currently selected lines.

**SURface,PLOT,ASsociation,POINT**  
Plot surfaces by association with currently selected points.

**SURface,PLOt,NONe**

Erase all surfaces.

**SURface,PLOt,SELEcted**

Plot selected surfaces. This command should be redundant.

**SURface,PLOt,range**

Plot surfaces within 'range'.

**SURface,REVERSENormals,ALL**

Reverse the normals of all surfaces.

**SURface,REVERSENormals,SELEcted**

Reverse the normals of the selected surfaces.

**SURface,REVERSENormals,range**

Reverse the normals of surfaces within 'range'.

**SURface,SELEct,ALL**

Select all surfaces.

**SURface,SELEct,ASsociation,LINe**

Select surfaces by association with currently selected lines.

**SURface,SELEct,ASsociation,POINt**

Select surfaces by association with currently selected points.

**SURface,SELEct,NONe**

Deselect all surfaces.

**SURface,SELEct,NOT**

Invert surface selection.

**SURface,SELEct,REVERSE**

Invert surface selection.

**SURface,SELEct,VISible**

Select all visible surfaces.

**SURface,SELEct,range**

Select surfaces within 'range'.

**SURface,UNSELEct,ALL**

Deselect all surfaces.

**SURface,UNSELEct,ASsociation,LINe**

Deselect surfaces by association with currently selected lines.

**SURface,UNSELEct,ASsociation,POINt**

Deselect surfaces by association with currently selected points.

**SURface,UNSELEct,NONe**

Select all surfaces.

**SURface,UNSELEct,NOT**

Invert surface selection.

**SURface,UNSELEct,REVERSE**

Invert surface selection.

**SURface,UNSElect,range**  
Deselect surfaces within 'range'.

**SURface,VISible,ALL**  
Plot all surfaces

**SURface,VISible,ASsociation,LINE**  
Plot surfaces by association with currently selected lines.

**SURface,VISible,ASsociation,POInt**  
Plot surfaces by association with currently selected points.

**SURface,VISible,NONE**  
Erase all surfaces.

**SURface,VISible,SElected**  
Plot selected surfaces. This command should be redundant.

**SURface,VISible,range**  
Plot surfaces within 'range'.



## 2.2.10 Meshing Commands

The following commands relate to meshing geometry to form nodes and elements.

### **POInt,MESh,ALL**

Mesh all points with point elements according to the current 'Point Element Mesh Settings'.

### **POInt,MESh,SELected**

Mesh selected points with point elements according to the current 'Point Element Mesh Settings'.

### **POInt,MESh,SETup,elementtype,propertynumber**

Set up the 'Point Element Mesh Settings'. The property number is 'propertynumber', while the element type must be one of the following:-

25	=	2D Concentrated Mass
26	=	3D Concentrated Mass
27	=	2D General Mass
28	=	3D General Mass
29	=	2D General Mass with Inertia
30	=	3D General Mass with Inertia

### **POInt,MESh,range**

Mesh all points within 'range' with point elements according to the current 'Point Element Mesh Settings'.

### **LINE,MESh,ALL**

Mesh all lines with line elements according to the current 'Line Element Mesh Settings'.

### **LINE,MESh,SELected**

Mesh selected lines with line elements according to the current 'Line Element Mesh Settings'.

### **LINE,MESh,SETup,elementtype,propertynumber,materialnumber,numberofelements**

Specify the current 'Line Element Mesh Settings'. The new property number is 'propertynumber', the new material number is 'materialnumber'. The number of elements along each line is specified by 'numberofelements'.

The element type must be one of the following:-

11	=	3-D Tapered Beam
12	=	3-D Beam
13	=	2-D Beam
14	=	3-D Spar
15	=	2-D Spar
17	=	3-D Translational Spring
18	=	2-D Translational Spring
21	=	3-D Torsional Spring
22	=	2-D Torsional Spring
38	=	3-D General Spring
39	=	3-D General Beam
42	=	2-D or Axisymmetric Gap
43	=	3-D Gap
45	=	3-D Cable element
46	=	3-D Straight Pipe Element
47	=	3-D Elbow Element
48	=	3-D Damper
49	=	2-D Gap/Friction Element
112	=	3-D Bar Element for Heat Transfer
149	=	3-D Convection Link Element
150	=	3-D Radiation Link Element

### **LINE,MESh,range**

Mesh lines within 'range' with elements specified by current 'Line Element Mesh Settings'.

**SURface, AUTomesh, ALL**, target edge length

Automesh all surfaces, with the elements of type specified by the current surface mesh settings. The number of elements along each edge of a surface is the closest interger multiple of the target edge length on that surface edge.

**SURface, AUTomesh, SElected**, target edge length

Automesh all selected surfaces, with the elements of type specified by the current surface mesh settings. The number of elements along each edge of a surface is the closest interger multiple of the target edge length on that surface edge..

**SURface, AUTomesh, SET**, target edge length

Specify the target edge length for automeshing.

**SURface, AUTomesh, SET**

Open an interactive window to specify the automesh parameters.

**SURface, MESH, ALL**

Mesh all surfaces with shell elements according to the current 'Shell Element Mesh Settings'.

**SURface, MESH, EDGE**, edge1, edge2, edge3, edge4

Specify numbers of elements to be meshed along each surface edge. If surface is triangular, edge4 is ignored.

**SURface, MESH, SElected**

Mesh selected surfaces with shell elements according to the current 'Shell Element Mesh Settings'.

**SURface, MESH, SET** up, elementtype, propertynumber, materialnumber, elementorder, Unumber, Vnumber, erase\_surface\_after\_meshing

Specify the surface mesh settings. The element type must be one of the following:-

20	=	3D General Shell
32	=	3D Laminated Composite Shell
33	=	3D Sandwich Shell
120	=	3D Thin Shell for Heat Transfer

The material number and property number are specified by 'materialnumber' and 'propertynumber'. The element order is as below:-

1	=	Linear
2	=	Parabolic

The 'Unumber' indicates the number of elements in the surface 'U' direction (along the first line in the surface) and the 'Vnumber' indicates the number of elements in the direction perpendicular to the first line in the surface.

If 'erase\_surface\_after\_meshing' = 1, then the surface that was meshed is erased, while if 'erase\_surface\_after\_meshing' = 0, it is not.

**SURface, MESH, range**

Mesh surfaces within 'range' with shell elements according to the current 'Shell Element Mesh Settings'.

### 2.2.11 Executive Commands

The following commands relate to the executive commands that control the FE analysis.

**EXECutiveCommands,ANALysistype,type**

Specifies the analysis type. 'type' may be either '**Static**' or '**Eigen**'. At the time of writing of the Command Reference, Static Analysis was the only fully implemented analysis type. For Eigen analysis some data cards will need to be added manually to the FE analysis file.

**EXECutiveCommands,AUTOconstrain,state**

NISA specific command. Specifies whether a NISA model is to have shells automatically constrained against spurious rotation about their normal. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,BLANKcommon,number**

NISA specific command. For a NISA analysis the size of the Blank Common scratch area needs to be defined. This must be at least 7x the number of nodes – but experience shows that this is a minimum figure that sometimes isn't enough. When the NISA file is exported (using '**EXPORT,NISA,filename**') the minimum value of Blank Common is calculated and rounded up to the next highest 50,000. The number that is exported is the higher of the figure entered here and the figure calculated.

**EXECutiveCommands,COMPutegeometricproperties,state**

NISA specific command. Specifies whether NISA is to compute the model geometric properties. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,ECHO,state**

Specifies whether the geometry is to be echoed in the output file. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,EIGEntype,type**

NISA specific command. Specifies the Eigen analysis type. Type must be one of the following:-

**AC**celerated  
**CO**nventional  
**IN**verse  
**LA**nczos  
**GU**yan

The default is none of the above.

**EXECutiveCommands,EXECutiontype,type**

NISA specific command. Specifies the way the analysis proceeds. Type must be one of the following:-

**EX**ecuteonly  
**CHECKO**nly  
**CHECKA**ndexecute

The default is **CHECKA**ndexecute

**EXECutiveCommands,FILEroot,name**

NISA specific command. Specify the starting section of the filename of the binary analysis files saved by NISA.

Eg     **EXECutiveCommands,FILEroot,Block**

Indicates that NISA will save the 26 and 27 files as 'Block26.dat' and 'Block27.dat'.

**EXECutiveCommands,INCLudekinematicconstraints,state**

NISA specific command. Specifies whether Kinematic constraints are to be included. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,MASSformulation,type**

NISA specific command. Specifies the way masses are considered. Type must be one of the following:-

**CO**nsistent  
**LU**mped

The default is **CO**nsistent.

**EXECutiveCommands,RESEQUENCEOptions,option**

NISA specific command. Element resequencing is carried out. Option must be one of the following:-

**ON**  
**OFF**  
**GEom**  
**BOth**

The default is **ON**.

**EXECutiveCommands,RESEQUENCEMax,state**

NISA specific command. Specifies whether the maximum is to be listed in element resequencing. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,RESEQUENCEList,state**

NISA specific command. Specifies whether the internal cross reference list during element resequencing is to be listed in the output file. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,TARGETprogram,name**

Specifies the model target program. Name must be one of the following:-

**MYstran**  
**NIsa**

The default is **NIsa**

**EXECutiveCommands,SAVEGeometarybinary,state**

NISA specific command. Specifies whether the '26.dat' file is to be saved. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,SAVEOutputbinary,state**

NISA specific command. Specifies whether the '27.dat' file is to be saved. 'state' is either '**ON**' or '**OFF**'. The default value of 'state' is '**ON**'.

**EXECutiveCommands,SORT,number**

NISA specific command. Specifies whether how many top stresses are to be sorted. No default.

**EXECutiveCommands,WARNingaction,action**

NISA specific command. Specifies the FE analysis behaviour when a warning occurs. Action must be one of the following:-

**GO**  
**STop**

The default is **GO**

### 2.2.12 Nodes

The following commands relate to Nodes.

#### **NODE,ADD**

Add a single node by typing x,y,z co-ordinates into a dialogue box.

#### **NODE,ADD,nodenum,ics,x,y,z[,selected,visible,colour,layernumber]**

Add node 'nodenum' which has local co-ordinate system defined by ics. The global Cartesian co-ordinates of the node are x,y,z. If defined, the node is selected if selected = 1, it is visible if visible = 1, the node colour is 'colour' a 24 bit RGB colour representation. The layer number of the node is layernumber.

#### **NODE,ALIGN,direction X or Y or Z,value**

Moves the X Y or Z value of the selected node co-ordinates to 'value'.

#### **NODE,COLOUR**

Set node colour by opening dialogue box.

#### **NODE,COLOUR,ALL,colour**

Change the colour of all nodes to 'colour'.

#### **NODE,COMPactids**

Open dialogue box to compact nodal ID numbers.

#### **NODE,COMPactids,start,increment**

Compact the ID numbers of selected nodes starting at 'start', with increment 'increment'. No dialogue box is opened.

#### **NODE,DELETE,ALL**

Delete all nodes.

#### **NODE,DELETE,INACTIVE**

Delete all inactive nodes.

#### **NODE,DELETE,SELECTED**

Deletes all selected nodes.

#### **NODE,DELETE,range**

Deletes a range of node numbers.

#### **NODE,ERASE,ALL**

Erase all nodes.

#### **NODE,ERASE,ASSOCIATION,ELEMENT**

Erase all nodes associated with the currently selected Elements.

#### **NODE,ERASE,ASSOCIATION,RIGIDLINK**

Erase all nodes associated with the currently selected Rigid Links

#### **NODE,ERASE,ASSOCIATION,NDISPLACEMENT**

Erase all nodes associated with the currently selected Displacement Constraints

#### **NODE,ERASE,ASSOCIATION,PRESSURE**

Erase all nodes associated with the currently selected Pressures

#### **NODE,ERASE,ASSOCIATION,NPRESSURE**

Erase all nodes associated with the currently selected Nodal Pressures

#### **NODE,ERASE,ASSOCIATION,NFORCE**

Erase all nodes associated with the currently selected Forces

**NODE,ERASE,NONE**

Plot all nodes.

**NODE,ERASE,SELEcted**

Erase selected nodes.

**NODE,ERASE,range**

Erases a range of node numbers.

**NODE,FINd,range**

Finds a range of node numbers and displays them on the screen with their labels temporarily switched on.

**NODE,INACTive**

Delete all inactive nodes, equivalent to '**NODE,DELEte,INACTive**'

**NODE,INSPect**

Opens a dialogue window listing locations of all nodes.

**NODE,MERge[,tolerance]**

Merge selected nodes. If defined, 'tolerance' is used as the maximum merge distance between nodes.

**NODE,MOVE,nodenumbeR,new x,new y,new z**

Move node 'nodenumbeR' to Cartesian co-ordinate 'new x,new y,new z'

**NODE,PLOt,ALL**

Plot all nodes.

**NODE,PLOt,ASsociation,ELEment**

Plot all nodes associated with the currently selected elements

**NODE,PLOt,ASsociation,RIGidLink**

Plot all nodes associated with the currently selected Rigid Links

**NODE,PLOt,ASsociation,NDISPlacement**

Plot all nodes associated with the currently selected Displacement Constraints

**NODE,PLOt,ASsociation,PRESSure**

Plot all nodes associated with the currently selected Pressures

**NODE,PLOt,ASsociation,NPRESSure**

Plot all nodes associated with the currently selected Nodal Pressures

**NODE,PLOt,ASsociation,NFORce**

Plot all nodes associated with the currently selected Forces

**NODE,PLOt,NONE**

Erase all nodes

**NODE,PLOt,SELEcted**

Plots all selected nodes. This should be redundant as all nodes should be made visible as they are selected.

**NODE,PLOt,range**

Plots a range of nodes.

**NODE,RENumber,old nodenumbeR,new nodenumbeR**

Renumber node with 'old nodenumbeR' as 'new nodenumbeR'

**NODE,SELEct,ALL**

Select all nodes. If nodes are invisible prior to being selected, they are made visible.

**NODE,SELEct,ASsociation,ELEment**

Select all nodes associated with the currently selected Elements. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElect,ASsociation,RIGidLink**

Select all nodes associated with the currently selected Rigid Links. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElect,ASsociation,NDISPlacement**

Select all nodes associated with the currently selected Displacement Constraints

**NODE,SElect,ASsociation,PRESSure**

Select all nodes associated with the currently selected Pressures

**NODE,SElect,ASsociation,NPRESSure**

Select all nodes associated with the currently selected Nodal Pressures

**NODE,SElect,ASsociation,NFORce**

Select all nodes associated with the currently selected Forces

**NODE,SElected,FALse,nodenumber**

Deselect node 'nodenumber'

**NODE,SElect,NONe**

Deselect all nodes.

**NODE,SElect,NOT**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODE,SElect,REVerse'

**NODE,SElect,REVerse**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODE,SElect,NOT'

**NODE,SElect,range**

Selects a range of node numbers. If nodes are invisible prior to being selected, they are made visible.

**NODE,SElected,TRUe,nodenumber**

Select node 'nodenumber'. If node is invisible prior to being selected, it is made visible.

**NODE,SElect,VISible**

Select all visible nodes.

**NODE,SElect,XRANGE, max value, min value**

Select all nodes with an X co-ordinate between Max value and Min value.

**NODE,SElect,YRANGE, max value, min value**

Select all nodes with an Y co-ordinate between Max value and Min value.

**NODE,SElect,ZRANGE, max value, min value**

Select all nodes with an Z co-ordinate between Max value and Min value.

**NODE,UNSElect,ALL**

Deselect all nodes.

**NODE,UNSElect,ASsociation,ELEment**

Deselect all nodes associated with the currently selected Elements. If nodes are invisible prior to being selected, they are made visible.

**NODE,UNSElect,ASsociation,RIGidLink**

Deselect all nodes associated with the currently selected Rigid Links. If nodes are invisible prior to being selected, they are made visible.

**NODE,UNSElect,ASsociation,NDISPlacement**

Deselect all nodes associated with the currently selected Displacement Constraints

**NODE,UNSElect,ASsociation,PRESSure**

Deselect all nodes associated with the currently selected Pressures

**NODE,UNSElect,ASsociation,NPRESSure**

Deselect all nodes associated with the currently selected Nodal Pressures

**NODE,UNSElect,ASsociation,NFORce**

Deselect all nodes associated with the currently selected Forces

**NODE,UNSElect,NONE**

Select all nodes.

**NODE,UNSElect,NOT**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODE,SElect,REVerse'

**NODE,UNSElect,REVerse**

Invert selection of all nodes. If nodes are invisible prior to being selected, they are made visible. Equivalent to 'NODE,SElect,REVerse'

**NODE,UNSElect,range**

Deselects a range of node numbers.

**NODE,VISible,ALL**

Plot all nodes.

**NODE,VISible,ASsociation,ELEment**

Plot all nodes associated with the currently selected elements

**NODE,VISible,ASsociation,RIGidLink**

Plot all nodes associated with the currently selected Rigid Links

**NODE,VISible,ASsociation,NDISPlacement**

Plot all nodes associated with the currently selected Displacement Constraints

**NODE,VISible,ASsociation,PRESSure**

Plot all nodes associated with the currently selected Pressures

**NODE,VISible,ASsociation,NPRESSure**

Plot all nodes associated with the currently selected Nodal Pressures

**NODE,VISible,ASsociation,NFORce**

Plot all nodes associated with the currently selected Forces

**NODE,VISible,NONE**

Erase all nodes

**NODE,VISible,SElected**

Plots all selected nodes. This should be redundant as all nodes should be made visible as they are selected.

**NODE,VISible,range**

Plots a range of nodes.



### 2.2.13 Elements

The following commands are element related.

**ELEment,ADD,elementnumber,materialnumber,propertynumber,elementtype,nodeorder,numberofnodes,node list,[layernumber,groupnumber,]**

Adds an element. The new element has number 'elementnumber', is allocated material 'materialnumber' and property 'propertynumber'. The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

The Node Order is the order of the element, and is defined as below:-

Node Order	Description
1	For all point or line elements
1	For all 2 noded axisymmetric shell elements

1	For all 4 noded shell elements/ 4 noded axisymmetric solid elements
1 or 12	For all 8 noded solid elements
2	For all 3 noded axisymmetric elements
2	For all 8 noded shell elements
2	For all 20 noded brick elements
10	For 3 Noded triangular shell elements
10	6 noded prism shaped brick elements
11	6 Noded triangular shell elements
11	15 noded prism brick elements
20	4 noded triangular pyramid solid elements
21	10 noded triangular pyramid solid elements
30	5 noded rectangular pyramid elements
31	13 noded rectangular pyramid elements

The layernumber and group numbers are optional.

**ELEment,CALculate,CENtroid**

Recalculates the element face centroid and centroid in the event of a bug causing the information to be out of date.

**ELEment,CALculate,NORmal**

Recalculates the element normal, in the event of a bug causing the information to be out of date.

**ELEment,CALculate,NORMALCentroid**

Recalculates the element normal, face centroid and centroid in the event of a bug causing the information to be out of date.

**ELEment,CENtroid,RECalculate**

Recalculates the element centroids. Intended to correct an error in element copy/move operations if the elements cannot be subsequently selected. In a program with no bugs, this will be redundant.

**ELEment,CHANGEMaterial,elementnumber,newnumber**

Change the material of element 'elementnumber' to 'newnumber'

**ELEment,CHANGEProperty,elementnumber,newnumber**

Change the property of element 'elementnumber' to 'newnumber'

**ELEment,COLour**

Set the colour of the selected elements

**ELEment,COLOURLIST**

Opens a window to set the sixteen colours used to differentiate between materials, groups, layers etc.

**ELEment,COLOURLIST,listnumber,colour**

Sets colour list 'listnumber' to 'colour'.

**ELEment,COLour,SELected**

Set the colour of the selected elements

**ELEment,COLour,elementnumber,colour**

Set the colour of element number 'elementnumber' to 'colour'

**ELEment,COMPactids**

Open dialogue box to compact element ID numbers.

**ELEment,DELeTe,ALL**

Delete all elements

**ELEment,DELeTe,SELected**

Delete Selected Elements

**ELEment,DELEte,range**  
Delete a range of elements

**ELEment,EDGEoutlines,state**  
Determine whether to plot element edge outlines. State is either ON or OFF.

**ELEment,ELIminateCommonFaces**  
Calculates internal element faces in solid models. Turns the plotting of such faces off to speed up plotting.

**ELEment,ERASE,ALL**  
Erase all elements

**ELEment,ERASE,ASsociation,NODE**  
Erase elements associated with selected nodes.

**ELEment,ERASE,ASsociation,NDISPlacements**  
Erase Elements by association with selected Nodal Displacements

**ELEment,ERASE,ASsociation,NFORce**  
Erase Elements by association with selected Forces

**ELEment,ERASE,ASsociation,NPRESSure**  
Erase t Elements by association with selected Nodal Pressures

**ELEment,ERASE,ASsociation,PRESSure**  
Erase Elements by association with selected Pressures

**ELEment,ERASE,ASsociation,RIGidLink**  
Erase elements by association with selected Rigid Links

**ELEment,ERASE,FORMulation,range**  
Erase the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment,ERASE,NONE**  
Plot all elements

**ELEment,ERASE,SELected**  
Erase selected elements

**ELEment,ERASE,MATerial,range**  
Erase all elements with material number specified in 'range'

**ELEment,ERASE,NORDR,range**  
Erase all elements with Node Order number specified in 'range'

**ELEment,ERASE,NSRL,range**  
Erase all elements with NSRL number specified in 'range'

**ELEment,ERASE,PROPerty,range**  
Erase all elements with property number specified in 'range'

**ELEment,ERASE,TYPE,range**  
Erase all elements with element type number specified in 'range'  
The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element

2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,ERASE,range**  
Erase all elements within 'range'

**ELEment,FILLcolour**  
Set the element fill colour to be by face number. Seemed a good idea at the time.

**ELEment,INSPECT**  
Starts the interactive element inspection routine

**ELEment,NODEReplace,elementnumber,oldnodenumber,newnodenumber**  
In element 'elementnumber', replace 'oldnodenumber' with 'newnodenumber'.

**ELEment,ORDER,LINeartoparabolic**  
Changes all selected linear elements to parabolic formulation

**ELEment,PLOt,ALL**

Plot all elements

**ELEment,PLOt,ASsociation,NODE**

Plot elements by association with selected nodes

**ELEment,PLOt,ASsociation,NDISPlacements**

Plot Elements by association with selected Nodal Displacements

**ELEment,PLOt,ASsociation,NFORce**

Plot Elements by association with selected Forces

**ELEment,PLOt,ASsociation,NPRESSure**

Plot Elements by association with selected Nodal Pressures

**ELEment,PLOt,ASsociation,PRESSure**

Plot Elements by association with selected Pressures

**ELEment,PLOt,ASsociation,RIGidLink**

Plot Elements by association with selected Rigid Links

**ELEment,PLOt,FORMulation,range**

Plot the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment,PLOt,MATerial,range**

Plot the elements by material range.

**ELEment,PLOt,NKTP,range**

Plot the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia

30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,PLOt,NORDR,range**

Plot the elements by Node Order range.

**ELEment,PLOt,NSRL,range**

Plot the elements by NSRL range.

**ELEment,PLOt,PROPerty,range**

Plot the elements by property range.

**ELEment,PLOt,TYPE,range**

Plot the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia

30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,PLOt,NONE**

Erase all elements

**ELEment,PLOt,SELEcted**

Plot selected elements

**ELEment,PLOt,range**

Plot element range.

**ELEment,ORDer,LINeartoparabolic**

Changes all selected linear elements to parabolic formulation

**ELEment,ORDer,PARabolictolinear**

Changes all selected parabolic elements to linear formulation

**ELEment,REPair,NORDR**

Run through all the elements and, based on the numbers of nodes in the element, sets the Node Order. This can be useful after a difficult to repair bug occurs.

**ELEment,REPair,NSRL**

Recalculates all the element NSRL numbers based on the element Node Order and Type. This can be useful after a difficult to repair bug occurs.

**ELEment,REVersenormal,SELEcted**

Reverse the normals on selected shell elements

**ELEment,REVersenormal,range**

Reverse normals on elements within 'range'.

**ELEment,SELEct,ALLNODESselected**

Select all elements where all their nodes are already selected. Any selected elements initially invisible are plotted.

**ELEment,SELEct,ALL**

Select all elements. Any selected elements initially invisible are plotted.

**ELEment,SELEct,ASsociation,NODE**

Select elements associated with selected nodes. Any selected elements initially invisible are plotted.

**ELEment,SELEct,ASsociation,NDISPlacements**

Select Elements by association with selected Nodal Displacements

**ELEment,SELEct,ASsociation,NFORce**

Select Elements by association with selected Forces

**ELEment,SELEct,ASsociation,NPRESSure**

Select Elements by association with selected Nodal Pressures

**ELEment,SELEct,ASsociation,PRESSure**

Select Elements by association with selected Pressures

**ELEment, SELEct, ASsociation,RIGidLink**

Select elements associated with selected Rigid Links. Any selected elements initially invisible are plotted.

**ELEment, SELEcted, CHANGEMaterial,newnumber**

Set the material number in all selected elements to 'newnumber'.

**ELEment, SELEcted, CHANGEProperty,newnumber**

Set the property number in all selected elements to 'newnumber'.

**ELEment, SELEcted, DETriangulate**

In the selected elements, detriangulate as many elements as possible. The function checks to see if any two triangular elements have a common long edge, and merge that edge to form a quadrilateral element. Only works with Element Type 20, Node Order 10 elements, turning them into Node Order 1 elements. Note: This function deletes the triangles. Texture mapping is preserved, but node normal directions are lost. If the operation is undone, the undeleted elements lose any texture mapping and node normal directions.

**ELEment, SELEct, DUPLIcates**

Select all duplicated elements – whether forward or reverse nodal definitions.

**ELEment,SELEct,FORMulation,range**

Select the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

Any selected elements initially invisible are plotted.

**ELEment,SELEct,MATERial,range**

Select all elements with material in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SELEct,NKTP,range**

Select all elements with element type in 'range'. Any selected elements initially invisible are plotted.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element



14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,SElect,NONe**  
Deselect all elements

**ELEment,SElect,NORDR,range**  
Select all elements with Node Order in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElection,NOT**  
Invert the selection of the element. Any selected elements initially invisible are plotted.

**ELEment,SElect,NSRL,range**  
Select all elements with NSRL in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElected,OFFSet, value**  
Copy the currently selected NKTP 20 Shell elements a distance of 'value' in the same direction as the shell normal.

**ELEment,SElect,PROPerty,range**  
Select all elements with property in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SElection,REVerse**  
Invert the selection of the element. Any selected elements initially invisible are plotted.

**ELEment,SElection,SHApe, range**  
Invert the selection of the element. Any selected elements initially invisible are plotted.

The element shape is as below:-

Shape	Description
1	Point Element
2	Line Element
3	Axisymmetric Shell Element
4	Axisymmetric Solid Element
5	Heat Transfer Element
6	Planar Element
7	Shell Element
8	Solid Element

**ELEment,SELEcted,THICKen**, value

Copy the currently selected NKTP 20 Shell elements a distance of 'value' in the opposite direction to the shell normal. Then the perimeter of the selected group of elements is sealed by the addition of shell elements around the open edge of the selected group. This is primarily used for OBJ files making them thick enough to be used in 3D printers.

**ELEment,SELEct,TYPE**,range

Select all elements with element type in 'range'. Any selected elements initially invisible are plotted.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction

50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,SELEct,VISible**  
Select all visible elements.

**ELEment,SELEct,range**  
Select all elements in 'range'. Any selected elements initially invisible are plotted.

**ELEment,SEParate,SELEcted**  
Separate the selected elements from the remainder of the model.

**ELEment,SHIftright,range**  
Rotate origin of all elements in 'range' by one corner anti-clockwise about the element normal.

**ELEment,SHIftright,SELEcted**  
Rotate origin of all elements in 'range' by one corner anti-clockwise about the element normal.

**ELEment,SLIce,Point1.x,Point1.y,Point1.z,Point2.x, Point2.y, Point2.z, Point3.x, Point3.y, Point3.z**  
This command slices all selected linear shell elements along the plane defined by points 1, 2 and 3.

**ELEment,TEXturement,ADD,ALL,left,bottom,right,top,stretch map to fit area**  
Texture map all elements into area defined by the rectangle left,bottom,right,top. All values defining the rectangle must be between 0.0 and 1.0. If the map is to be stretched over the area defined 'stretch map to fit area' = 1, to retain the model proportions, it needs to be 0; eg

```
ELE,TEXturement,ADD,ALL,0.0,0.0,1.0,1.0,1
```

**ELEment,TEXturement,ADD,SELEcted,left,bottom,right,top,stretch map to fit area**  
Texture map selected elements into area defined by the rectangle left,bottom,right,top. All values defining the rectangle must be between 0.0 and 1.0. If the map is to be stretched over the area defined 'stretch map to fit area' = 1, to retain the model proportions, it needs to be 0.

**ELEment,TEXturement,BOX,ADD,ALL,left,bottom,right,top**  
Texture map all elements into area defined by the rectangle left,bottom,right,top, using a box texture mapping method assuming +Y = up. All values defining the rectangle area of the map the elements are mapped onto must be between 0.0 and 1.0.; eg

```
ELE,TEXturement,BOX,ADD,ALL,0.0,0.0,1.0,1.0
```

**ELEment,TEXturement,BOX,ADD,SELEcted,left,bottom,right,top**  
Texture map the selected elements into area defined by the rectangle left,bottom,right,top, using a box texture mapping method assuming +Y = up. All values defining the rectangle area of the map the elements are mapped onto must be between 0.0 and 1.0.

**ELEment,TEXturement,DELEte,ALL**  
Delete texture mapping on all elements

**ELEment,TEXturement,DELEte,FACe,elementnumber,face number,**  
Delete the texture mapping on element 'elementnumber', face 'face number'

**ELEment,TEXturement,DELEte,SELEcted**  
Delete texture mapping on the selected elements.

**ELEment,TEXturement,FACe,elementnumber,face number,texturenodenumberlist...**

Apply a texture map to element number 'elementnumber' and face 'facenumber'. The texture node numbers are specified in texturenodenumberslist .

**ELEment,UNSELEct,ALL**

Deselect all elements

**ELEment,UNSELEct,ASsociation,NODE**

Deselect elements associated with selected nodes. Any selected elements initially invisible are plotted.

**ELEment,UNSELEct,ASsociation,NDISPlacements**

Deselect Elements by association with selected Nodal Displacements

**ELEment,UNSELEct,ASsociation,NFORce**

Deselect Elements by association with selected Forces

**ELEment,UNSELEct,ASsociation,NPRESSure**

Deselect Elements by association with selected Nodal Pressures

**ELEment,UNSELEct,ASsociation,PRESSure**

Deselect Elements by association with selected Pressures

**ELEment,UNSELEct,ASsociation,RIGidLink**

Deselect elements associated with selected Rigid Links. Any selected elements initially invisible are plotted.

**ELEment,UNSELEct,FORMulation,range**

Deselect the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment,UNSELEct,MATerial,range**

Deselect the elements by material range.

**ELEment,UNSELEct,NKTP,range**

Deselect the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass

29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,UNSElect,NONE**

Select all elements.

**ELEment,UNSElect,NOT**

Invert element selection. Any selected elements initially invisible are plotted.

**ELEment,UNSElect,NORDR,range**

Deselect the elements by Node Order range.

**ELEment,UNSElect,NSRL,range**

Deselect the elements by nsrl range.

**ELEment,UNSElect,PROPerTy,range**

Deselect the elements by property range.

**ELEment,UNSElect,REVerse**

Invert element selection. Any selected elements initially invisible are plotted.

**ELEment,UNSElect,TYPE,range**

Deselect the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar

17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment,UNSElect,range**  
Deselect the elements by range.

**ELEment,VISible,ALL**  
Plot all elements

**ELEment,VISible,ASsociation,NODE**  
Plot elements by association with selected nodes

**ELEment,VISible,ASsociation,NDISPlacements**  
Plot Elements by association with selected Nodal Displacements

**ELEment,VISible,ASsociation,NFORce**  
Plot Elements by association with selected Forces

**ELEment,VISible,ASsociation,NPRESSure**  
Plot Elements by association with selected Nodal Pressures

**ELEment,VISible,ASsociation,PRESSure**  
Plot Elements by association with selected Pressures

**ELEment,VISible,ASsociation,RIGidLink**  
Plot Elements by association with selected Rigid Links

**ELEment,VISible,FORMulation,range**

Plot the elements by formulation. The value of the integers in range are:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**ELEment, VISible, MATerial, range**

Plot the elements by material range.

**ELEment, VISible, NKTP, range**

Plot the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment, VISible, NONe**

Erase all elements

**ELEment, VISible, NORDR, range**

Plot the elements by Node Order range.

**ELEment, VISible, NSRL, range**

Plot the elements by NSRL range.

**ELEment, VISible, PROPerty, range**

Plot the elements by property range.

**ELEment, VISible, TYPE, range**

Plot the elements by element type range.

The element type is the NISA NKTP number and is as below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer



112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**ELEment, VISible, SElected**  
Plot selected elements

**ELEment, VISible, range**  
Plot element range.

**GROup, ADD, SElected**  
Add selected elements to the current group.

**GROup, DESElect, groupname**  
Deselect elements in group 'groupname'.

**GROup, ERAse, groupname**  
Erase elements in 'groupname'

**GROup, PLOt, groupname**  
Plot elements in group 'groupname'.

**GROup, SElect, groupname**  
Select elements in group 'groupname'.

**GROup, SETcurrent, groupname**  
Sets the current group as 'groupname'.

**GROup, UNSElect, groupname**  
Deselect elements in group 'groupname'.

**GROup, VISible, groupname**  
Plot elements in group 'groupname'.

## 2.2.14 Rigid Links

The following commands relate to Rigid Links

**RIGidlink,ADD,linknumber,masternodenummer,UX,UY,UZ,MX,MY,MZ,slavenodenumbers**  
Adds Rigid Link number 'linknumber'. The master node number is 'masternodenummer'. The slave nodes are restrained in the UX, UY, UZ, MX, MY and MZ directions if a '1' is specified for each direction, and not if a '0' is specified. The rigid link has slave nodes specified by the comma separated 'slavenodenumbers' list. There must be at least 1 slave node for the link to be created.

It is not envisaged that this command will need to be used manually.

**RIGidlink,COLour**  
Opens a dialogue box to specify the rigid link colour.

**RIGidlink,COMPactids[,start,increment]**  
Compact Rigid Link ID's starting at the optional 'start', with an increment of 'increment'.

**RIGidlink,DELeTe,ALL**  
Delete all Rigid Links.

**RIGidlink,DELeTe,SELeCted**  
Delete selected Rigid Links

**RIGidlink,DELeTe,range**  
Delete a range of Rigid Links

**RIGidlink,DOF,UX,UY,UZ,MX,MY,MZ**  
Sets the degrees of freedom for the next Rigid Link added interactively. The slave nodes are restrained in the UX, UY, UZ, MX, MY and MZ directions if a '1' is specified for each direction, and not if a '0' is specified.

**RIGidlink,ERAsE,ALL**  
Erase all Rigid Links

**RIGidlink,ERAsE,NONe**  
Plot All Rigid Links

**RIGidlink,ERAsE,SELeCted**  
Erase selected Rigid Links

**RIGidlink,ERAsE,range**  
Erase all Rigid Links in 'range'

**RIGidink,INSPect**  
Starts the interactive Rigid Link inspection routine

**RIGidlink,NODEReplace,rigidlinknumber,oldnodenummer,newnodenummer**  
Replace all instances of 'oldnodenummer' with 'newnodenummer' in Rigid Link 'rigidlinknumber'.

**RIGidlink,PLOt,ALL**  
Plot all rigid links

**RIGidlink,PLOt,NONe**  
Erase all rigid links

**RIGidlink,PLOt,SELeCted**  
Plot all selected rigid links

**RIGidlink,PLOt,range**  
Plot all rigid links in 'range'

**RIGidlink,REN**umber,old linknumber,newlinknumber  
Renumber link with 'oldlinknumber' as 'newlinknumber'.

**RIGidlink,SE**Lect,**ALL**  
Select all rigid links.

**RIGidlink,SE**Lect,**NON**e  
Deselect all Rigid Links

**RIGidlink,SE**Lect, **NOT**  
Invert Rigid Link Selection

**RIGidlink,SE**Lect,**RE**Verse  
Invert Rigid Link Selection

**RIGidlink,SE**Lect,range  
Select all rigid links within 'range'

**RIGidlink,UNSE**Lect,**ALL**  
Deselect all rigid links

**RIGidlink,UNSE**Lect,**NON**e  
Select all rigid links

**RIGidlink,UNSE**Lect, **NOT**  
Invert rigid link selection

**RIGidlink,UNSE**Lect,**RE**Verse  
Invert rigid link selection

**RIGidlink,UNSE**Lect,range  
Deselect all rigid links within 'range'

**RIGidlink,VIS**ible,**ALL**  
Plot all rigid links

**RIGidlink,VIS**ible,**NON**e  
Erase all rigid links

**RIGidlink,VIS**ible,**SE**Lected  
Plot all selected rigid links

**RIGidlink,VIS**ible,range  
Plot all rigid links in 'range'

### 2.2.15 Analysis Titles

The following commands relate to the Analysis Titles.

**TITLe,ADD,title**

Replaces the current analysis title with 'title'

**TITLe,CLEar**

Clears the current analysis title.

## 2.2.16 Materials

The following commands relate to Materials.

**MATER**ial,**ADD**,materialnumber,materialname,EX,EY,EZ,NUXY,NUXZ,NUYZ,GXY,GXZ,GYZ,DENS,FXT,FYT,FZT,FXC,FYC,FZC,FSXY,FSXZ,FSYZ,F12,F13,F23,FS,FTS,ALPX,ALPY,ALPZ,KXX,KYY,KZZ,C,isotropic

Add material 'materialnumber' with name 'materialname'.

EX,EY and EZ are the Young's modulus in the X,Y and Z directions.

NUXY,NUXZ,NUYZ = Poisson's ratio in XY,XZ and YZ directions.

GXY,GXZ,GYZ = Shear Moduli in XY,XZ and YZ directions.

DENS = Density.

FXT,FYT,FZT = Tensile failure stresses in X,Y and Z directions.

FXC,FYC,FZC = Compressive failure stresses in X,Y and Z directions.

FSXY,FSXZ,FSYZ,F12,F13,F23,FS,FTS = shear failure stresses.

ALPX,ALPY,ALPZ = Thermal expansion co-efficients in X,Y and Z directions.

KXX,KYY,KZZ = Thermal Conductivities in X,Y and Z directions.

C = Specific heat.

Isotropic = flag to indicate whether isotropic. 1=isotropic, 0 = orthotropic.

It is not envisaged that this command will need to be used manually.

**MATER**ial,**DELE**te,**ALL**

Delete all materials

**MATER**ial,**DELE**te,range

Delete all materials with material number within range.

**MATER**ial,**INSP**ect

Opens dialogue box to dynamically inspect materials.

**MATER**ial,**REPL**ace,materialnumber,materialname,EX,EY,EZ,NUXY,NUXZ,NUYZ,GXY,GXZ,GYZ,DENS,FXT,FYT,FZT,FXC,FYC,FZC,FSXY,FSXZ,FSYZ,F12,F13,F23,FS,FTS,ALPX,ALPY,ALPZ,KXX,KYY,KZZ,C,isotropic  
Replaces Material 'materialnumber' with new data.

It is not envisaged that this command will need to be used manually.

### 2.2.17 Laminations

The following commands relate to the NISA specific 'LAMInation Definition Sets'. Where a composite element is used, the element material does not refer directly to the normal material number, it refers to the Lamination Definition set. It is the lamination definition set that specifies the material number, fibre orientation number and thickness number of each layer. NISA knows to look in the right place for the laminated elements.

**LAMInation,ADD**,laminatenum,ibreorientation,laminatename,{propertynumber,lamanglenumber,materialnumber} repeated for each layer

Adds a lamination definition set. The set number is defined by 'laminatenum'. The fibreorientation is defined by

0	=	Right Hand Rule about shell normal
1	=	RH rule about global Z axis, starting at global X axis.
2	=	RH rule about global X axis, starting at global Y axis.
3	=	RH rule about global Y axis, starting at global Z axis.
4	=	RH rule about shell normal, starting at projection of Global X axis on shell.
5	=	RH rule about shell normal, starting at projection of Global Y axis on shell.
6	=	RH rule about shell normal, starting at projection of Global Z axis on shell.
7	=	RH Rule about top surface normal.
8	=	RH Rule about mid surface normal.

For each layer {propertynumber,lamanglenumber,materialnumber} needs to be repeated.

For example:-

```
Lamination,Add,1,5,Two Layer Laminate,1,1,1,2,1,3
```

Is a two layer laminate, with number 1, the zero degree fibre orientation is based on the RH rule, starting at the projection of the Global X axis on the shell. Layer 1 has Property, Lamination Angle and Material numbers all set to 1. Layer 2 has thickness defined by Property 2, Lamination Angle set 1 and Material 3.

It is not envisaged that this command will need to be used manually.

**LAMInation,DELeTe,ALL**

Delete all laminates

**LAMInation,DELeTe,range**

Delete all laminates with laminate number within 'range'.

**LAMInation,INSPect**

Opens dialogue box to dynamically inspect lamination definition sets.

### 2.2.18 Lamination Angles

The following commands relate to the NISA specific '**LAM**ination **A**ngle Definition Sets'.

**LAM**Angle,**ADD**,lamanglenumber,lamanglename,nodal angle data

Adds a lamination angle set with number 'lamanglenumber'. It has name 'lamanglename'. The angle at each specified node is given in 'nodal angle data'. The data is comma separated.

**LAM**Angle,**DEL**ete,**ALL**

Delete all lamination angles

**LAM**Angle,**DEL**ete,range

Delete all lamination angle sets within 'range'.

**LAM**Angle,**INSP**ect

Opens dialogue box to dynamically inspect lamination angle sets.

### 2.2.19 Properties

These commands relate to properties. Properties are used for numerous different meanings. For mass elements, the property defines the mass and (if appropriate) inertia of the element. For Beams, the section properties are defined. For shells it's the section thickness at each node &c.

**PROPerTy,ADD**,propertynumber,property type,propertyname,propertydata

Add property number 'propertynumber' with property name 'propertyname'. The property data is specified as comma separated data values. The property type value is to be:-

0	=	Shell Nodal Thicknesses
1	=	3D Beam Section Properties
2	=	2D Beam Section Properties
3	=	2D Translational Spring
4	=	3D Translational Spring
5	=	Torsional Spring
6	=	General Spring
7	=	Mass
8	=	2D Mass/Inertia <b>**NISA Only**</b>
9	=	3D Mass/Inertia <b>**NISA Only**</b>
10	=	2D General Mass <b>**NISA Only**</b>
11	=	3D General Mass <b>**NISA Only**</b>
12	=	2D Gap
13	=	3D Gap
14	=	2D Spar
15	=	3D Spar/Strut
16	=	<b>**For NISA Compatibility only **</b> 3D Cable
17	=	<b>**For NISA Compatibility only **</b> 3D Damper
18	=	<b>**For NISA Compatibility only **</b> 3D Elbow
19	=	<b>**For NISA Compatibility only **</b> 3D Pipe
20	=	<b>**For NISA Compatibility only **</b> 2D Gap/Friction
21	=	<b>**For NISA Compatibility only **</b> 3D Gap/Friction
22	=	<b>**For NISA Compatibility only **</b> 3D Thermal Bar
23	=	<b>**For NISA Compatibility only **</b> 3D Convection Link
24	=	<b>**For NISA Compatibility only **</b> 3D Radiation Link

**PROPerTy,DELEte,ALL**

Delete all properties.

**PROPerTy,DELEte,range**

Delete all properties with property number within 'range'

**PROPerTy,INSPEct**

Opens dialogue box to dynamically inspect properties.



## 2.2.20 Load Case Commands

The following commands relate to load case sets.

**LCTitle,ADD**,loadcasenumber,title

Add load case title 'title' to load case set 'loadcasenumber'.

**LCTitle,DELEte,ALL**

Delete all load case titles.

**LCTitle,DELEte,range**

Delete all load case titles with load case number within 'range'.

**LOADcase,ADD**,loadcasenumber,tsfree,tol,kelfr,krctn,kstr,kstn,lq1,lq2,lq7

Add load case number 'loadcasenumber'.

tsfree = Stress free temperature,

tol = tolerance for node point force unbalance printout.

Kelfr = element strain energy, element internal forces and rigid element forces calculation flag. 1 = calculate, 0 = do not calculate.

Krctn = reaction force calculation flag. 1 = calculate, 0 = do not calculate.

Kstr = element stress calculation key. 0 = Do not calculate. 1 = at Gauss points only. 2 = at node points only. 3 = at Gauss and node points. 4 = at element centroid. 5 = at Gauss points and centroid. 6 = at node points and centroid. 7 = at Gauss, node points and centroid.

Kstn = element centroidal strain calculation flag. 1 = calculate, 0 = do not calculate.

Lq1 = Element Stress printout key. -1 = do not print. 0 = Print all. 1 = print global components. 2 = print principal and equivalent stresses (eg Von Mises). 3 = print local stresses only. 4 = print global, principal and equivalent stresses. 5 = Print global and local stresses. 6 = Print local, principal and equivalent stresses.

Lq2 = Average Nodal stresses. 0 = do not print. 1 = print global, principal and equivalent stresses. 2 = print global stresses. 3 = print principal and equivalent stresses. 4 = calculate all, but do not print.

Lq7 = displacement co-ordinate system. 0 = print in local system at node if defined, and also in global system. 1 = print in global system only. 2 = print in local system at node if defined, or in global system if local system not defined at node.

**LOADcase,AUTOadd,1/0**

Specify whether to autocreate load case definitions when a boundary condition is defined. If 1 = autocreate, 0 = do not autocreate.

**LOADcase,DELEte,ALL**

Delete all load cases

**LOADcase,DELEte,range**

Delete all load cases with load case number within 'range'.

**LOADcase,INSPect**

Opens dialogue box to dynamically inspect load case definition sets

**LOADcase,SETtings**,loadcasenumber,tsfree,tol,kelfr,krctn,kstr,kstn,lq1,lq2,lq7

Specify the default settings for the auto creation load case.

### 2.2.21 Nodal Displacements

The following commands relate to Nodal Displacements. The 'NDisp' of each command is derived from 'Nodal Displacement'.

#### **NDisp,ADD**

Opens a dialogue box to enable addition of nodal displacements to the currently selected nodes.

**NDisp,ADD,nodenum,loadcase,ux,uy,uz,mx,my,mz,uxdefined,uydefined,uzdefined,mxdefined,mydefined,mzdefined[,visible,selected]**

Add a nodal displacement to node 'nodenum' for load case 'loadcase'. The displacements are specified in 'ux,uy,uz,mx,my,mz'. Whether a displacement is applicable is defined by the Boolean value of 'uxdefined,uydefined,uzdefined,mxdefined,mydefined,mzdefined', where '1' is entered if the displacement is defined and '0' is entered if the displacement is not defined. The optional entries are 1 or 0 values indicating whether the displacement is to be visible or selected (1 = yes, 0 = no). The default values if not specified are visible, but not selected.

For example:-

```
NDisp,Add,1,2,0,0,2,0,0,0,1,1,1,0,1,0,1,0
```

Indicates that Node 1 is to be restrained in load case 2. The UX and UY displacements are 0.0. The UZ displacement is defined as 2.0. My is also specified as 0.0. The displacement is to be visible, but not selected.

It is not envisaged that this command will need to be used manually.

#### **NDisp,DELEte,ALL**

Delete all nodal displacements

#### **NDisp,DELEte,SELEcted**

Delete selected nodal displacements

#### **NDisp,DELEte,nodenum,casenum**

Delete nodal displacement on node 'nodenum' for load case 'casenum'.

#### **NDisp,ERASE,ALL**

Erase all nodal displacements

#### **NDisp,ERASE,ASSociation,NODE**

Erase all nodal displacements associated with currently selected nodes. Any selected nodal displacements are deselected as they are erased.

#### **NDisp,ERASE,ASSociation, ELEment**

Erase all nodal displacements associated with currently selected elements. Any selected nodal displacements are deselected as they are erased.

#### **NDisp,ERASE,CASE,range**

Erase all nodal displacements with load case numbers within 'range'

#### **NDisp,ERASE,INDividual,nodenum,casenum**

Erase nodal displacement on node 'nodenum' for load case 'casenum'.

#### **NDisp,ERASE,NODE,range**

Erase all nodal displacements on nodes with nodenumbers within 'range'.

#### **NDisp,ERASE,NONE**

Plot all nodal displacements

#### **NDisp,ERASE,SELEcted**

Erase selected nodal displacements

**NDISp,MODify,nodenum**ber,loadcase,ux,uy,uz,mx,my,mz,uxdefined,uydefined,uzdefined,mxdefined,mydefined,mzdefined[,visible,selected]

Change a nodal displacement to node 'nodenum' for load case 'loadcase'. The displacements are specified in 'ux,uy,uz,mx,my,mz'. Whether a displacement is applicable is defined by the Boolean value of 'uxdefined,uydefined,uzdefined,mxdefined,mydefined,mzdefined', where '1' is entered if the displacement is defined and '0' is entered if the displacement is not defined. The optional entries are 1 or 0 values indicating whether the displacement is to be visible or selected (1 = yes, 0 = no). The default values if not specified are visible, but not selected.

For example:-

```
NDISp,Add,1,2,0,0,2,0,0,0,1,1,1,0,1,0,1,0
```

Indicates that Node 1 is to be restrained in load case 2. The UX and UY displacements are 0.0. The UZ displacement is defined as 2.0. My is also specified as 0.0. The displacement is to be visible, but not selected.

Where a node is not constrained in the applicable loadcase, the node is ignored.

It is not envisaged that this command will need to be used manually.

**NDISp,NODE**replace,oldnodenum,ewnodenum,loadcasenum,ux,uy,uz,mx,my,mz,uxdefined,uydefined,uzdefined,mxdefined,mydefined,mzdefined,visible,selected

Similar to **NDISp,ADD,nodenum**... except that if the loadcase is found on 'oldnodenum' then that is deleted and a nodal displacement is added on node 'ewnodenum'. If one or other node is not found, the operation has no consequence.

**NDISp,PLOT,ALL**

Plot all nodal displacements

**NDISp,PLOT,ASSociation,NODE**

Plot nodal displacements by association with currently selected nodes.

**NDISp,PLOT,ASSociation, ELEMent**

Plot nodal displacements by association with currently selected elements.

**NDISp,PLOT,CASe,range**

Plot all nodal displacements with load case within 'range'.

**NDISp,PLOT,INDividual,nodenum,casenum**

Plot nodal displacement on node 'nodenum' for load case 'casenum'.

**NDISp,PLOT,NODE,range**

Plot nodal displacements with nodenum within 'range'.

**NDISp,PLOT,NONE**

Erase all nodal displacements

**NDISp,SElect,ALL**

Select all nodal displacements

**NDISp,SElect,ASSociation,NODE**

Select nodal displacements by association with currently selected nodes.

**NDISp,SElect,ASSociation,ELEMent**

Select nodal displacements by association with currently selected elements.

**NDISp,SElect,CASe,range**

Select all nodal displacements with load case within 'range'.

**NDISp,SElect,INDividual,nodenum,casenum**

Select nodal displacement on node 'nodenum' for load case 'casenum'.

**NDISp,SElect,NODE,range**

Select nodal displacements with nodenumber within 'range'.

**NDISp,SElect,NONe**

Deselect all nodal displacements

**NDISp,SElect,NOT**

Invert selection of nodal displacements

**NDISp,SElect,REVerse**

Invert selection of nodal displacements

**NDISp,UNSElect,INDividual,nodenumber,casenumbe**

Deselect nodal displacement on node 'nodenumber' for load case 'casenumbe'.

**NDISp,VISible,ALL**

Plot all nodal displacements

**NDISp,VISible,ASSociation,NODE**

Plot nodal displacements by association with currently selected nodes.

**NDISp,VISible,ASSociation, ELEment**

Plot nodal displacements by association with currently selected elements.

**NDISp,VISible,CASe,range**

Plot all nodal displacements with load case within 'range'.

**NDISp,VISible,INDividual,nodenumber,casenumbe**

Plot nodal displacement on node 'nodenumber' for load case 'casenumbe'.

**NDISp,VISible,NODE,range**

Plot nodal displacements with nodenumber within 'range'.

**NDISp,VISible,NONe**

Erase all nodal displacements

## 2.2.22 Pressures

The following commands relate to Pressures on element faces.

### **PRESsure,ADD**

Opens a dialogue box to enable addition of pressures to the currently selected elements.

### **PRESsure,ADD, elementnumber,loadcase,facenumber,pressure[,visible,selected]**

Add a pressure load to element 'elementnumber', face 'facenumber' for load case 'loadcase'. The pressure replaces any existing loads on the element face for the specified load case. The pressure value is 'pressure'. Visible and selected have value of 1 or 0 to indicate whether the force is to be visible or selected (1 = yes, 0 = no). The default values if not specified are visible, but not selected.

For example:-

```
Pressure,Add,1,6,2,1.4e-1,1,0
```

Indicates that Element 1, face number 6 is to be loaded in load case 2. The pressure is 1.4e-1. The pressure is to be visible, but not selected.

It is not envisaged that this command will need to be used manually.

### **PRESsure,DELEte,ALL**

Delete all pressure forces

### **PRESsure,DELEte,SELEcted**

Delete selected pressure forces

### **PRESsure,DELEte,elementnumber,casenumber,facenumber**

Delete nodal pressure force on element 'elementnumber' for load case 'casenumber'. The pressure deleted is the one applied to face 'facenumber'.

### **PRESsure,ERAsE,ALL**

Erase all pressure forces

### **PRESsure,ERAsE,ASSociation,NODE**

Erase all pressures associated with currently selected nodes. Any selected pressures are deselected as they are erased.

### **PRESsure,ERAsE,ASSociation,ELEment**

Erase all pressure forces associated with currently selected elements. Any selected forces are deselected as they are erased.

### **PRESsure,ERAsE,CAsE,range**

Erase all pressure forces with load case numbers within 'range'

### **PRESsure,ERAsE,INDividual,elementnumber,facenumber,casenumber**

Erase pressure force on element 'elementnumber', face 'facenumber' for load case 'casenumber'.

### **PRESsure,ERAsE,ELEment,range**

Erase all pressures on elements with element numbers within 'range'.

### **PRESsure,ERAsE,NONE**

Plot all nodal pressure forces

### **PRESsure,ERAsE,SELEcted**

Erase selected pressure forces

### **PRESsure,PLOt,ALL**

Plot all pressure forces

**PRESsure,PLOt,ASSociation,NODe**

Plot pressure forces by association with currently selected nodes.

**PRESsure,PLOt,ASSociation,ELEment**

Plot pressure forces by association with currently selected elements.

**PRESsure,PLOt,CASe,range**

Plot all pressure forces with load case within 'range'.

**PRESsure,PLOt,INDividual,elementnumber,face,case,case,case**

Plot pressure force on element 'elementnumber', face 'face' for load case 'case'.

**PRESsure,PLOt,ELEment,range**

Plot pressure forces with element number within 'range'.

**PRESsure,PLOt,NONe**

Erase all pressure forces

**PRESsure,SELEct,ALL**

Select all pressure forces

**PRESsure,SELEct,ASSociation,NODe**

Select pressure forces by association with currently selected nodes.

**PRESsure,SELEct,ASSociation,ELEment**

Select pressure forces by association with currently selected elements.

**PRESsure,SELEct,CASe,range**

Select all pressure forces with load case within 'range'.

**PRESsure,SELEct,INDividual,elementnumber,face,case,case,case**

Select pressure force on element 'elementnumber', face 'face' for load case 'case'.

**PRESsure,SELEct,ELEment,range**

Select pressure forces with element number within 'range'.

**PRESsure,SELEct,NONe**

Deselect all pressure forces

**PRESsure,SELEct,NOT**

Invert selection of pressure forces

**PRESsure,SELEct,REVErse**

Invert selection of pressure forces

**PRESsure,UNSELEct,INDividual,elementnumber,face,case,case,case**

Deselect nodal pressure force on element 'elementnumber', face 'face' for load case 'case'.

**PRESsure,VISible,ALL**

Plot all pressure forces

**PRESsure,VISible,ASSociation,NODe**

Plot pressures by association with currently selected nodes.

**PRESsure,VISible,ASSociation,ELEment**

Plot pressure forces by association with currently selected elements.

**PRESsure,VISible,CASe,range**

Plot all pressure forces with load case within 'range'.

**PRES**sure,**VIS**ible,**IND**ividual,elementnumber,facenumber,casenumber

Plot pressure force on element 'elementnumber', face 'facenumber' for load case 'casenumber'.

**PRES**sure,**VIS**ible,**NOD**e,range

Plot pressure forces with nodenumber within 'range'.

**PRES**sure,**VIS**ible,**NON**e

Erase all pressure forces

### 2.2.23 Nodal Pressures

The following commands relate to Nodal Pressures on nodes. The 'NPres' of each command is derived from 'Nodal Pressure'.

#### **NPRESsure,ADD**

Opens a dialogue box to enable addition of nodal pressures to the currently selected elements. This command is limited to shell elements only.

#### **NPRESsure,ADD,nodenumber,loadcase,fx,fy,fz[,visible,selected]**

Add a force representing a component of a nodal pressure load to node 'nodenumber' for load case 'loadcase'. The load is added to any existing loads on the node for the specified load case. The forces are specified in 'fx,fy,fz'. Visible and selected have value of 1 or 0 to indicate whether the force is to be visible or selected (1 = yes, 0 = no). The default values, if not specified, are visible but not selected.

For example:-

```
NPPressure,Add,1,2,20,10,2,1,0
```

Indicates that Node 1 is to be loaded in load case 2. The FX, FY and FZ forces are 20, 10 and 2. The force is to be visible, but not selected.

It is not envisaged that this command will need to be used manually.

#### **NPRESsure,DELEte,ALL**

Delete all nodal pressure forces

#### **NPRESsure,DELEte,SELEcted**

Delete selected nodal pressure forces

#### **NPRESsure,DELEte,nodenumber,casenumber**

Delete nodal pressure force on node 'nodenumber' for load case 'casenumber'.

#### **NPRESsure,ERASE,ALL**

Erase all nodal pressure forces. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,ASSociation,NODE**

Erase all nodal pressure forces associated with currently selected nodes. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,ASSociation, ELEMENT**

Erase all nodal pressure forces associated with currently selected elements. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,CASE,range**

Erase all nodal pressure forces with load case numbers within 'range'. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,INDividual,nodenumber,casenumber**

Erase nodal pressure force on node 'nodenumber' for load case 'casenumber'. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,NODE,range**

Erase all forces on nodes with node numbers within 'range'. Any selected nodal pressure forces are deselected as they are erased.

#### **NPRESsure,ERASE,NONE**

Plot all nodal pressure forces

#### **NPRESsure,ERASE,SELEcted**

Erase selected nodal pressure forces. All selected nodal pressure forces are deselected as they are erased.



**NPRESsure,PLOt,ALL**

Plot all nodal pressure forces

**NPRESsure,PLOt,ASSociation,NODE**

Plot nodal pressure forces by association with currently selected nodes.

**NPRESsure,PLOt,ASSociation,ELEment**

Plot nodal pressure forces by association with currently selected elements.

**NPRESsure,PLOt,CASe,range**

Plot all nodal pressure forces with load case within 'range'.

**NPRESsure,PLOt,INDividual,nodenumbeR,casenumbeR**

Plot nodal pressure force on node 'nodenumbeR' for load case 'casenumbeR'.

**NPRESsure,PLOt,NODE,range**

Plot nodal pressure forces with nodenumbeR within 'range'.

**NPRESsure,PLOt,NONE**

Erase all nodal pressure forces

**NPRESsure,SELEct,ALL**

Select all nodal pressure forces

**NPRESsure,SELEct,ASSociation,NODE**

Select nodal pressure forces by association with currently selected nodes.

**NPRESsure,SELEct,ASSociation,ELEment**

Select nodal pressure forces by association with currently selected elements.

**NPRESsure,SELEct,CASe,range**

Select all nodal pressure forces with load case within 'range'.

**NPRESsure,SELEct,INDividual,nodenumbeR,casenumbeR**

Select nodal pressure force on node 'nodenumbeR' for load case 'casenumbeR'.

**NPRESsure,SELEct,NODE,range**

Select nodal pressure forces with nodenumbeR within 'range'.

**NPRESsure,SELEct,NONE**

Deselect all nodal pressure forces

**NPRESsure,SELEct,NOT**

Invert selection of nodal pressure forces

**NPRESsure,SELEct,REVERSE**

Invert selection of nodal pressure forces

**NPRESsure,UNSELEct,INDividual,nodenumbeR,casenumbeR**

Deselect nodal pressure force on node 'nodenumbeR' for load case 'casenumbeR'.

**NPRESsure,VISible,ALL**

Plot all nodal pressure forces

**NPRESsure,VISible,ASSociation,NODE**

Plot nodal pressure forces by association with currently selected nodes.

**NPRESsure,VISible,ASSociation,ELEment**

Plot nodal pressure forces by association with currently selected elements.

**NPRESsure, VISible, CASe, range**

Plot all nodal pressure forces with load case within 'range'.

**NPRESsure, VISible, INDividual, nodenumber, casenumber**

Plot nodal pressure force on node 'nodenumber' for load case 'casenumber'.

**NPRESsure, VISible, NODE, range**

Plot nodal pressure forces with nodenumber within 'range'.

**NPRESsure, VISible, NONE**

Erase all nodal pressure forces

## 2.2.24 Forces

The following commands relate to Forces on nodes. The 'NForce' of each command is derived from 'Nodal Forces'.

### **NFORce,ADD**

Opens a dialogue box to enable addition of forces to the currently selected nodes.

**NFORce,ADD,nodenumber,loadcase,fx,fy,fz[,mx,my,mz,fxdefined,fydefined,fzdefined,mxdefined,mydefined,mzdefined,visible,selected]**

Add a force to node 'nodenumber' for load case 'loadcase'. The forces are specified in 'fx,fy,fz,mx,my,mz'. Whether a force is applicable is defined by the Boolean value of 'uxdefined, uydefined, uzdefined, mxdefined, mydefined, mzdefined', where '1' is entered if the force is defined and '0' is entered if the force is not defined. Visible and selected have value of 1 or 0 to indicate whether the force is to be visible or selected (1 = yes, 0 = no). The default values if not specified are visible, but not selected.

For example:-

```
NForce,Add,1,2,20,10,2,0,.01,995e77,1,1,1,0,1,0,1,0
```

Indicates that Node 1 is to be loaded in load case 2. The FX, FY and FZ forces are 20, 10 and 2. My is also specified as 0.01. Despite a value being given for Mz, as mzdefined = 0, no moment about the z axis is registered. The force is to be visible, but not selected.

Alternatively:-

```
NForce,Add,1,2,20,10,2
```

Indicates that Node 1 is to be loaded in load case 2. The FX, FY and FZ forces are 20, 10 and 2.

It is not envisaged that this command will need to be used manually.

### **NFORce,DELeTe,ALL**

Delete all forces

### **NFORce,DELeTe,SELeCted**

Delete selected forces

### **NFORce,DELeTe,nodenumber,casenumbeR**

Delete force on node 'nodenumbeR' for load case 'casenumbeR'.

### **NFORce,ERAsE,ALL**

Erase all forces

### **NFORce,ERAsE,ASSociation,NODE**

Erase all forces associated with currently selected nodes. Any selected forces are deselected as they are erased.

### **NFORce,ERAsE,ASSociation,ELEment**

Erase all forces associated with currently selected elements. Any selected forces are deselected as they are erased.

### **NFORce,ERAsE,CASe,range**

Erase all forces with load case numbers within 'range'

### **NFORce,ERAsE,INDividual,nodenumbeR,casenumbeR**

Erase force on node 'nodenumbeR' for load case 'casenumbeR'.

### **NFORce,ERAsE,NODE,range**

Erase all forces on nodes with node numbers within 'range'.

**NFORCE,ERASE,NONE**

Plot all forces

**NFORCE,ERASE,SELECTED**

Erase selected forces

**NFORCE,MODIFY,nodenum,loadcase,fx,fy,fz[,mx,my,mz,fxdefined,fydefined,fzdefined,mxdefined,mydefined,mzdefined,visible,selected]**

Modify the force to node 'nodenum' for load case 'loadcase'. The forces are specified in 'fx,fy,fz,mx,my,mz'. Whether a force is applicable is defined by the Boolean value of 'uxdefined, uydefined, uzdefined, mxdefined, mydefined, mzdefined', where '1' is entered if the force is defined and '0' is entered if the force is not defined. Visible and selected have value of 1 or 0 to indicate whether the force is to be visible or selected (1 = yes, 0 = no). The default values if not specified are visible, but not selected.

For example:-

```
NForce,Add,1,2,20,10,2,0,.01,995e77,1,1,1,0,1,0,1,0
```

Indicates that Node 1 is to be loaded in load case 2. The FX, FY and FZ forces are 20, 10 and 2. My is also specified as 0.01. Despite a value being given for Mz, as mzdefined = 0, no moment about the z axis is registered. The force is to be visible, but not selected.

Alternatively:-

```
NForce,Add,1,2,20,10,2
```

Indicates that Node 1 is to be loaded in load case 2. The FX, FY and FZ forces are 20, 10 and 2.

Where nodes are not loaded in the model for the applicable loadcase, the load will not be applied

It is not envisaged that this command will need to be used manually.

**NFORCE,PLOT,ALL**

Plot all forces

**NFORCE,PLOT,ASSOCIATION,NODE**

Plot forces by association with currently selected nodes.

**NFORCE,PLOT,ASSOCIATION,ELEMENT**

Plot forces by association with currently selected elements.

**NFORCE,PLOT,CASE,range**

Plot all forces with load case within 'range'.

**NFORCE,PLOT,INDIVIDUAL,nodenum,casenum**

Plot force on node 'nodenum' for load case 'casenum'.

**NFORCE,PLOT,NODE,range**

Plot forces with nodenum within 'range'.

**NFORCE,PLOT,NONE**

Erase all forces

**NFORCE,SELECT,ALL**

Select all forces

**NFORCE,SELECT,ASSOCIATION,NODE**

Select forces by association with currently selected nodes.

**NFORCE,SELECT,ASSOCIATION,ELEMENT**

Select forces by association with currently selected elements.

**NFORce,SELEct,CASe,range**

Select all forces with load case within 'range'.

**NFORce,SELEct,INDividual,nodenumbe,r,casenumbe,r**

Select force on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NFORce,SELEct,NODE,range**

Select forces with nodenumbe,r within 'range'.

**NFORce,SELEct,NONE**

Deselect all forces

**NFORce,SELEct,NOT**

Invert selection of forces

**NFORce,SELEct,REVERSE**

Invert selection of forces

**NFORce,UNSELEct,INDividual,nodenumbe,r,casenumbe,r**

Deselect force on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NFORce,VISible,ALL**

Plot all forces

**NFORce,VISible,ASSociation,NODE**

Plot forces by association with currently selected nodes.

**NFORce,VISible,ASSociation, ELEMENT**

Plot forces by association with currently selected elements.

**NFORce,VISible,CASe,range**

Plot all forces with load case within 'range'.

**NFORce,VISible,INDividual,nodenumbe,r,casenumbe,r**

Plot force on node 'nodenumbe,r' for load case 'casenumbe,r'.

**NFORce,VISible,NODE,range**

Plot forces with nodenumbe,r within 'range'.

**NFORce,VISible,NONE**

Erase all forces

### 2.2.25 Accelerations

The following commands relate to accelerations. The '**BodyF**' of each command is derived from '**Body Force**'.

**BODYForce,ADD**, loadcasenumber,omegax,omegay,omegaz,accelx,accely,accelz,alphax,alphay,alphaz,  
xo,yo,zo

Add an acceleration to the model for load case 'loadcasenumber'. The linear accelerations are defined by 'accelx,accely,accelz'. For centrifugal accelerations the origin of the rotation is as 'xo,yo,zo'. The angular velocity is defined by 'omegax,omegay,omegaz' and the angular acceleration is defined by 'alphax,alphay,alphaz'.

It is not envisaged that this command will need to be used manually.

**BODYForce,DELeTe,ALL**

Delete all accelerations.

**BODYForce,DELeTe,range**

Delete all accelerations with load case number within 'range'

**BODYForce,INSPect**

Open a dialogue window to inspect the existing accelerations within the model.

## 2.2.26 Texture Mapping

The following commands relate to texture mapping.

**ELEment,TEXTuremap,ADD,ALL,left,bottom,right,top**

Texture map all elements into area defined by the rectangle left,bottom,right,top. All values defining the rectangle must be between 0.0 and 1.0. eg

```
ELE,TEXTuremap,ADD,ALL,0.0,0.0,1.0,1.0
```

**ELEment,TEXTuremap,ADD,SElected,left,bottom,right,top**

Texture map selected elements into area defined by the rectangle left,bottom,right,top. All values defining the rectangle must be between 0.0 and 1.0.

**ELEment,TEXTuremap,BOX,ALL,left,bottom,right,top**

Texture map all elements. The mapping is by means of creating an imaginary box around the model. All faces are then projected onto the appropriate face of the box, with the face normal being the vector that decides which box face the element face projects onto. The imaginary box is then unwrapped into area defined by the rectangle left,bottom,right,top while keeping all proportions consistent. All existing texturemapping is deleted and all existing texture nodes are deleted. All values defining the rectangle must be between 0.0 and 1.0. eg

```
ELE,TEXTuremap,BOX,ALL,0.0,0.0,1.0,1.0
```

**ELEment,TEXTuremap,BOX,SElected,left,bottom,right,top**

Texture map selected elements. The mapping is by means of creating an imaginary box around the selected nodes in the model. All faces are then projected onto the appropriate face of the box, with the face normal being the vector that decides which box face the element face projects onto. The imaginary box is then unwrapped into area defined by the rectangle left,bottom,right,top while keeping all proportions consistent. All values defining the rectangle must be between 0.0 and 1.0. Note: the mapping size and proportions WILL NOT be the same as if all faces were mapped at that time. All existing texture mapping on the element faces is deleted, but no texture nodes are deleted.

**ELEment,TEXTuremap,DELEte,ALL**

Delete texture mapping on all elements

**ELEment,TEXTuremap,DELEte,FACe,elementnumber,face,face,face**

Delete the texture mapping on element 'elementnumber', face 'face,face,face'

**ELEment,TEXTuremap,DELEte,SElected**

Delete texture mapping on the selected elements.

**ELEment,TEXTuremap,FACe,elementnumber,face,face,face,texturenodenumberlist....**

Apply a texture map to element number 'elementnumber' and face 'face,face,face'. The texture node numbers are specified in texturenodenumberlist .

**TEXTURENodes,ADD,nodenum,location,location**

Add texture node 'nodenum' at location x,y

**TEXTURENodes,DELEte,ALL**

Delete texture mapping on all elements

**TEXTURENodes,DELEte,nodenum**

Delete texture node 'nodenum'

**TEXTURETemplate,SIZE,x,y**

Sets the width of the texture template as 'x', and the height as 'y'.

## 2.2.27 Miscellaneous

The following miscellaneous commands may be entered.

### **CLOSE**

Closes the current modeller window.

### **EXIT**

Leave the program.

### **EXPORT,OBJ,filename**

Exports the nodes and element faces as a wavefront .obj model. Where texture mapping is defined, this is exported too.

### **EXPORT,MYSTRAN,filename**

Export a MYSTRAN FE analysis file

### **EXPORT,NEU,filename**

Export as much of the model as possible in EMRC's Display III Neutral Format. The write up on the format is limited, consequently it is not possible to export all the entities in the model.

### **EXPORT,NE2,filename**

Export as much of the model as possible in EMRC's Display II Neutral Format. The format is limited, consequently it is not possible to export all the entities in the model.

### **EXPORT,NISA,filename**

Export a NISA FE analysis file.

### **GROup,ADD,SELEcted**

Add selected elements to the current group.

### **GROup,DESELEct,groupname**

Deselect elements in group 'groupname'.

### **GROup,ERASE,groupname**

Erase elements in 'groupname'

### **GROup,PLOt,groupname**

Plot elements in group 'groupname'.

### **GROup,SELEct,groupname**

Select elements in group 'groupname'.

### **GROup,SETcurrent,groupname**

Sets the current group as 'groupname'.

### **GROup,UNSELEct,groupname**

Deselect elements in group 'groupname'.

### **GROup,VISible,groupname**

Plot elements in group 'groupname'.

### **HELP**

Opens the help file, an action which won't help, because help files never do.

### **IMPORT,DXF,filename**

Imports AutoCAD .dxf file 'filename'. New file is merged with existing model.

### **IMPORT,GEO,filename**

Imports COSMOSWorks .geo file 'filename'. New file is merged with existing model.



**IMPORT,OBJ,filename**

Imports Wavefront .obj file 'filename'. New file is merged with existing model.

**IMPORT,RADioss,filename**

Imports Radioss .rad file 'filename'. New file is merged with existing model.

**IMPORT,STL,filename**

Imports Stereo Lithography .stl file 'filename'. New file is merged with existing model.

**LAYer, ADD, SElected**

Add selected entities to the current layer.

**LAYer,DESElect,layername**

Deselect all entities on layer 'layername'.

**LAYer,ERASE,layername**

Erase all entities on layer 'layername'.

**LAYer,PLOt,layername**

Plot all entities on layer 'layername'.

**LAYer,SElect,layername**

Select all entities on layer 'layername'

**LAYer,SETcurrent,layername**

Sets the current layer as 'layername'.

**LAYer,UNSElect,layername**

Deselect all entities on layer 'layername'.

**LAYer,VISible,layername**

Plot all entities on layer 'layername'.

**OPEN,filename**

Opens file 'filename' in the modeller. The existing model is overwritten. May open the following file types:-

- AutoCAD drawing exchange files (.dxf)

- Modeller Data files (.mdf)

- Modeller Session files (.msf)

- NISA Input Files (.nis)

**QUIT**

Leave the program.

**STATUS**

Opens a dialogue window to dynamically inspect information about the model.

**STOP**

Stops the current action. Probably. Well, occasionally. Some sections of the program are written to permit interruptions and simulate multi tasking – but not all as it introduces a processor overhead and slows the program down. If the program permits you to type 'STOP' while an action is occurring, then it can be stopped. However, as the PC will be operating at very high processor utilisation rates and will react very slowly to typing 'stop'. It is a last resort if something is taking an absurd length of time and should only be tried just before stopping the process with 'Ctrl-Alt-Delete'.

### 2.2.28 Commands to fix bugs

The following commands may be entered to try to repair the model if bugs have been found to corrupt the model (like not being able to snap to line midpoints because the midpoint has been calculated in the wrong place).

**LINE,CALculate,MIDPOINTS**

Calculates midpoints on all lines. This is a remedial function to help, in the event of a bug, if lines have their mid points calculated in the wrong place.

**SURface,CALculate,NORmalcentroid**

Recalculates the surface normal and centroid in the event of a bug causing the information to be out of date.

**ELEment,CALculate,CENtroid**

Recalculates the element face centroid and centroid in the event of a bug causing the information to be out of date.

**ELEment,CALculate,NORmal**

Recalculates the element normal, in the event of a bug causing the information to be out of date.

**ELEment,CALculate,NORmalcentroid**

Recalculates the element normal, face centroid and centroid in the event of a bug causing the information to be out of date.

**ELEment,CENtroid,RECalculate**

Recalculates the element centroids. Intended to correct an error in element copy/move operations if the elements cannot be subsequently selected. In a program with no bugs, this will be redundant.

**ELEment,REPair,NORDR**

Run through all the elements and, based on the numbers of nodes in the element and the element type, sets the Node Order. This can be useful after a difficult to repair bug occurs.

**ELEment,REPair,NSRL**

Recalculates all the element NSRL numbers based on the element Node Order and Type. This can be useful after a difficult to repair bug occurs.

### 3. Menu Command Reference

#### 3.1 General Information

This menu command reference identifies all the menu commands in the Chestnut Pens Modelling Studio. All the entries are in the order found in their respective menus, rather than alphabetically.

Each menu item is prefaced by its menu path. So 'View| Plot| Displacement Constraints| All' may be found by going to the 'View' menu, 'Plot' submenu, 'Displacement Constraints' sub-submenu and selecting the 'All' menu item.

Many menu commands require an additional range to be entered so that the selection/ plotting etc. may be applied to a limited range of entities. Where a range is specified it's possible to use various delimiters. More than one integer may be specified separated by commas or slashes. Also a consecutive range may be indicated by a '-' or a 't'. In consecutive ranges make sure that the first number is smaller than the second.

For example:-

Node, Delete, 2/4, 1, 7t9, 15-19

Indicates that nodes 1, 2, 4, 7, 8, 9, 15, 16, 17, 18 & 19 are to be deleted, whereas the following:-

Node, Delete, 19-15

Is not acceptable.

#### 3.2 Menu List

The following menu items are categorised:-

3.2.1	File Menu .....	109
3.2.2	Edit Menu .....	111
3.2.3	View Menu .....	114
	View  Plot Menu .....	114
	View  Erase Menu .....	124
	View  Zoom Menu .....	134
	View  Angle Menu .....	134
	View  Perspective Menu.....	135
	View  Display Options Menu .....	136
	View  Colours Menu .....	139
	View  Labels Menu .....	144
	View  Light Source Menu .....	145
	View  Layers Menu.....	146
	View  Groups Menu.....	149
	View  Status .....	152
3.2.4	Selection Menu .....	153
3.2.5	Geometry Menu .....	164
3.2.6	FE Model Menu.....	174
	FE Model  Executive Commands.....	174
	FE Model  Nodes .....	177
	FE Model  Elements.....	180
	FE Model  Rigid Links .....	194
	FE Model  Analysis Title.....	196
	FE Model  Material .....	197
	FE Model  Laminations .....	200
	FE Model  Lamination Angles .....	203
	FE Model  Properties .....	205
	FE Model  Load Case .....	214
	FE Model  Displacement Constraints.....	217

	FE Model  Pressures.....	219
	FE Model  Nodal Pressures .....	221
	FE Model  Forces.....	223
	FE Model  Accelerations .....	225
	FE Model  Load Combiner .....	227
	FE Model  Analysis Readiness Check .....	229
3.2.7	Mapping Menu .....	231
3.2.8	Window Menu .....	236
3.2.9	Help Menu.....	237
3.2.10	Pop-up menu over model Viewport.....	238

### 3.2.1 File Menu

The following menu items are found in the 'File' menu.

#### 'File| New'

##### 'File| New| Text Editor'

Creates a new text editor. It forms a basic word processor that can cope with text or Rich Text Files. If the file is plain text, the text size is unlimited. For a Rich Text File, the maximum size is 64K as the MS Common Controls .dll found in Windows XP imposes that limit on the file size. For a full description of the text editor's functions, see the help file attached to the program.

##### 'File| New| Modeller'

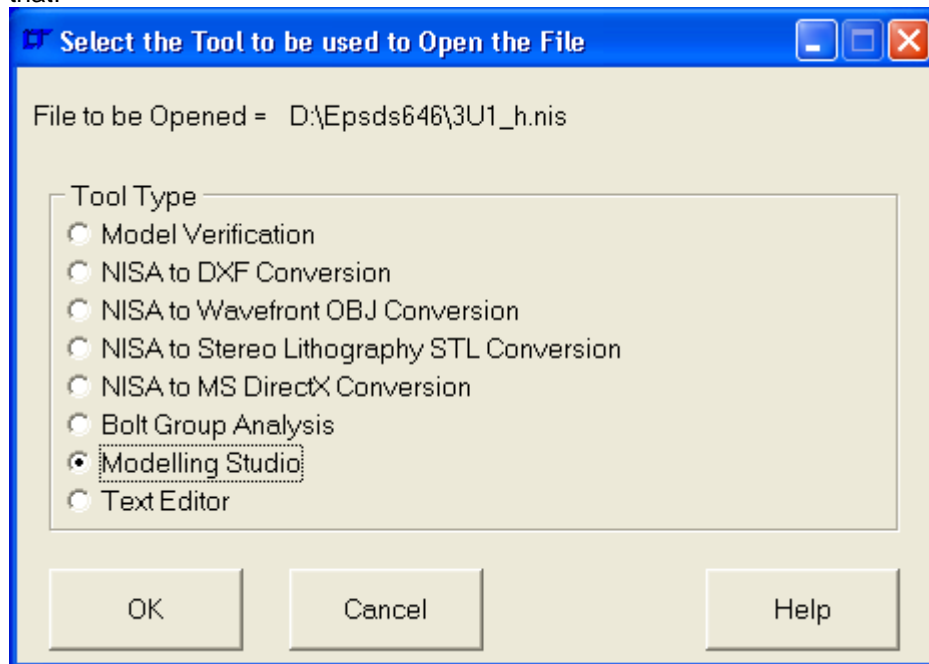
Opens a new Modelling Studio window.

##### 'File| New| Bolt Group Analysis'

Opens a new Bolt Group Analysis window. The Bolt Group analysis is outside the remit of this command reference, but it is fully explained in the program help file.

#### 'File| Open'

Open a file. The file may be one of a number of extensions. Where an option is possible, a dialogue window will open asking what is to be the destination tool. If the Modelling Studio is an option – select that!



#### 'File| Save'

Saves the current modelling studio model with the filename shown in the window frame. This will always end in .mdf for 'Modeller Data File'. The file is in a proprietary binary data format. The format is fixed so that files will always be backwards and forwards compatible. Later versions of the Modelling Studio will simply extend the data structure, and will retain all existing structures.

#### 'File| Save As'

Saves the current modelling studio model with a user entered filename. This will always end in .mdf for 'Modeller data file'. The file is in a proprietary binary data format. The format is fixed so that files will always be backwards and forwards compatible. Later versions will simply extend the data structure, and will retain all existing structures.

#### 'File| Import'

Allows the importation of a Wavefront .obj file, COSMOSWorks .geo file, Stereo Lithography .stl file or a DXF file. See the 'User Guide' for a full description of all supported commands.

**'File| Export'**

Allows the export of a Wavefront .obj file (nodes, elements and texture mapping), NISA .nis analysis file, MYSTRAN .dat analysis file. Parts of the model may also be exported as a Display III session file, a Display III .neu file, Display II .ne2 file, Ascii Stereo Lithography file or a basic DXF file. No export format is capable of supporting the full capabilities of the Modelling Studio. See the 'User Guide' for a full description of all supported commands.

**'File| Merge'**

Merge another modeller data file into the existing file. All references are kept consistent so that no part of the merged model over-writes any part of the existing model.

**'File| File Preferences'****'File| File Preferences| Import: Rotate Geo File About X axis'**

This option, if set, automatically rotates a COSMOSWorks file by +90° about the X axis to take into account the fact that SolidWorks has the Y Axis as the vertical up axis rather than the Z axis. The option is persistent and will affect the way a .geo file is opened even before a Modelling Studio is opened.

**'File| File Preferences| Import: STL as FE Entities'**

This option, if set, imports a Stereo Lithography (stl) file as elements and nodes. If not set, the file is opened as Points, Lines and Surfaces. The option is persistent and will affect the way a .stl file is opened even before a Modelling Studio is opened.

**'File| Print'**

Prints the contents of the **Progress Information** Box.

**'File| Printer Setup'**

Enables the user to configure the printer.

**'File| Send'**

Send the existing .mdf file as an e-mail attachment. This is occasionally difficult, depending on how rigorous the e-mail application is at complying with Microsoft standards. Consequently it is not usually possible to send the model directly as an attachment if MS Outlook 2003 is the e-mail provider. (Microsoft rules are for other people to follow, not Microsoft..).

**'File| Exit'**

Leave the program.

**'File| Quick List'**

List of previously opened files to speed file opening. The number of possible files listed can be altered by using 'Edit| Preferences...' and choosing the 'Quick List' tab. The maximum number of files is 10. If you need more (and there seems no reason why you should!), edit the .ini file in the program directory so that the QuickListLength under [Preferences] reads:-

```
[Preferences]
QuickListLength=[enter new number here]
```

### 3.2.2 Edit Menu

The following menu items are found in the 'Edit' menu. The 'Edit' menu is not used a great deal in creating a model.

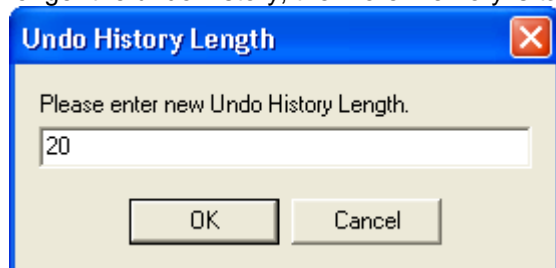
#### 'Edit| Undo'

Undo the previous operation. Due to the programming effort involved, not all operations can be undone at the time of writing this command reference. Check the **Status Bar** hint for the name of the command that would be undone.

When an undo event executes, a session file is run through the command line (for a full explanation of this see the 'Understanding Undo's' How to... Guide in the 'User Guide'). Several thousand command line commands may make up a single undo event. If the undo event takes too long to execute, press the 'Esc' key to abort the undo event.

#### 'Edit| Undo History Length'

Sets the number of undo events that can be stored. The setting is persistent, and is initially 20. The longer the undo history, the more memory is taken up.



#### 'Edit| Cut'

Deletes selected entities.

#### 'Edit| Copy'

Copies selected entities for on-screen translation.

#### 'Edit| Paste'

Pastes text from clipboard into the **Progress Information** Box.

#### 'Edit| Select All'

Selects all Points, Lines, Surfaces, Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

#### 'Edit| Font'

Changes the font used for the labels in the **Viewport**.

#### 'Edit| Word Wrap'

Permits lines to wrap in the **Progress Information** box.

#### 'Edit| Text Style'

##### 'Edit| Text Style| Bold'

Sets the text to bold at the current cursor position in the **Progress Information** box.

##### 'Edit| Text Style| Italics'

Sets the text to italics at the current cursor position in the **Progress Information** box.

##### 'Edit| Text Style| Underline'

Sets the text to underline at the current cursor position in the **Progress Information** box.

##### 'Edit| Text Style| Strikeout'

Sets the text to strikeout at the current cursor position in the **Progress Information** box.

**'Edit| Text Alignment'**

**'Edit| Text Alignment | Left'**

Sets the text alignment to left justified at the current cursor position in the **Progress Information** box.

**'Edit| Text Alignment | Centre'**

Sets the text alignment to centre justified at the current cursor position in the **Progress Information** box.

**'Edit| Text Alignment | Right'**

Sets the text alignment to right justified at the current cursor position in the **Progress Information** box.

**'Edit| Text Case'**

**'Edit| Text Case| UPPER CASE'**

Changes the selected text in the **Progress Information** box to UPPER CASE.

**'Edit| Text Case| Lower case'**

Changes the selected text in the **Progress Information** box to lower case.

**'Edit| Text Case| Initial Capitals'**

Changes the selected text in the **Progress Information** box to Initial Capitals.

**'Edit| Spell Check'**

Spell checks the **Progress Information** box.

**'Edit| Find'**

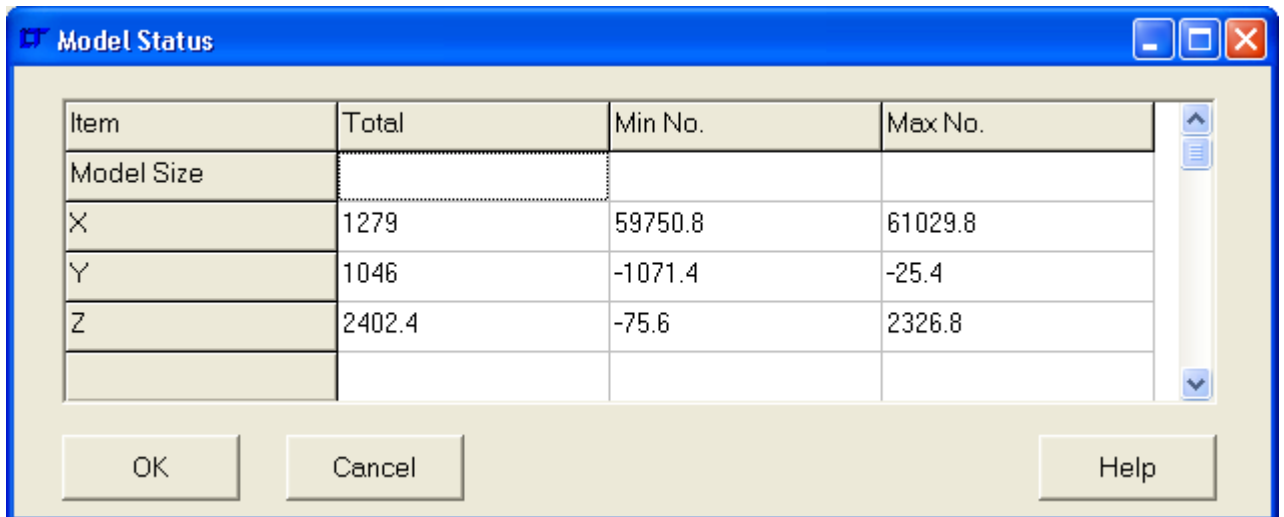
Finds text in the **Progress Information** box.

**'Edit| Replace'**

Finds and replaces text in the **Progress Information** box.

**'Edit| Statistics'**

Opens a dialogue window to dynamically inspect information about the model.

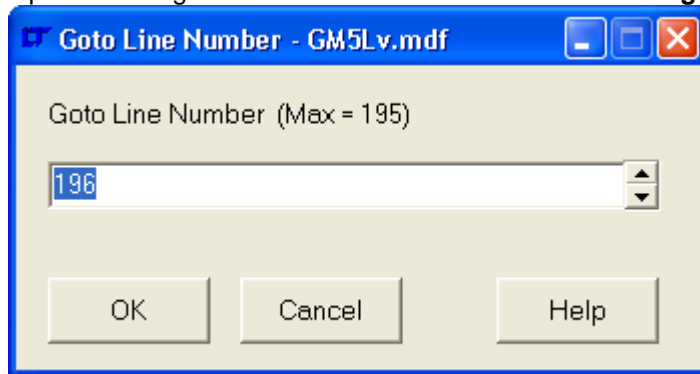


The dialogue data may be copied to the clipboard by right clicking over the data grid and selecting pop-up menu option **'Copy to Clipboard'**. The data is Tab separated on the clipboard.

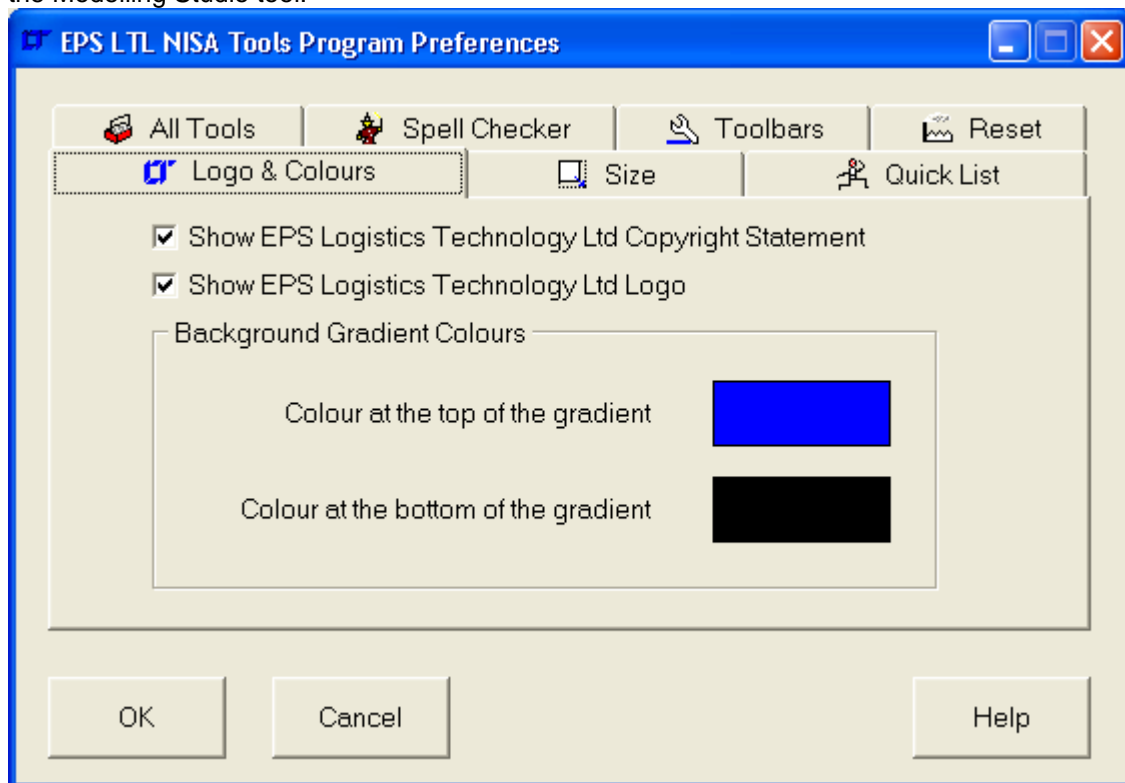


**'Edit| Goto Line..'**

Opens a dialogue window to enter the text line in the **Progress Information** box to jump to.

**'Edit| Preferences'**

Opens a dialogue window to specify the general program preferences. No preferences directly affect the Modelling Studio tool.

**'Edit| Delete Session Files'**

Delete all session files in the program directory except the session file for the current model.

**'Edit| Toolbars'**

View/erase text editing toolbars. Only the Edit toolbar is of any applicability to the modeller. The rest may be turned off without detriment to modelling. All the toolbars (except the modelling toolbar) may be pulled into floating toolbars and positioned around the screen should it be desired.

### 3.2.3 View Menu

The following menu items are found in the 'View' menu.

#### 'View| Plot'

##### 'View| Plot| All'

Plot all Points, Lines, Surfaces, Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

##### 'View| Plot| All Geometry'

Plot all Points, Lines, Surfaces and Solids.

##### 'View| Plot| All FE Model'

Plot all Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

##### 'View| Plot| All Boundary Conditions'

###### 'View| Plot| All Boundary Conditions| All'

Plot all Displacement Constraints, Pressures, Nodal Pressures and Forces.

###### 'View| Plot| All Boundary Conditions| Range by Case Number'

Plot all Displacement Constraints, Pressures, Nodal Pressures and Forces in a range of load cases. Typed input of the range is necessary in the command line.

#### 'View| Plot| Points'

##### 'View| Plot| Points| All'

Plot all Points.

##### 'View| Plot| Points| Selected'

Plot selected Points.

##### 'View| Plot| Points| Range'

Plot Points within range. Typed input of the range is necessary in the command line.

##### 'View| Plot| Points| Selected Lines'

Plot Points by association with selected Lines.

##### 'View| Plot| Points| Selected Surfaces'

Plot Points by association with selected Surfaces.

##### 'View| Plot| Points| Selected Solids'

Plot Points by association with selected Solids.

#### 'View| Plot| Lines'

##### 'View| Plot| Lines| All'

Plot all Lines.

##### 'View| Plot| Lines| Selected'

Plot selected Lines.

##### 'View| Plot| Lines| Range'

Plot Lines within range. Typed input of the range is necessary in the command line.

##### 'View| Plot| Lines| Selected Points'

Plot Lines by association with selected Points.

##### 'View| Plot| Lines| Selected Surfaces'

Plot Lines by association with selected Surfaces.

##### 'View| Plot| Lines| Selected Solids'

Plot Lines by association with selected Solids.

**'View| Plot| Surfaces'****'View| Plot| Surfaces| All'**

Plot all Surfaces.

**'View| Plot| Surfaces| Selected'**

Plot selected Surfaces.

**'View| Plot| Surfaces| Range'**

Plot Surfaces within range. Typed input of the range is necessary in the command line.

**'View| Plot| Surfaces| Selected Points'**

Plot Surfaces by association with selected Points.

**'View| Plot| Surfaces| Selected Lines'**

Plot Surfaces by association with selected Lines.

**'View| Plot| Surfaces| Selected Solids'**

Plot Surfaces by association with selected Solids.

**'View| Plot| Nodes'****'View| Plot| Nodes| All'**

Plot all Nodes.

**'View| Plot| Nodes| Selected'**

Plot selected Nodes.

**'View| Plot| Nodes| Range'**

Plot Nodes within range. Typed input of the range is necessary in the command line.

**'View| Plot| Nodes| Selected Elements'**

Plot Nodes by association with selected Elements.

**'View| Plot| Nodes| Selected Rigid Links'**

Plot Nodes by association with selected Rigid Links.

**'View| Plot| Nodes| Selected Displacements'**

Plot Nodes by association with selected Displacement Constraints.

**'View| Plot| Nodes| Selected Pressures'**

Plot Nodes by association with selected Pressures.

**'View| Plot| Nodes| Selected Nodal Pressures'**

Plot Nodes by association with selected Nodal Pressures.

**'View| Plot| Nodes| Selected Forces'**

Plot Nodes by association with selected Forces.

**'View| Plot| Elements'****'View| Plot| Elements| All'**

Plot all Elements.

**'View| Plot| Elements| Selected'**

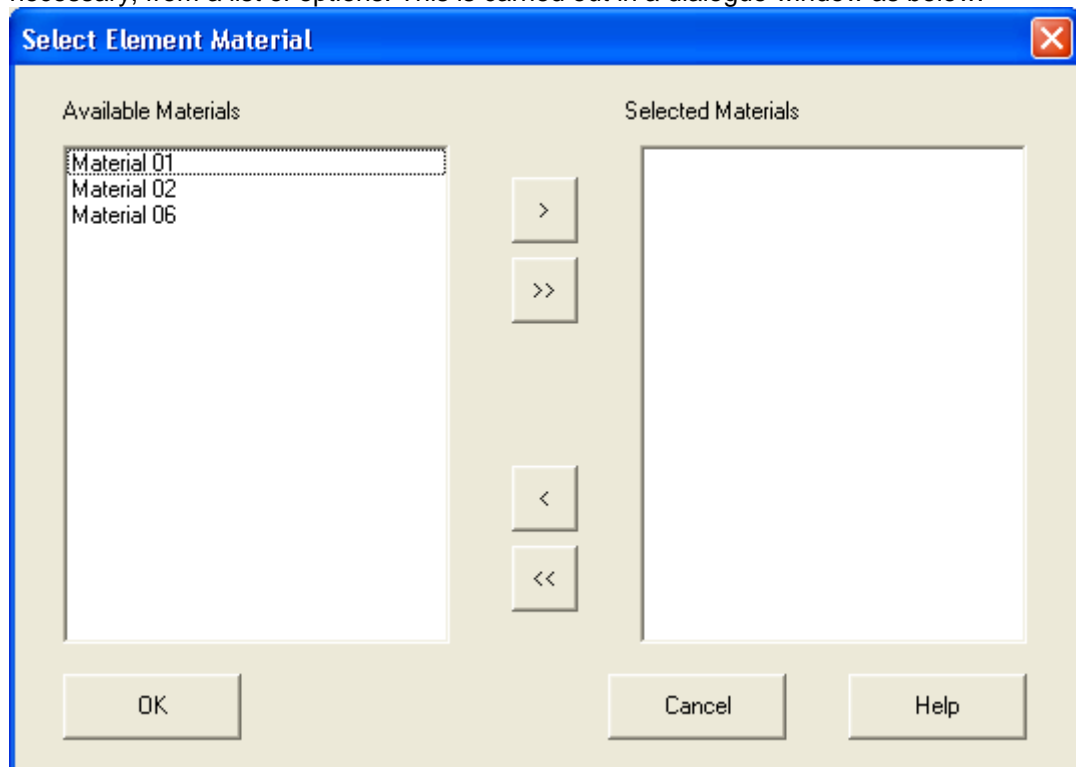
Plot selected Elements.

**'View| Plot| Elements| Range'**

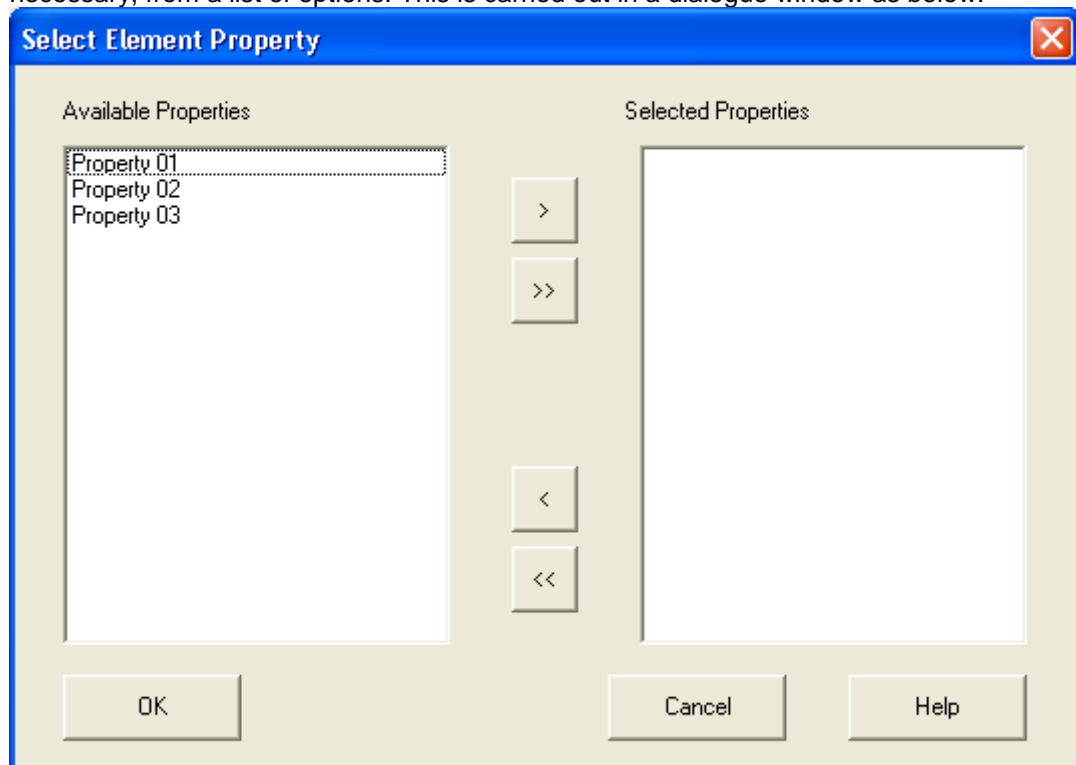
Plot Elements within range. Typed input of the range is necessary in the command line.

**'View| Plot| Elements| by Material'**

Plot Elements with material number in range. Selection of the element material range is necessary, from a list of options. This is carried out in a dialogue window as below:-

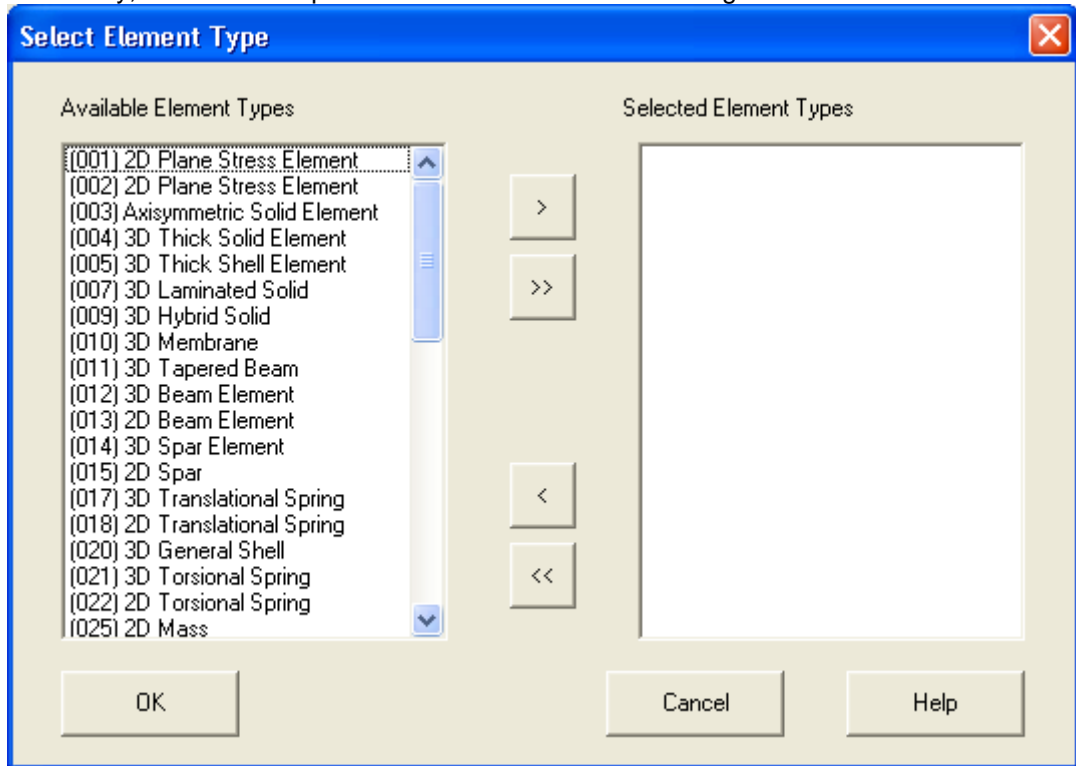
**'View| Plot| Elements| by Property'**

Plot Elements with property number in range. Selection of the element property range is necessary, from a list of options. This is carried out in a dialogue window as below:-



**‘View| Plot| Elements| by Element Type’**

Plot Elements with element type number in range. Selection of the element type range is necessary, from a list of options. This is carried out in a dialogue window as below:-



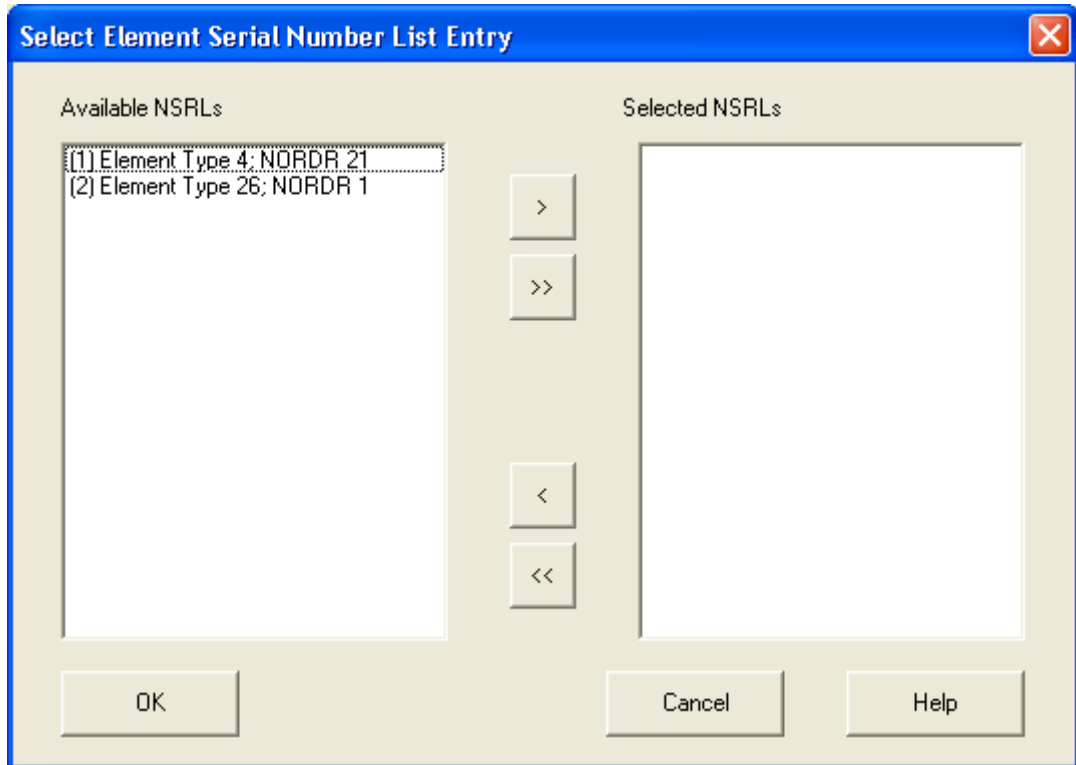
The element type number is from the list below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam

- 40 3D Thin Shell
- 41 3D Mixed Interpolation 8 Node General Shell
- 42 2D or Axisymmetric Gap
- 43 3D Gap
- 45 3D Cable
- 46 3D Straight Pipe
- 47 3D Elbow
- 48 3D Translational Damper
- 49 2D Gap/Friction
- 50 3D Gap/Friction
- 104 3D Solid for Heat Transfer
- 105 3D Thick Shell for Heat Transfer
- 112 3D Bar for Heat Transfer
- 120 3D Thin Shell for Heat Transfer
- 149 3D Convection Link for Heat Transfer
- 150 3D Radiation Link for Heat Transfer

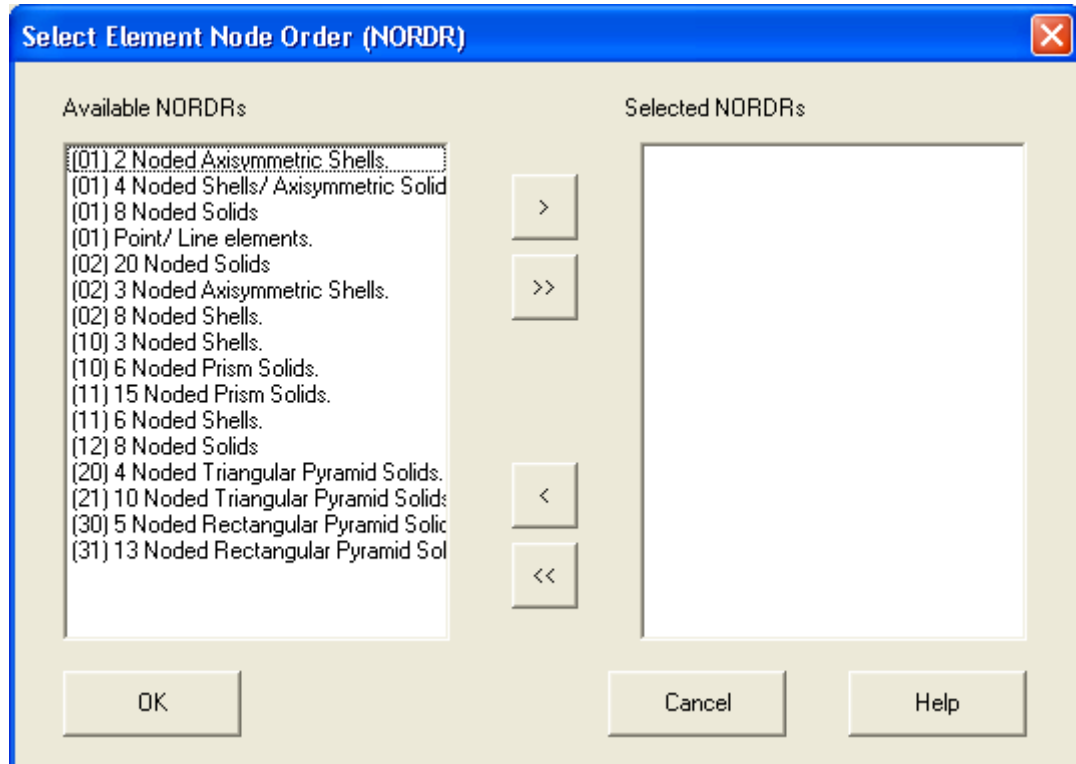
**'View| Plot| Elements| by NSRL'**

Plot Elements with NSRL number in range. The NSRL of the elements to be plotted is selected from a list of Serial Numbers generated for the current model. The selection list may be similar to the list below:-



**'View| Plot| Elements| by NORDR'**

Plot Elements with Node Order number in range. Selection of the Node Order is from a list as below:-

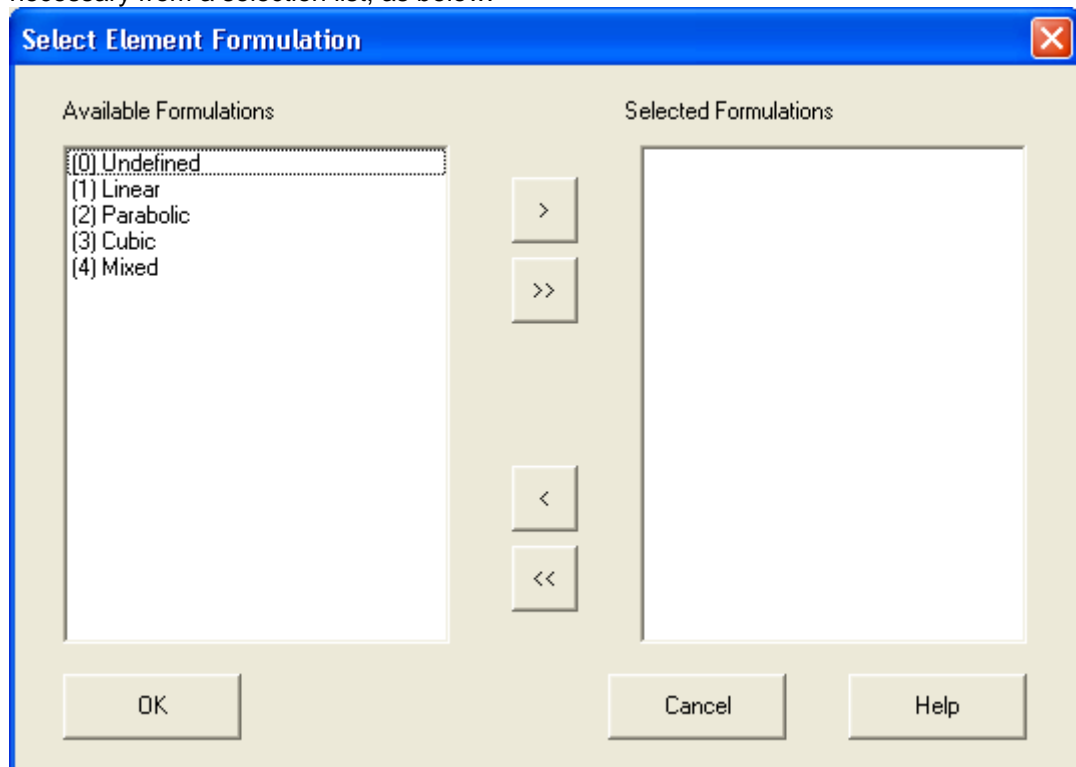


The Node Order is the node order of the element, and is defined as below:-

Node Order	Description
1	For all point or line elements
1	For all 2 noded axisymmetric shell elements
1	For all 4 noded shell elements/ 4 noded axisymmetric solid elements
1 or 12	For all 8 noded solid elements
2	For all 3 noded axisymmetric elements
2	For all 8 noded shell elements
2	For all 20 noded brick elements
10	For 3 Noded triangular shell elements
10	6 noded prism shaped brick elements
11	6 Noded triangular shell elements
11	15 noded prism brick elements
20	4 noded triangular pyramid solid elements
21	10 noded triangular pyramid solid elements
30	5 noded rectangular pyramid elements
31	13 noded rectangular pyramid elements

**'View| Plot| Elements| by Formulation'**

Plot Elements with formulation number in range. Selection of the formulation range is necessary from a selection list, as below:-



The element formulation is as below:-

Formulation	Description
0	Undefined – for selecting damaged elements prior to deletion.
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**'View| Plot| Elements| Selected Nodes'**

Plot Elements by association with selected Nodes.

**'View| Plot| Elements| Selected Rigid Links'**

Plot Elements by association with selected Rigid Links.

**'View| Plot| Elements| Selected Displacements'**

Plot Elements by association with selected Displacement Constraints.

**'View| Plot| Elements| Selected Pressures'**

Plot Elements by association with selected Pressures.

**'View| Plot| Elements| Selected Nodal Pressures'**

Plot Elements by association with selected Nodal Pressures.

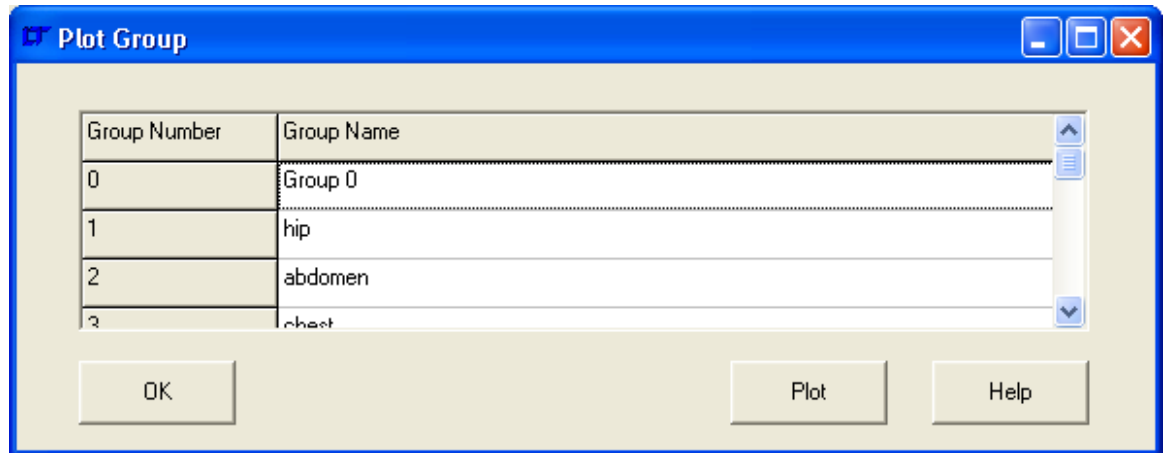
**'View| Plot| Elements| Selected Forces'**

Plot Elements by association with selected Forces.



**'View| Plot| Elements| Plot by Group Name'**

Opens a dialogue window to choose the element group name to plot.

**'View| Plot| Rigid Links'****'View| Plot| Rigid Links| All'**

Plot all Rigid Links.

**'View| Plot| Rigid Links| Selected'**

Plot selected Rigid Links.

**'View| Plot| Rigid Links| Range'**

Plot Rigid Links within range. Typed input of the range is necessary in the command line.

**'View| Plot| Displacement Constraints'****'View| Plot| Displacement Constraints| All'**

Plot all Displacement Constraints.

**'View| Plot| Displacement Constraints| Range by Node Association'**

Plot Displacement Constraints by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Plot| Displacement Constraints| Range by Case Number'**

Plot Displacement Constraints by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Plot| Displacement Constraints | Selected Nodes'**

Plot Displacement Constraints by association with selected Nodes.

**'View| Plot| Displacement Constraints | Selected Elements'**

Plot Displacement Constraints by association with selected Elements.

**'View| Plot| Pressures'****'View| Plot| Pressures| All'**

Plot all Pressures.

**'View| Plot| Pressures| Range by Element Association'**

Plot Pressures by association with a range of element numbers. Typed input of the element number range is necessary in the command line.

**'View| Plot| Pressures| Range by Face Association'**

Plot Pressures by association with a range of element face numbers. Typed input of the element face number range is necessary in the command line.

**'View| Plot| Pressures| Range by Case Number'**

Plot Pressures by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Plot| Pressures| Selected Nodes'**

Plot Pressures by association with selected Nodes.

**'View| Plot| Pressures| Selected Elements'**

Plot Pressures by association with selected Elements.

**'View| Plot| Nodal Pressures'**

**'View| Plot| Nodal Pressures| All'**

Plot all Nodal Pressures.

**'View| Plot| Nodal Pressures| Range by Node Association'**

Plot Nodal Pressures by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Plot| Nodal Pressures| Range by Case Number'**

Plot Nodal Pressures by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Plot| Nodal Pressures| Selected Nodes'**

Plot Nodal Pressures by association with selected Nodes.

**'View| Plot| Nodal Pressures| Selected Elements'**

Plot Nodal Pressures by association with selected Elements.

**'View| Plot| Forces'**

**'View| Plot| Forces| All'**

Plot all Forces.

**'View| Plot| Forces| Range by Node Association'**

Plot Forces by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Plot| Forces| Range by Case Number'**

Plot Forces by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Plot| Forces| Selected Nodes'**

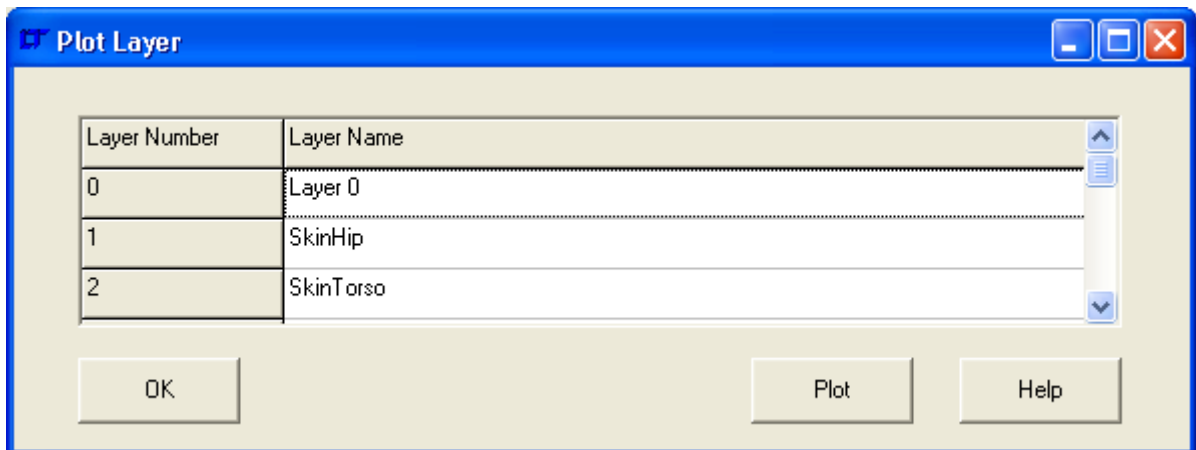
Plot Forces by association with selected Nodes.

**'View| Plot| Forces| Selected Elements'**

Plot Forces by association with selected Elements.

**'View| Plot| Plot by Layer Name'**

Opens a dialogue box to select the layers to plot.

**'View| Plot| Refresh'**

Refreshes the view. This is useful when creating surfaces as each new surface is drawn over the top of the existing model regardless of whether it should be obscured by other entities. This refreshes the plot and shuffles all entities into their correct depth order.

**'View| Plot| Shaded View'**

Determines whether shading of the plotted entities is carried out. The default is 'ON' as this enables depth shading, infinite light shading etc.

**'View| Erase'****'View| Erase| All'**

Erase all Points, Lines, Surfaces, Solids, Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'View| Erase| All Geometry'**

Erase all Points, Lines, Surfaces and Solids.

**'View| Erase| All FE Model'**

Erase all Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'View| Erase| All Boundary Conditions'****'View| Erase| All Boundary Conditions| All'**

Erase all Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'View| Erase| All Boundary Conditions| Range by Case Number'**

Erase all Displacement Constraints, Pressures, Nodal Pressures and Forces in a range of load cases. Typed input of the range is necessary in the command line.

**'View| Erase| Points'****'View| Erase| Points| All'**

Erase all Points.

**'View| Erase| Points| Selected'**

Erase selected Points.

**'View| Erase| Points| Range'**

Erase Points within range. Typed input of the range is necessary in the command line.

**'View| Erase| Points| Selected Lines'**

Erase Points by association with selected Lines.

**'View| Erase| Points| Selected Surfaces'**

Erase Points by association with selected Surfaces.

**'View| Erase| Points| Selected Solids'**

Erase Points by association with selected Solids.

**'View| Erase| Lines'****'View| Erase| Lines| All'**

Erase all Lines.

**'View| Erase| Lines| Selected'**

Erase selected Lines.

**'View| Erase| Lines| Range'**

Erase Lines within range. Typed input of the range is necessary in the command line.

**'View| Erase| Lines| Selected Points'**

Erase Lines by association with selected Points.

**'View| Erase| Lines| Selected Surfaces'**

Erase Lines by association with selected Surfaces.

**'View| Erase| Lines| Selected Solids'**

Erase Lines by association with selected Solids.

**'View| Erase| Surfaces'****'View| Erase| Surfaces| All'**

Erase all Surfaces.

**'View| Erase| Surfaces| Selected'**

Erase selected Surfaces.

**'View| Erase| Surfaces| Range'**

Erase Surfaces within range. Typed input of the range is necessary in the command line.

**'View| Erase| Surfaces| Selected Points'**

Erase Surfaces by association with selected Points.

**'View| Erase| Surfaces| Selected Lines'**

Erase Surfaces by association with selected Lines.

**'View| Erase| Surfaces| Selected Solids'**

Erase Surfaces by association with selected Solids.

**'View| Erase| Nodes'****'View| Erase| Nodes| All'**

Erase all Nodes.

**'View| Erase| Nodes| Selected'**

Erase selected Nodes.

**'View| Erase| Nodes| Range'**

Erase Nodes within range. Typed input of the range is necessary in the command line.

**'View| Erase| Nodes| Selected Elements'**

Erase Nodes by association with selected Elements.

**'View| Erase| Nodes| Selected Rigid Links'**

Erase Nodes by association with selected Rigid Links.

**'View| Erase| Nodes| Selected Displacements'**

Erase Nodes by association with selected Displacement Constraints.

**'View| Erase| Nodes| Selected Pressures'**

Erase Nodes by association with selected Pressures.

**'View| Erase| Nodes| Selected Nodal Pressures'**

Erase Nodes by association with selected Nodal Pressures.

**'View| Erase| Nodes| Selected Forces'**

Erase Nodes by association with selected Forces.

**'View| Erase| Elements'****'View| Erase| Elements| All'**

Erase all Elements.

**'View| Erase| Elements| Selected'**

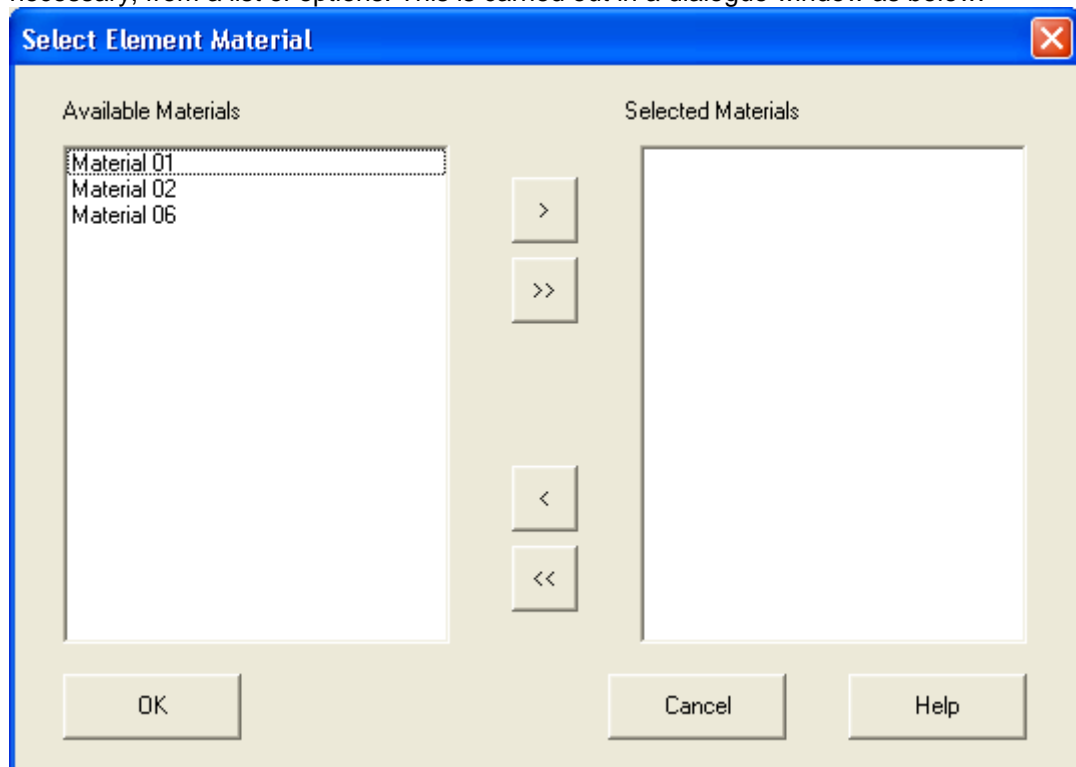
Erase selected Elements.

**'View| Erase| Elements| Range'**

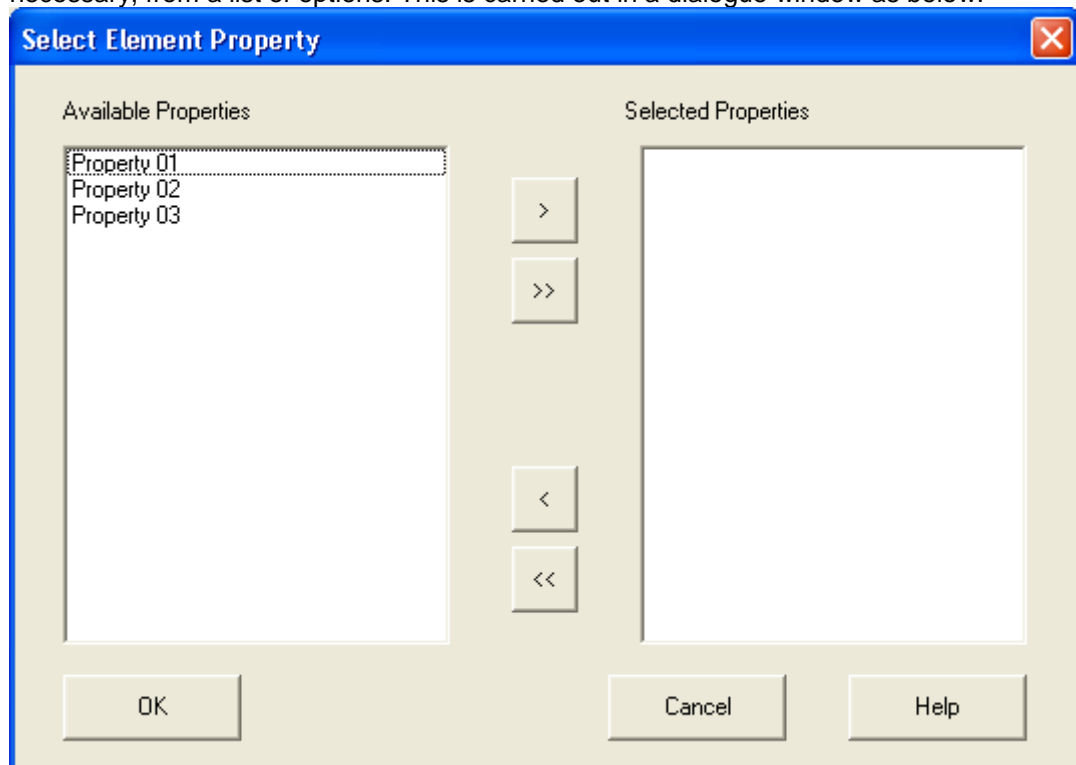
Erase Elements within range. Typed input of the range is necessary in the command line.

**'View| Erase| Elements| by Material'**

Erase Elements with material number in range. Selection of the element material range is necessary, from a list of options. This is carried out in a dialogue window as below:-

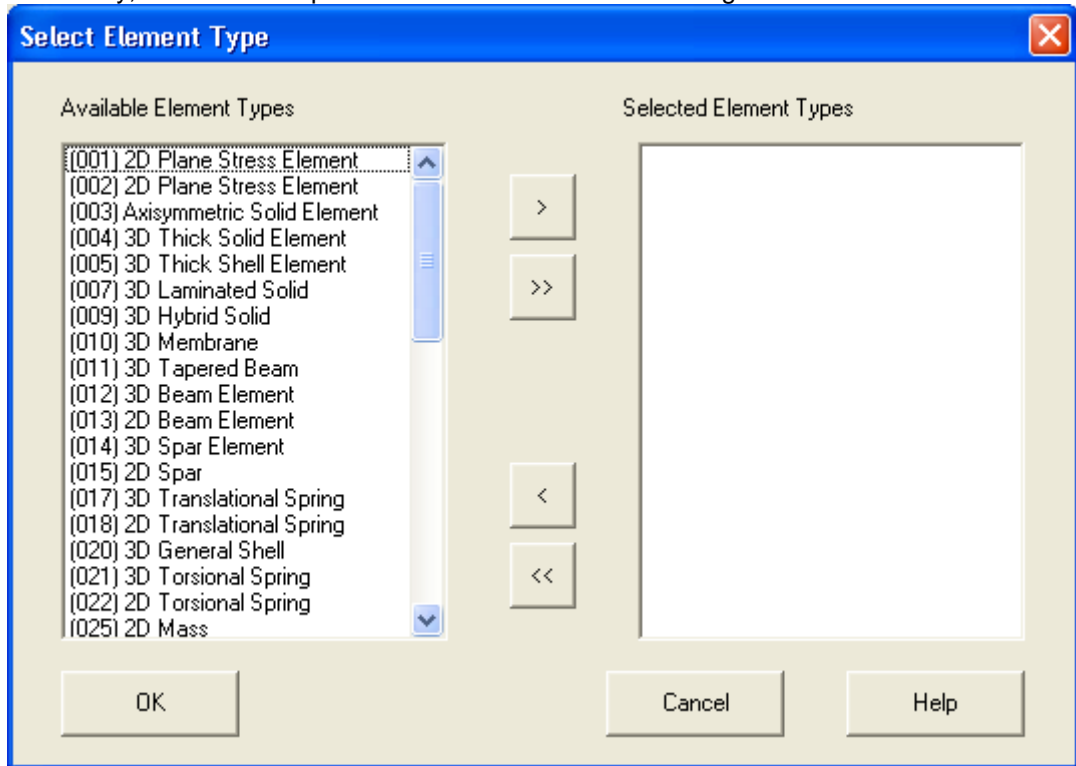
**'View| Erase| Elements| by Property'**

Erase Elements with property number in range. Selection of the element property range is necessary, from a list of options. This is carried out in a dialogue window as below:-



**‘View| Erase| Elements| by Element Type’**

Erase Elements with element type number in range. Selection of the element type range is necessary, from a list of options. This is carried out in a dialogue window as below:-



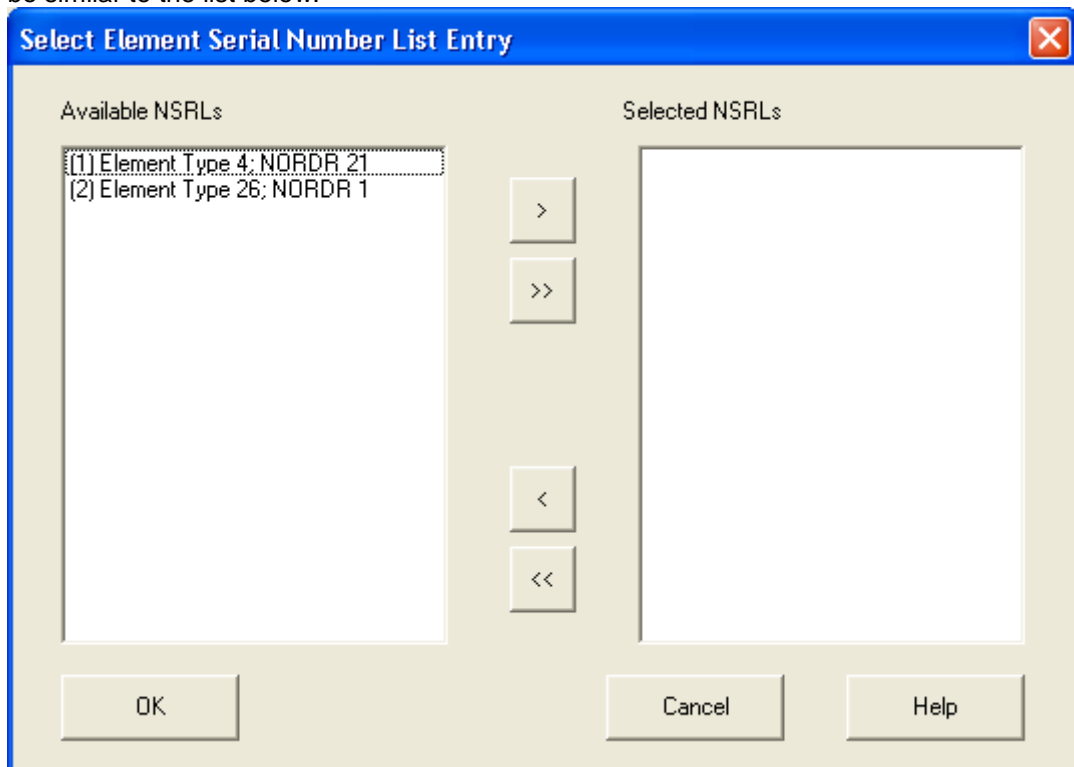
The element type number is from the list below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam

40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

**'View| Erase| Elements| by NSRL'**

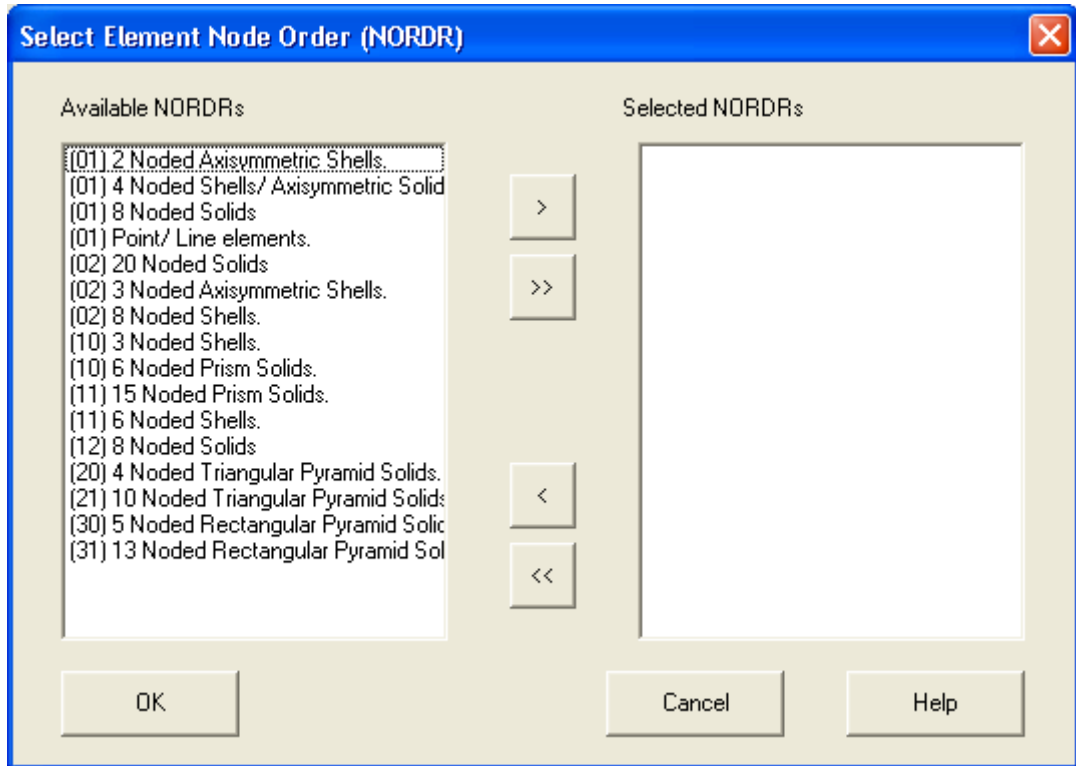
Erase Elements with NSRL number in range. The NSRL of the elements to be erased is selected from a list of Serial Numbers generated for the current model. The selection list may be similar to the list below:-





**'View| Erase| Elements| by NORDR'**

Erase Elements with Node Order number in range. Selection of the Node Order is from a list as below:-

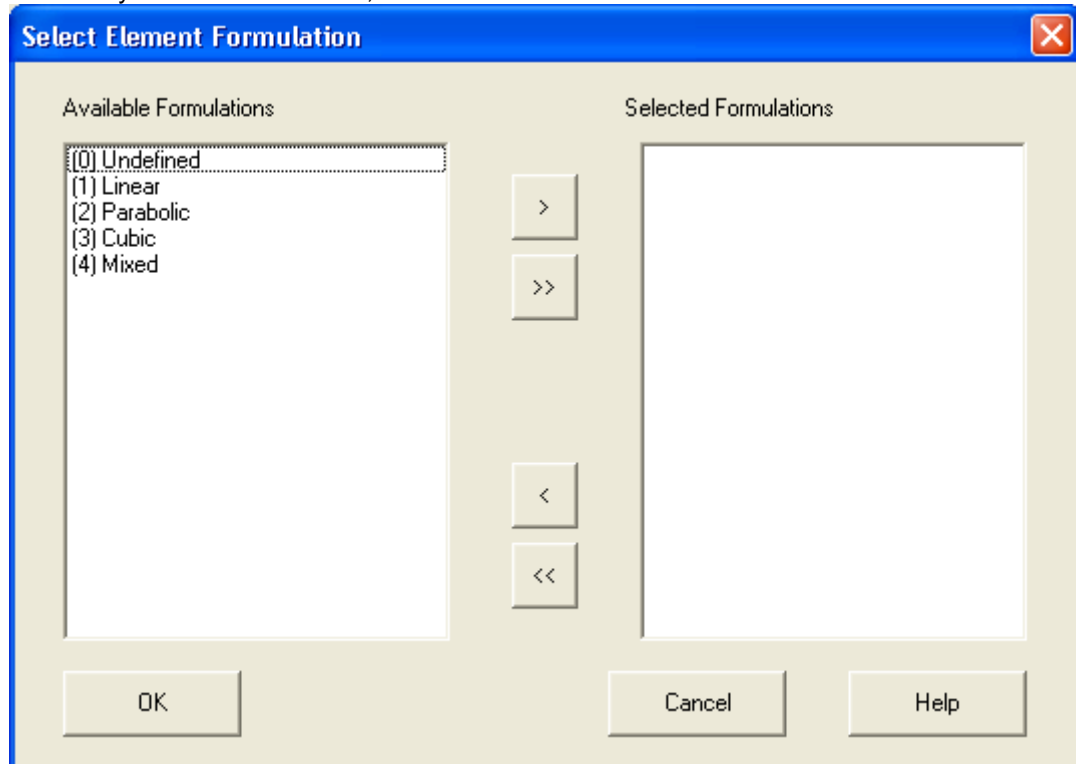


The Node Order is the NISA node order of the element, and is defined as below:-

Node Order	Description
1	For all point or line elements
1	For all 2 noded axisymmetric shell elements
1	For all 4 noded shell elements/ 4 noded axisymmetric solid elements
1 or 12	For all 8 noded solid elements
2	For all 3 noded axisymmetric elements
2	For all 8 noded shell elements
2	For all 20 noded brick elements
10	For 3 Noded triangular shell elements
10	6 noded prism shaped brick elements
11	6 Noded triangular shell elements
11	15 noded prism brick elements
20	4 noded triangular pyramid solid elements
21	10 noded triangular pyramid solid elements
30	5 noded rectangular pyramid elements
31	13 noded rectangular pyramid elements

**'View| Erase| Elements| by Formulation'**

Erase Elements with formulation number in range. Selection of the formulation range is necessary from a selection list, as below:-



Formulation is as below:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**'View| Erase| Elements| Selected Nodes'**

Erase Elements by association with selected Nodes.

**'View| Erase| Elements| Selected Rigid Links'**

Erase Elements by association with selected Rigid Links.

**'View| Erase| Elements| Selected Displacements'**

Erase Elements by association with selected Displacement Constraints.

**'View| Erase| Elements| Selected Pressures'**

Erase Elements by association with selected Pressures.

**'View| Erase| Elements| Selected Nodal Pressures'**

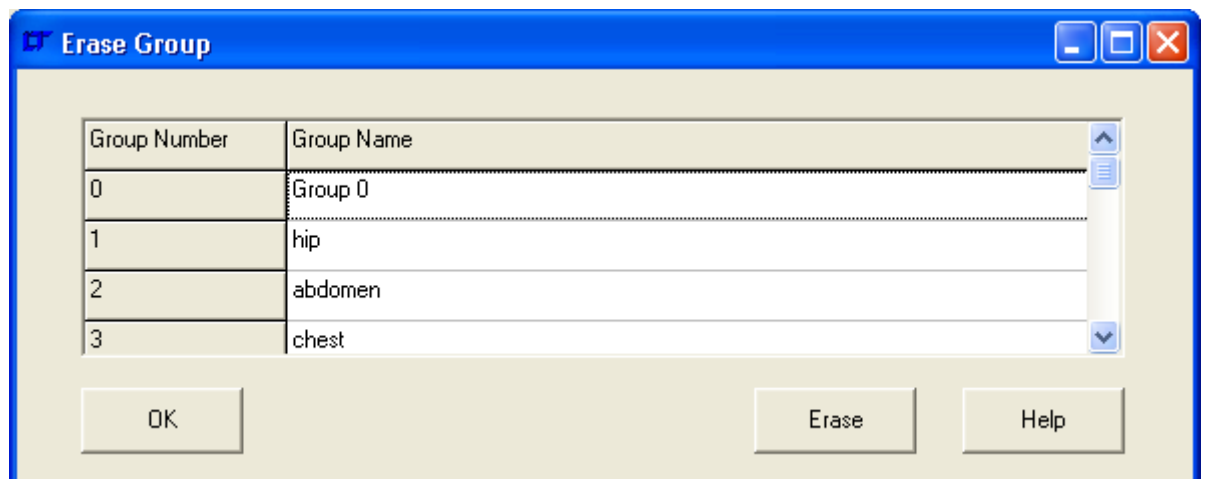
Erase Elements by association with selected Nodal Pressures.

**'View| Erase| Elements| Selected Forces'**

Erase Elements by association with selected Forces.

**'View| Erase| Elements| Erase by Group Name'**

Opens a dialogue window to choose the element group name to Erase.

**'View| Erase| Rigid Links'****'View| Erase| Rigid Links| All'**

Erase all Rigid Links.

**'View| Erase| Rigid Links| Selected'**

Erase selected Rigid Links.

**'View| Erase| Rigid Links| Range'**

Erase Rigid Links within range. Typed input of the range is necessary in the command line.

**'View| Erase| Displacement Constraints'****'View| Erase| Displacement Constraints| All'**

Erase all Displacement Constraints.

**'View| Erase| Displacement Constraints| Selected'**

Erase selected Displacement Constraints.

**'View| Erase| Displacement Constraints| Range by Node Association'**

Erase Displacement Constraints by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Erase| Displacement Constraints| Range by Case Number'**

Erase Displacement Constraints by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Erase| Displacement Constraints | Selected Nodes'**

Erase Displacement Constraints by association with selected Nodes.

**'View| Erase| Displacement Constraints | Selected Elements'**

Erase Displacement Constraints by association with selected Elements.

**'View| Erase| Pressures'****'View| Erase| Pressures| All'**

Erase all Pressures.

**'View| Erase| Pressures| Selected'**

Erase selected Pressures.

**'View| Erase| Pressures| Range by Element Association'**

Erase Pressures by association with a range of element numbers. Typed input of the element number range is necessary in the command line.

**'View| Erase| Pressures| Range by Face Association'**

Erase Pressures by association with a range of element face numbers. Typed input of the element face number range is necessary in the command line.

**'View| Erase| Pressures| Range by Case Number'**

Erase Pressures by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Erase| Pressures| Selected Nodes'**

Erase Pressures by association with selected Nodes.

**'View| Erase| Pressures| Selected Elements'**

Erase Pressures by association with selected Elements.

**'View| Erase| Nodal Pressures'****'View| Erase| Nodal Pressures| All'**

Erase all Nodal Pressures.

**'View| Erase| Nodal Pressures| Selected'**

Erase selected Nodal Pressures.

**'View| Erase| Nodal Pressures| Range by Node Association'**

Erase Nodal Pressures by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Erase| Nodal Pressures| Range by Case Number'**

Erase Nodal Pressures by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Erase| Nodal Pressures| Selected Nodes'**

Erase Nodal Pressures by association with selected Nodes.

**'View| Erase| Nodal Pressures| Selected Elements'**

Erase Nodal Pressures by association with selected Elements.

**'View| Erase| Forces'****'View| Erase| Forces| All'**

Erase all Forces.

**'View| Erase| Forces| Selected'**

Erase selected Forces.

**'View| Erase| Forces| Range by Node Association'**

Erase Forces by association with a range of node numbers. Typed input of the node number range is necessary in the command line.

**'View| Erase| Forces| Range by Case Number'**

Erase Forces by association with a range of load case numbers. Typed input of the load case number range is necessary in the command line.

**'View| Erase| Forces| Selected Nodes'**

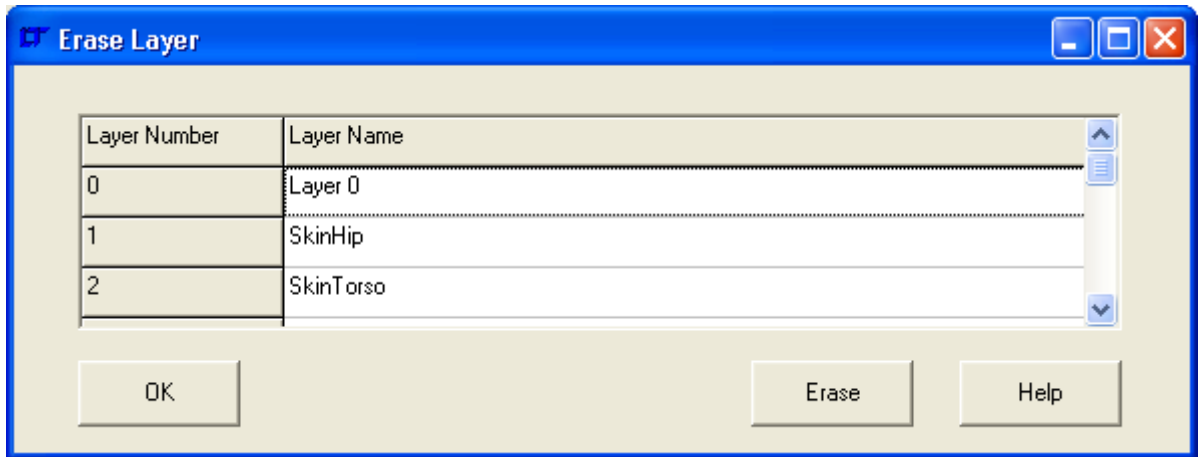
Erase Forces by association with selected Nodes.

**'View| Erase| Forces| Selected Elements'**

Erase Forces by association with selected Elements.

**'View| Erase| Erase by Layer Name'**

Opens a dialogue box to select the layers to Erase.



**'View| Zoom'****'View| Zoom| All'**

Zoom to the extents of the currently displayed entities (which may be smaller than the full model).

**'View| Zoom| Previous'**

Zoom to the previous model view. When the 'Zoom| All' menu command is used or the model is rotated, the zoom history is cleared so it is not possible to go to the previous view.

**'View| Zoom| In'**

Zoom in to the model and increase the size of the view by 15%.

**'View| Zoom| Out'**

Zoom out from the model and decrease the size of the view by 15%.

**'View| Zoom| Area'**

Zoom to an area of the model by dragging a rectangle over a section of the **Viewport**.

**'View| Angle'****'View| Angle| Front'**

View the front of the model, along the +Y axis.

**'View| Angle| Back'**

View the back of the model along the -Y axis.

**'View| Angle| Left'**

View the left side of the model along the +X axis.

**'View| Angle| Right'**

View the right side of the model along the -X axis.

**'View| Angle| Top'**

View the top of the model along the -Z axis.

**'View| Angle| Bottom'**

View the bottom of the model along the +Z axis.

**'View| Angle| Iso 1'**

View the Iso 1 view of the model.

**'View| Angle| Iso 2'**

View the Iso 1 view of the model.

**'View| Angle| Iso 3'**

View the Iso 1 view of the model.

**'View| Angle| Iso 4'**

View the Iso 1 view of the model.

**'View| Angle| User Defined'**

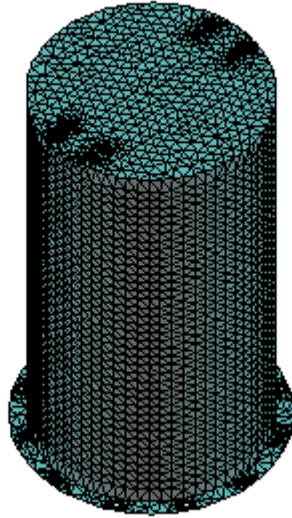
Open a dialogue box to specify the viewing angle.

**'View| Angle| Fly Around'**

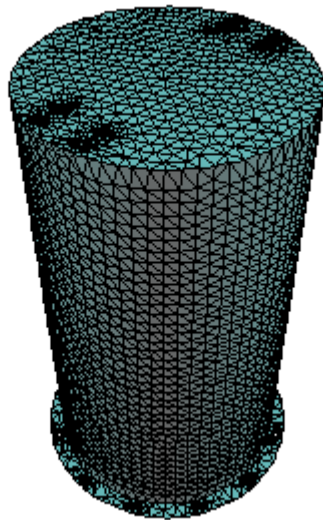
Fly around the model. The view starts at Rot X = -45 degrees, Rot Z = 0. The Z rotation is increased 15 degrees every 2 seconds. After each complete rotation about the Z axis, the X axis rotation is decreased by 15 degrees until it reaches -135, whereupon the view goes back to Rot X = -45 degrees. Press any key to abort the fly-around view.

**'View| Perspective'****'View| Perspective| None'**

No perspective effects are applied to the model view in the **Viewport**. As shown below:-

**'View| Perspective| Wide Angle Lens'**

Strong perspective effects to simulate a wide angle lens are applied to the model view in the **Viewport**. As shown below, this is from the same angle of view as the zero perspective view above.

**'View| Perspective| Standard Lens'**

Medium strength perspective effects to simulate a standard lens are applied to the model view in the **Viewport**.

**'View| Perspective| Telephoto Lens'**

Mild perspective effects to simulate a telephoto lens are applied to the model view in the **Viewport**.

**'View| Display Options'****'View| Display Options| Points'****'View| Display Options| Points| Shape'****'View| Display Options| Points| Shape| Square'**

Display points as squares in the **Viewport**.

**'View| Display Options| Points| Shape| Diamond'**

Display points as diamonds in the **Viewport**.

**'View| Display Options| Points| Shape| Circle'**

Display points as circles in the **Viewport**.

**'View| Display Options| Points| Shape| Cross'**

Display points as crosses in the **Viewport**.

**'View| Display Options| Points| Size'****'View| Display Options| Points| Size| Small'**

Set the point symbol size to small in the **Viewport**.

**'View| Display Options| Points| Size| Medium'**

Set the point symbol size to medium in the **Viewport**.

**'View| Display Options| Points| Size| Large'**

Set the point symbol size to large in the **Viewport**.

**'View| Display Options| Surfaces'****'View| Display Options| Surfaces| Fill Surfaces'**

Toggle the display style for filling surfaces.

**'View| Display Options| Surfaces| Show Edges'**

Toggle the display style for showing the edges of surfaces.

**'View| Display Options| Surfaces| Show Normals'**

Toggle the display style for showing the surface normals.

**'View| Display Options| Surfaces| Normal Arrow Size'****'View| Display Options| Surfaces| Normal Arrow Size| Small'**

Set the normal arrow symbol size to small in the **Viewport**.

**'View| Display Options| Surfaces| Normal Arrow Size| Medium'**

Set the normal arrow size to medium in the **Viewport**.

**'View| Display Options| Surfaces| Normal Arrow Size| Large'**

Set the normal arrow size to large in the **Viewport**.

**'View| Display Options| Nodes'****'View| Display Options| Nodes| Shape'****'View| Display Options| Nodes| Shape| Square'**

Display Nodes as squares in the **Viewport**.

**'View| Display Options| Nodes| Shape| Diamond'**

Display Nodes as diamonds in the **Viewport**.

**'View| Display Options| Nodes| Shape| Circle'**

Display Nodes as circles in the **Viewport**.



**'View| Display Options| Nodes| Size'****'View| Display Options| Nodes| Size| Small'**

Set the Node symbol size to small in the **Viewport**.

**'View| Display Options| Nodes| Size| Medium'**

Set the Node symbol size to medium in the **Viewport**.

**'View| Display Options| Nodes| Size| Large'**

Set the Node symbol size to large in the **Viewport**.

**'View| Display Options| Elements'****'View| Display Options| Elements| Fill Elements'**

Toggle the display style for filling Elements.

**'View| Display Options| Elements| Show Internal Faces'**

Toggle the display style for showing the internal faces of Elements. When off, this can speed the plotting of solid models as all faces marked as internal are skipped.

**'View| Display Options| Elements| Show Edges'**

Toggle the display style for showing the edges of Elements.

**'View| Display Options| Elements| Show Normals'**

Toggle the display style for showing the Element normals.

**'View| Display Options| Elements| Normal Arrow Size'****'View| Display Options| Elements| Normal Arrow Size| Small'**

Set the normal arrow symbol size to small in the **Viewport**.

**'View| Display Options| Elements| Normal Arrow Size| Medium'**

Set the normal arrow size to medium in the **Viewport**.

**'View| Display Options| Elements| Normal Arrow Size| Large'**

Set the normal arrow size to large in the **Viewport**.

**'View| Display Options| Displacement Constraints'****'View| Display Options| Displacement Constraints| Size'****'View| Display Options| Displacement Constraints| Size| Small'**

Set the Displacement Constraint symbol size to small in the **Viewport**.

**'View| Display Options| Displacement Constraints| Size| Medium'**

Set the Displacement Constraint symbol size to medium in the **Viewport**.

**'View| Display Options| Displacement Constraints| Size| Large'**

Set the Displacement Constraint symbol size to large in the **Viewport**.

**'View| Display Options| Displacement Constraints| Highlight Non Zero Values'**

Toggles whether non zero values are highlighted. This can be useful to visually indicate which are not the most common zero values.

**'View| Display Options| Pressures'****'View| Display Options| Pressures| Arrow Size'****'View| Display Options| Pressures| Arrow Size| Small'**

Set the arrow symbol size to small in the **Viewport**.

**'View| Display Options| Pressures| Normal Arrow Size| Medium'**

Set the arrow symbol size to medium in the **Viewport**.

**'View| Display Options| Pressures| Arrow Size| Large'**

Set the arrow symbol size to large in the **Viewport**.

**'View| Display Options| Nodal Pressures'****'View| Display Options| Nodal Pressures| Arrow Size'****'View| Display Options| Nodal Pressures| Arrow Size| Small'**

Set the arrow symbol size to small in the **Viewport**.

**'View| Display Options| Nodal Pressures| Normal Arrow Size| Medium'**

Set the arrow symbol size to medium in the **Viewport**.

**'View| Display Options| Nodal Pressures| Arrow Size| Large'**

Set the arrow symbol size to large in the **Viewport**.

**'View| Display Options| Forces'****'View| Display Options| Forces| Arrow Size'****'View| Display Options| Forces| Arrow Size| Small'**

Set the arrow symbol size to small in the **Viewport**.

**'View| Display Options| Forces| Normal Arrow Size| Medium'**

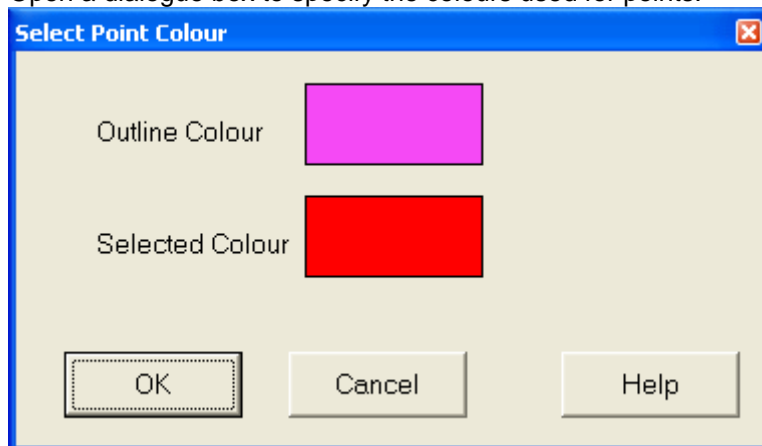
Set the arrow symbol size to medium in the **Viewport**.

**'View| Display Options| Forces| Arrow Size| Large'**

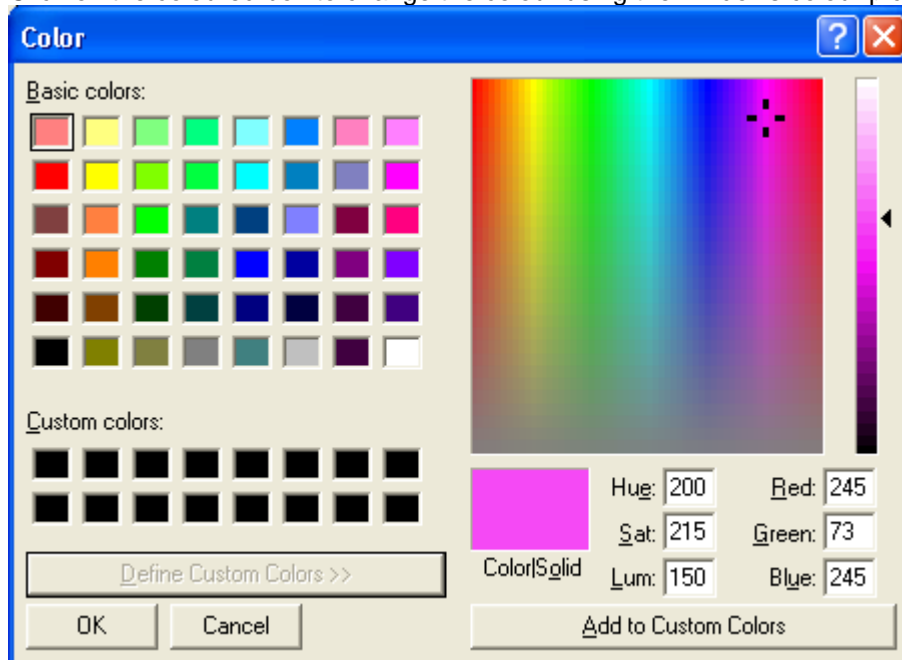
Set the arrow symbol size to large in the **Viewport**.

**'View| Colours'****'View| Colours| Points'**

Open a dialogue box to specify the colours used for points.



Click on the coloured box to change the colour using the windows colour picker dialogue.

**'View| Colours| Lines'**

Open a dialogue box to specify the colours used for lines.

**'View| Colours| Surfaces'****'View| Colours| Surfaces| Depth Shaded'**

Set the colours of surfaces to be depth shaded.

**'View| Colours| Surfaces| Infinite Light Shaded'**

Set the colours of surfaces to be infinite light shaded.

**'View| Colours| Surfaces| Colour by ID number'**

Set the colours of surfaces to be coloured by ID number.

**'View| Colours| Surfaces| Surface Colour'**

Open a dialogue box to specify the colours used for surfaces.

**'View| Colours| Surfaces| Selected Colour'**

Open a dialogue box to specify the colours used for surfaces that are selected.

**'View| Colours| Surfaces| Surface Normal Colour'**

Open a dialogue box to specify the colours used for surface normal arrows.

**'View| Colours| Nodes'**

Open a dialogue box to specify the colours used for nodes.

**'View| Colours| Elements'****'View| Colours| Elements| Depth Shaded'**

Set the colours of Elements to be depth shaded.

**'View| Colours| Elements| Infinite Light Shaded'**

Set the colours of Elements to be infinite light shaded.

**'View| Colours| Elements| Colour by ID number'**

Set the colours of Elements to be coloured by ID number. The colours are intended to be *tasteful* rather than informative.

**'View| Colours| Elements| Colour by Material'**

Set the colours of Elements to be coloured by Material number. The Colour will be selected by taking the remainder of the material number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by Property'**

Set the colours of Elements to be coloured by Property number. The Colour will be selected by taking the remainder of the property number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by Type'**

Set the colours of Elements to be coloured by Type number. The Colour will be selected by taking the remainder of the type number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by NSRL'**

Set the colours of Elements to be coloured by NSRL number. The Colour will be selected by taking the remainder of the NSRL number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by NORDR'**

Set the colours of Elements to be coloured by Node Order number. The Colour will be selected by taking the remainder of the Node Order number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by Formulation'**

Set the colours of Elements to be coloured by Formulation number. The Colour will be selected by taking the remainder of the formulation number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by Face Number'**

Set the colours of Elements to be coloured by Face number. The Colour for each face will be selected by taking the remainder of the face number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

**'View| Colours| Elements| Colour by Layer'**

Set the colours of Elements to be coloured by Layer number. The Colour will be selected by taking the remainder of the layer number when divided by 16, and looking up the colour from the 16 specified in **'View| Colours| Colour List'**.

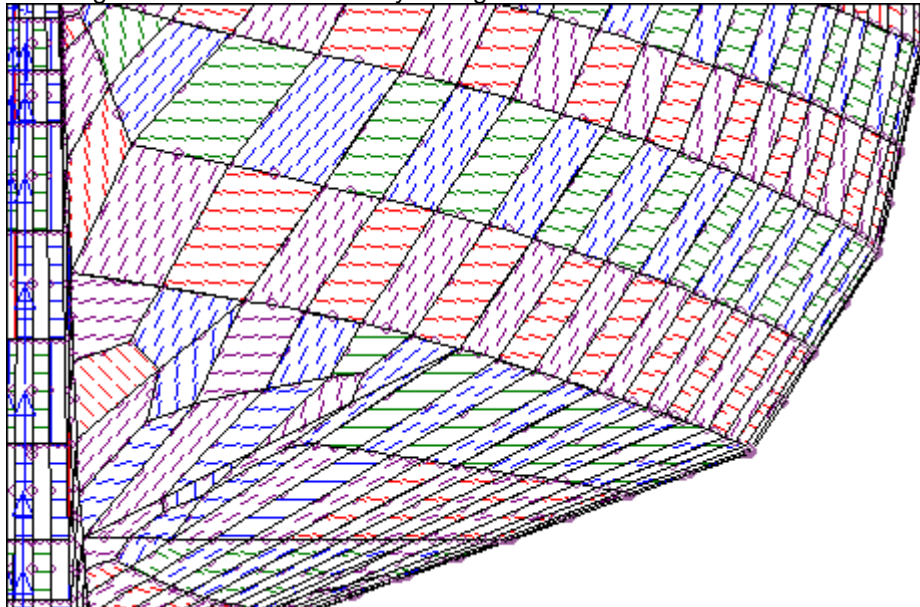
**'View| Colours| Elements| Show Material X Direction'**

Set the display type to show the Material X direction for shell elements and face 1 of solid elements only. This option is particularly useful for composite elements or elements with orthotropic materials, as the element face is shaded with the direction of the material X direction. In addition, the shading is coloured dependant on the direction displayed on the screen to further indicate the direction. The colours are as follows:-

Blue	0°	to	90°
Green	91°	to	180°
Purple	181°	to	269°
Red	270°	to	359°

When viewing the element, 0° is horizontal to the right, with positive angle anti clockwise. The resolution of the X direction angle is 14°, so the angle shown must be taken as a guide rather than as an exact orientation.

The picture below is an example of a model with poor material X direction definitions. Due to the multitude of different directions, an isotropic material *has* to be used otherwise the resulting stresses will be seriously wrong.

**'View| Colours| Elements| Colour by Group'**

Set the colours of Elements to be coloured by Group number. The Colour will be selected by taking the remainder of the group number when divided by 16, and looking up the colour from the 16 specified in 'View| Colours| Colour List'.

**'View| Colours| Elements| Element Colour'**

Open a dialogue box to specify the colours used for Elements.

**'View| Colours| Elements| Selected Colour'**

Open a dialogue box to specify the colours used for Elements that are selected.

**'View| Colours| Elements| Element Normal Colour'**

Open a dialogue box to specify the colours used for Element normal arrows.

**'View| Colours| Rigid Links'**

Open a dialogue box to specify the colours used for Rigid Links.

**'View| Colours| Displacement Constraints'**

Open a dialogue box to specify the colours used for Displacement Constraints.

**'View| Colours| Pressures'**

Open a dialogue box to specify the colours used for Pressures.

**'View| Colours| Nodal Pressures'**

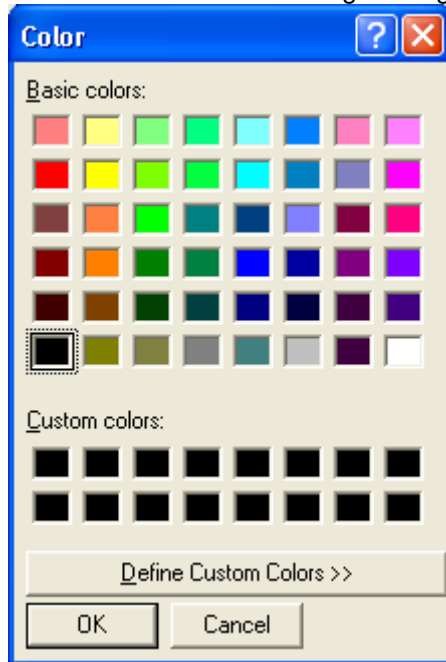
Open a dialogue box to specify the colours used for Nodal Pressures.

**'View| Colours| Forces'**

Open a dialogue box to specify the colours used for Forces.

**'View| Colours| Image Border'**

Open a colour picker dialogue box to specify the colour used for the border around the image. Make it the same as the image background to make it disappear.

**'View| Colours| Image Background'**

Open a dialogue box to specify the colour used for the image background.

**'View| Colours| Bounding Box'**

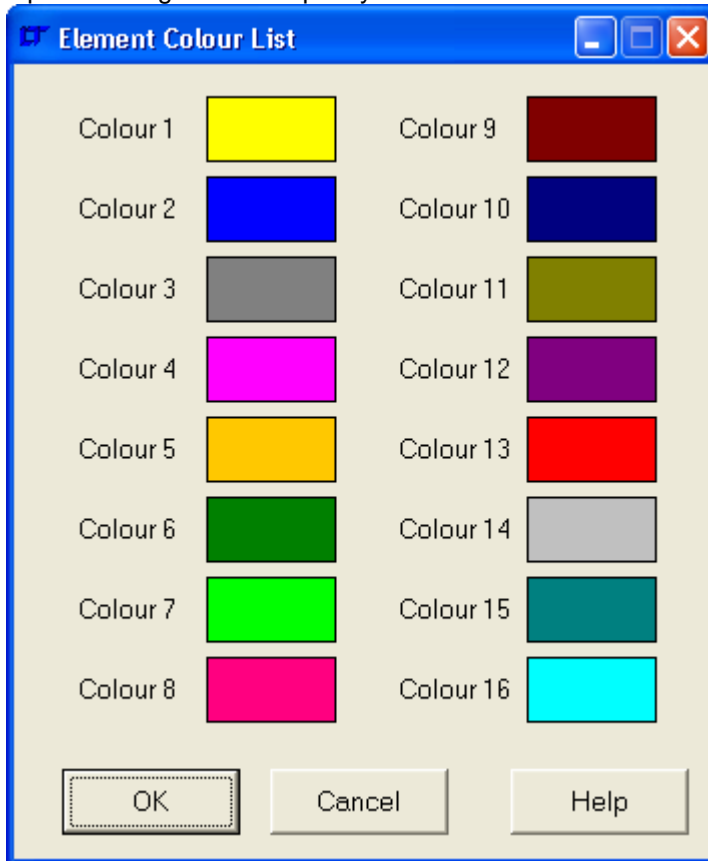
Open a dialogue box to specify the colour used for the bounding box that appears when plotting takes more than 0.1s.

**'View| Colours| Entity Snap'**

Open a dialogue box to specify the colours used for the snap marker when snapping to an entity.

**'View| Colours| Colour List'**

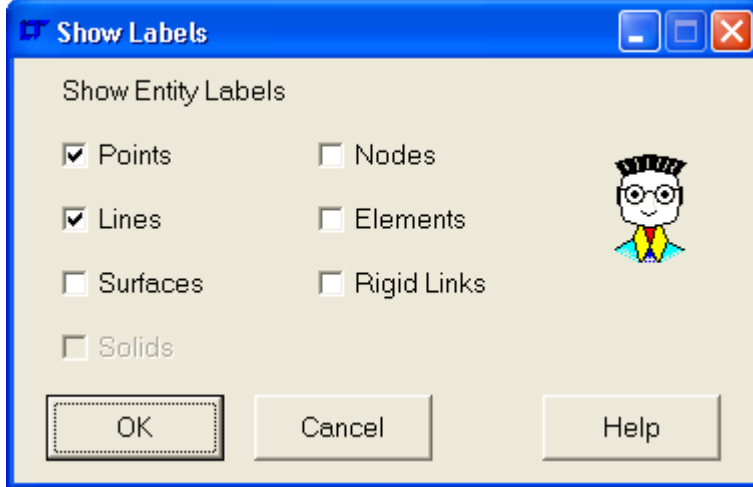
Open a dialogue box to specify the 16 colours used for various identification requirements.



Click on a colour to change it with a colour picker dialogue box.

**'View| Labels'****'View| Labels| All'**

Open a dialogue box to specify whether the various labels are toggled on or off.



The label is on when the checkbox is ticked.

**'View| Labels| Points'**

Toggle the display of labels for Points.

**'View| Labels| Lines'**

Toggle the display of labels for Lines.

**'View| Labels| Surfaces'**

Toggle the display of labels for Surfaces.

**'View| Labels| Nodes'**

Toggle the display of labels for Nodes.

**'View| Labels| Elements'**

Toggle the display of labels for Elements.

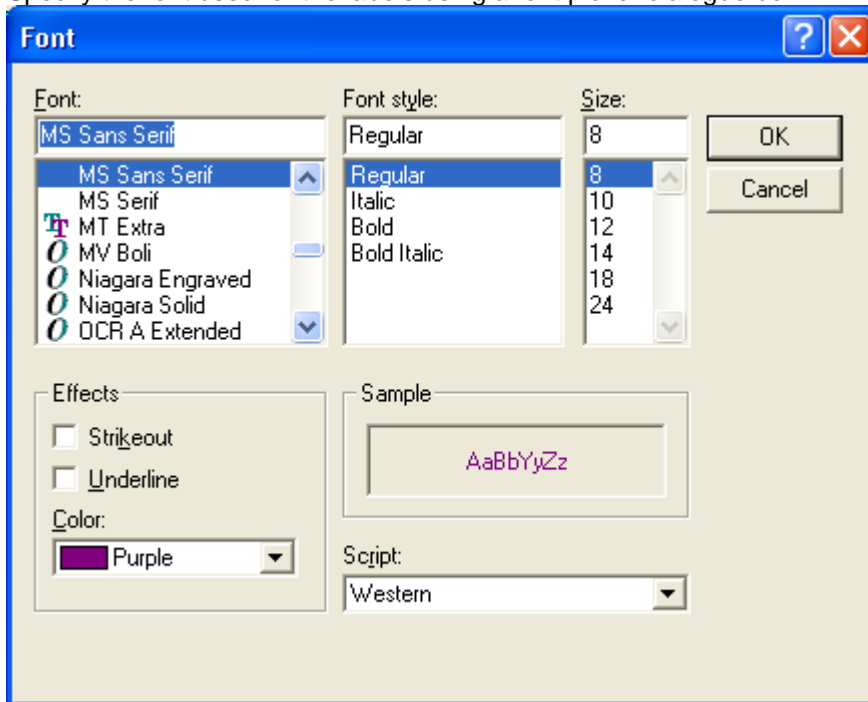
**'View| Labels| Rigid Links'**

Toggle the display of labels for Rigid Links.



**'View| Labels| Font'**

Specify the font used for the labels using a font picker dialogue box.

**'View| Light Source'****'View| Light Source| Top Left'**

When shading the model simulate the infinite light source as coming from the top left of the view.

**'View| Light Source| Top Centre'**

When shading the model simulate the infinite light source as coming from the top centre of the view.

**'View| Light Source| Top Right'**

When shading the model simulate the infinite light source as coming from the top right of the view.

**'View| Light Source| Centre Left'**

When shading the model simulate the infinite light source as coming from the centre left of the view.

**'View| Light Source| Centre'**

When shading the model simulate the infinite light source as coming from the centre of the view.

**'View| Light Source| Centre Right'**

When shading the model simulate the infinite light source as coming from the centre right of the view.

**'View| Light Source| Bottom Left'**

When shading the model simulate the infinite light source as coming from the bottom left of the view.

**'View| Light Source| Bottom Centre'**

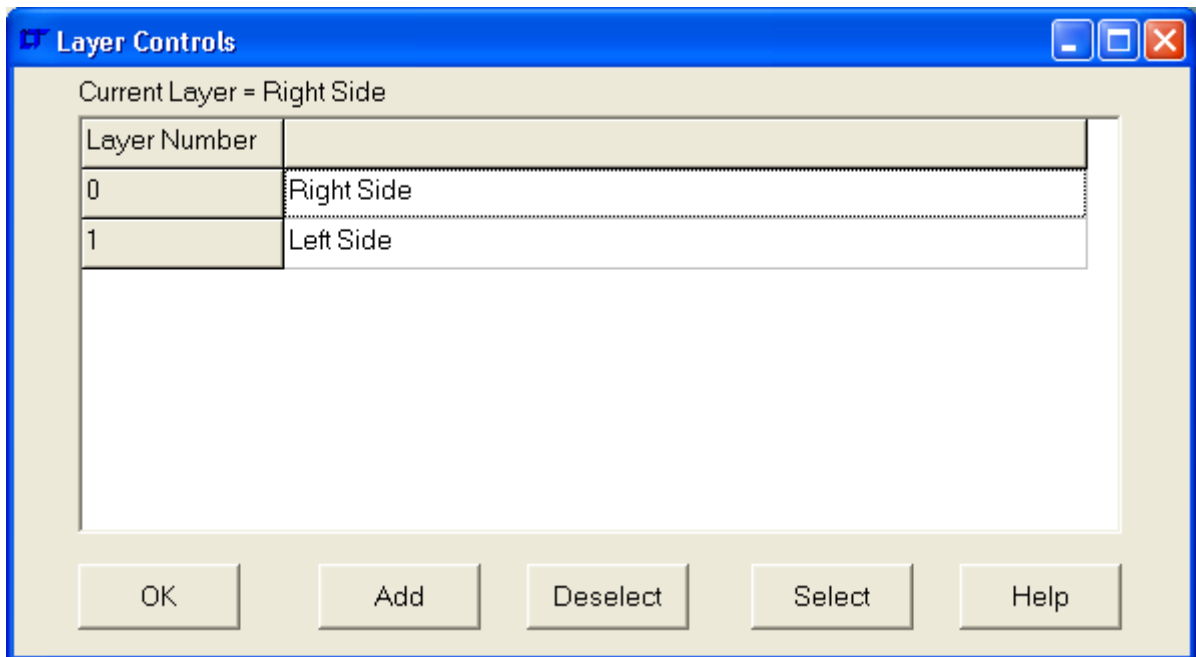
When shading the model simulate the infinite light source as coming from the bottom centre of the view.

**'View| Light Source| Bottom Right'**

When shading the model simulate the infinite light source as coming from the bottom right of the view.

**'View| Layers'****'View| Layers | Layer Controls'**

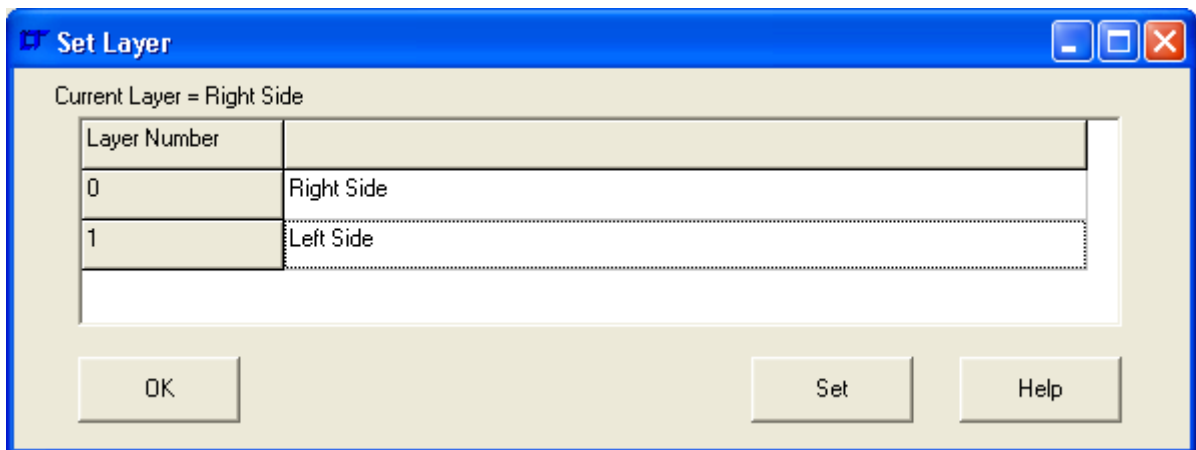
Opens a dialogue box to control, add, delete, select layers. The dialogue box is as below:-



Double click the layer name to make it the current layer, right click to rename the layer.

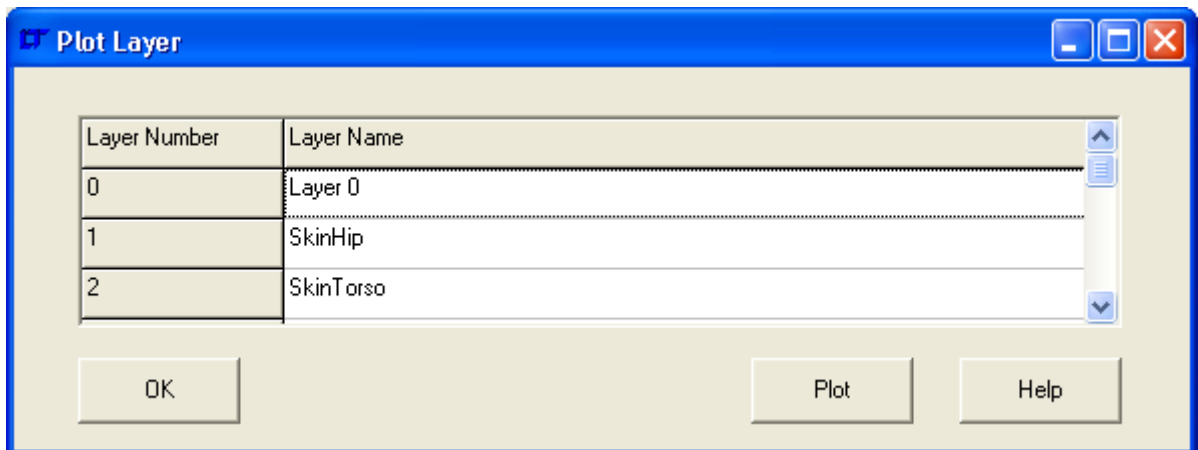
**'View| Layers | Set Current'**

Opens a dialogue box to set the current layer that new entities are added to.

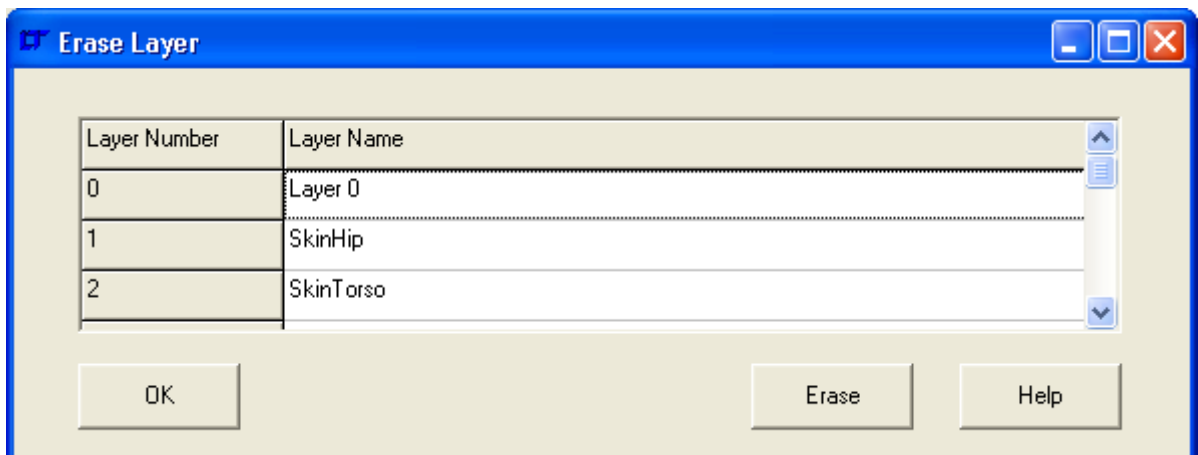


**'View| Layers | Plot by Layer Name'**

Opens a dialogue box to plot layers by their name.

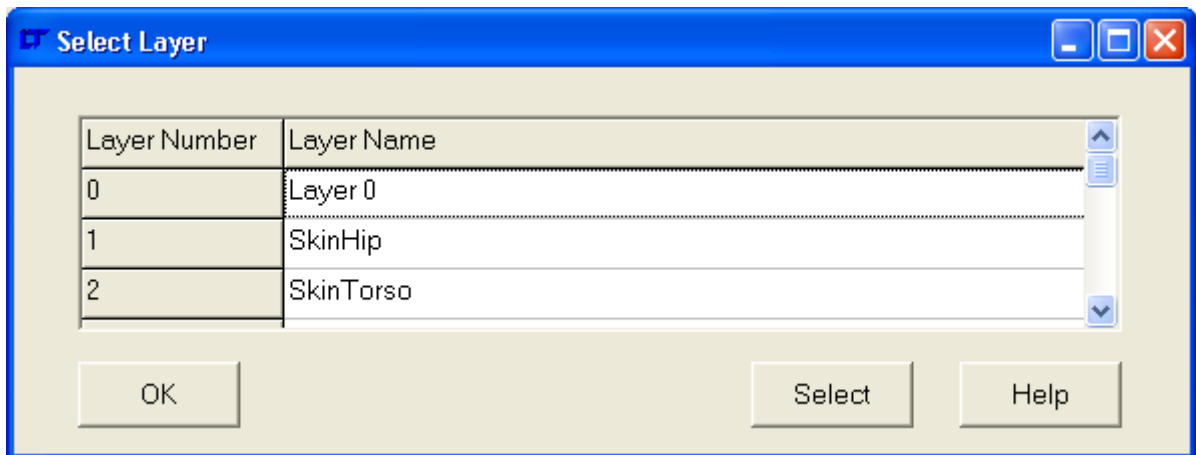
**'View| Layers | Erase by Layer Name'**

Opens a dialogue box to erase layers by their name.

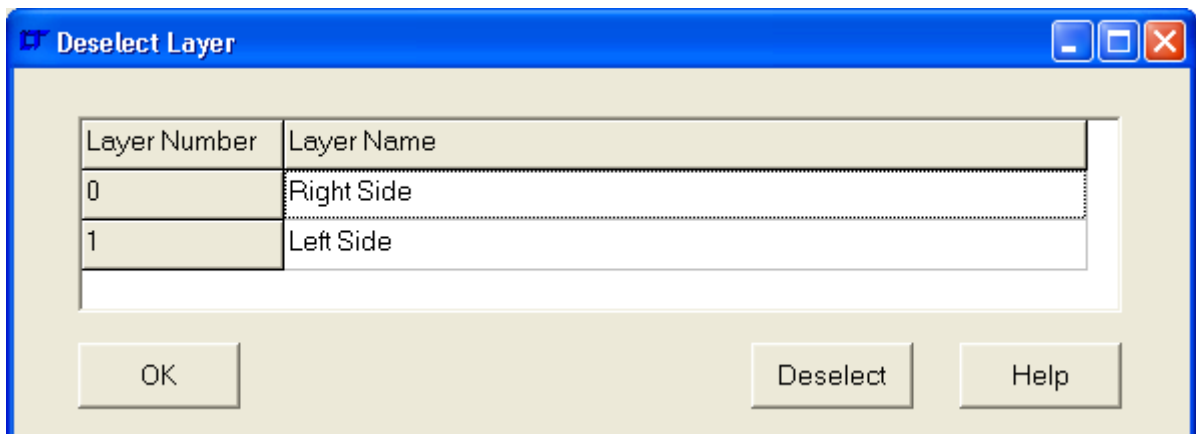


**'View| Layers | Select by Layer Name'**

Opens a dialogue box to select layers by their name.

**'View| Layers | Deselect by Layer Name'**

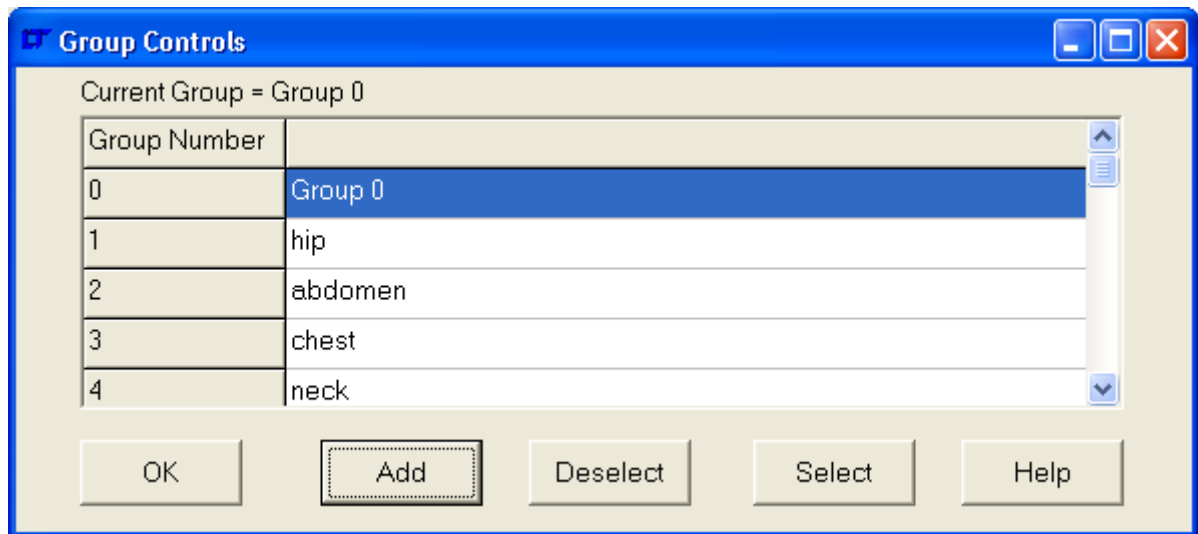
Opens a dialogue box to deselect layers by their name.

**'View| Layers | Add to Current Layer'**

Adds the currently selected entities to the current layer. All visible entity types except Boundary conditions (Displacement Constraints, Pressures, Nodal Pressures and Forces) can be added to layers.

**'View| Groups'****'View| Groups | Group Controls'**

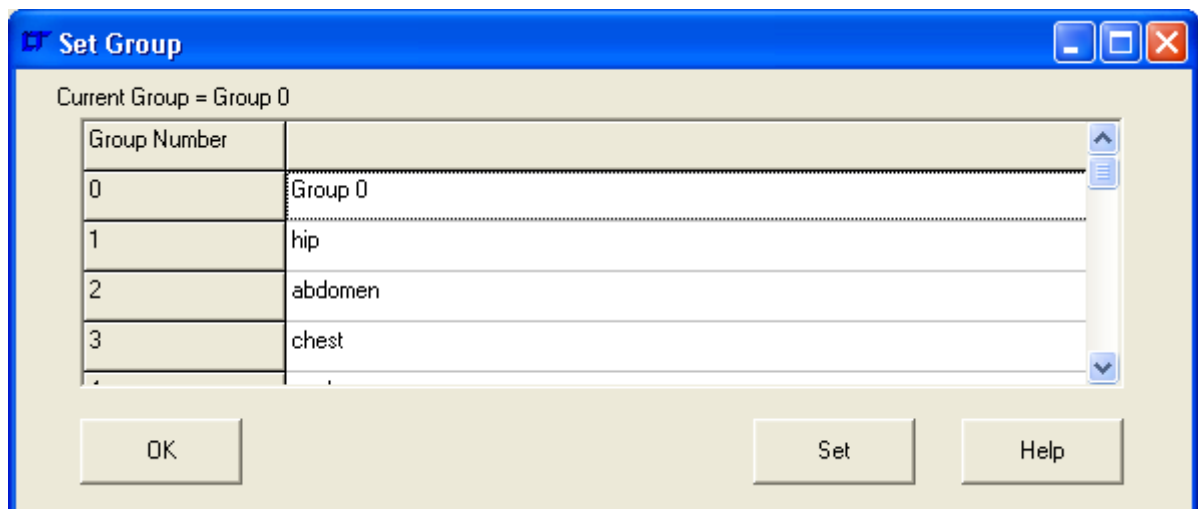
Opens a dialogue box to control, add, delete, select Groups. The dialogue box is as below:-



Double click the Group name to make it the current Group, right click to rename the Group.

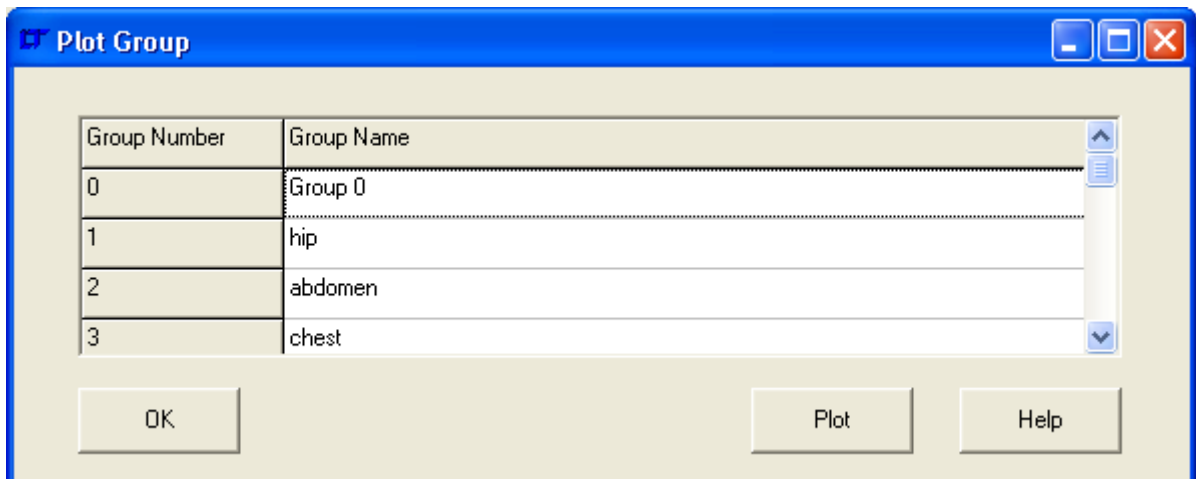
**'View| Groups | Set Current'**

Opens a dialogue box to set the current Group that new entities are added to.

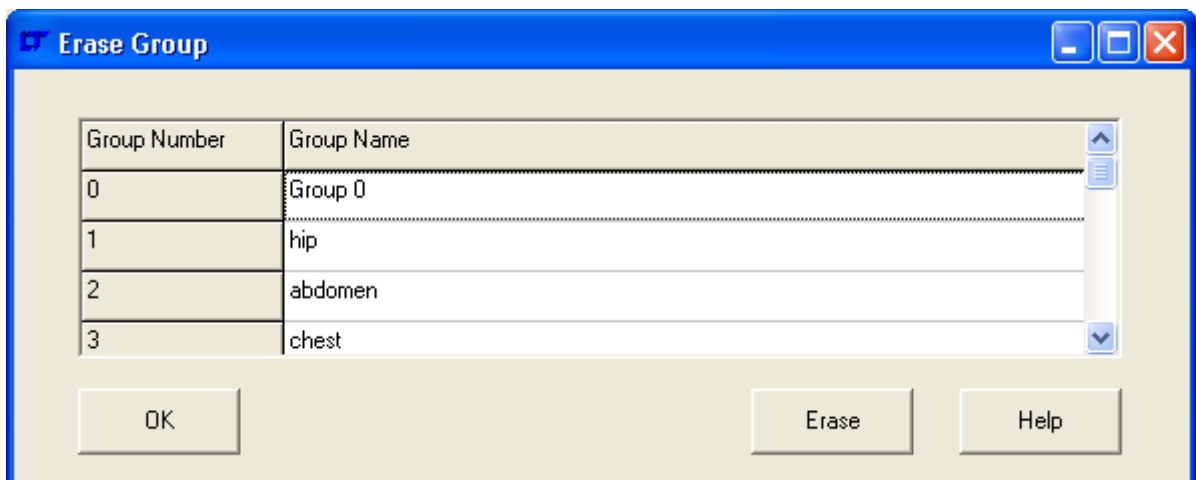


**'View| Groups | Plot by Group Name'**

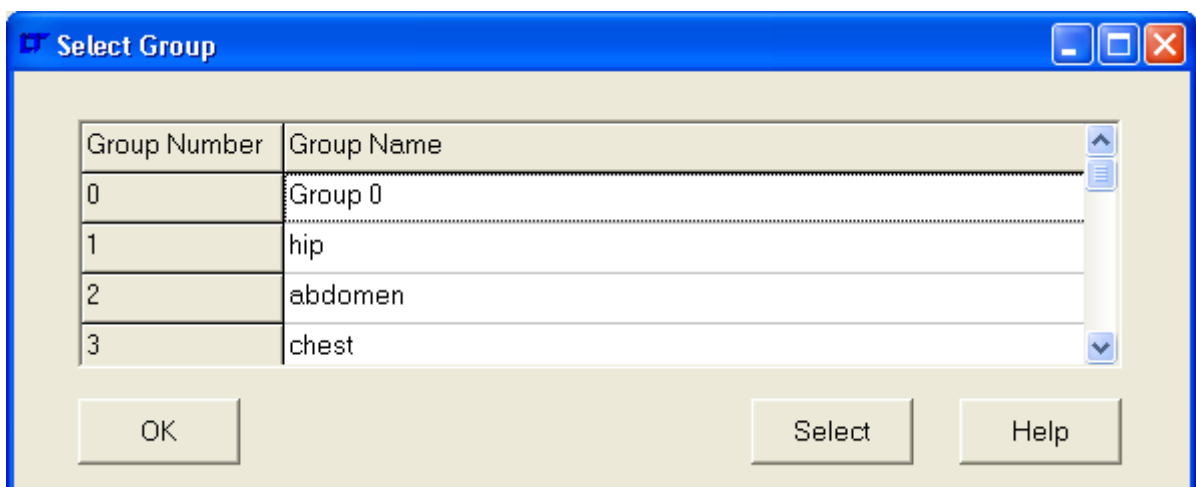
Opens a dialogue box to plot Groups by their name.

**'View| Groups | Erase by Group Name'**

Opens a dialogue box to erase Groups by their name.

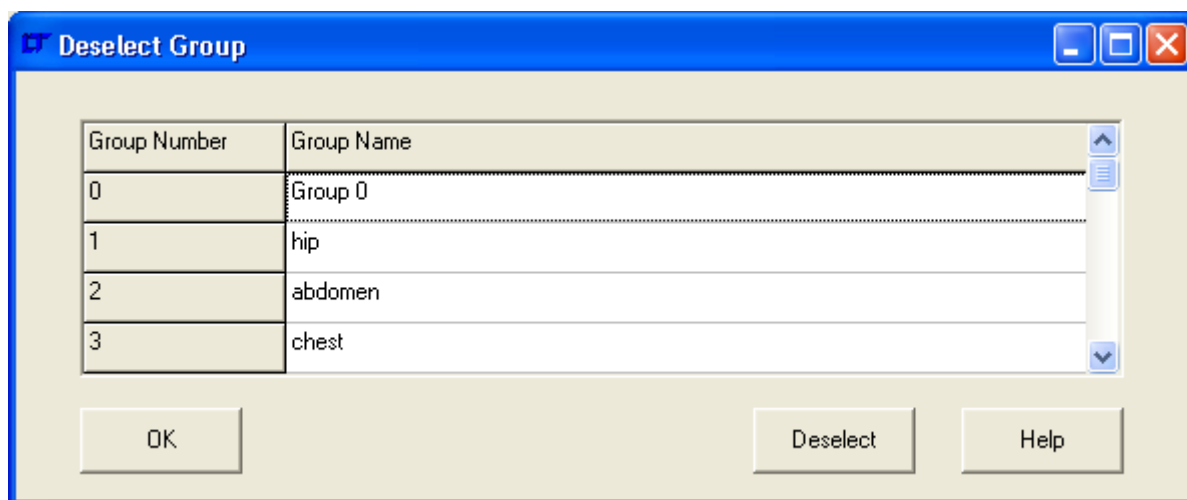
**'View| Groups | Select by Group Name'**

Opens a dialogue box to select Groups by their name.



**'View| Groups | Deselect by Group Name'**

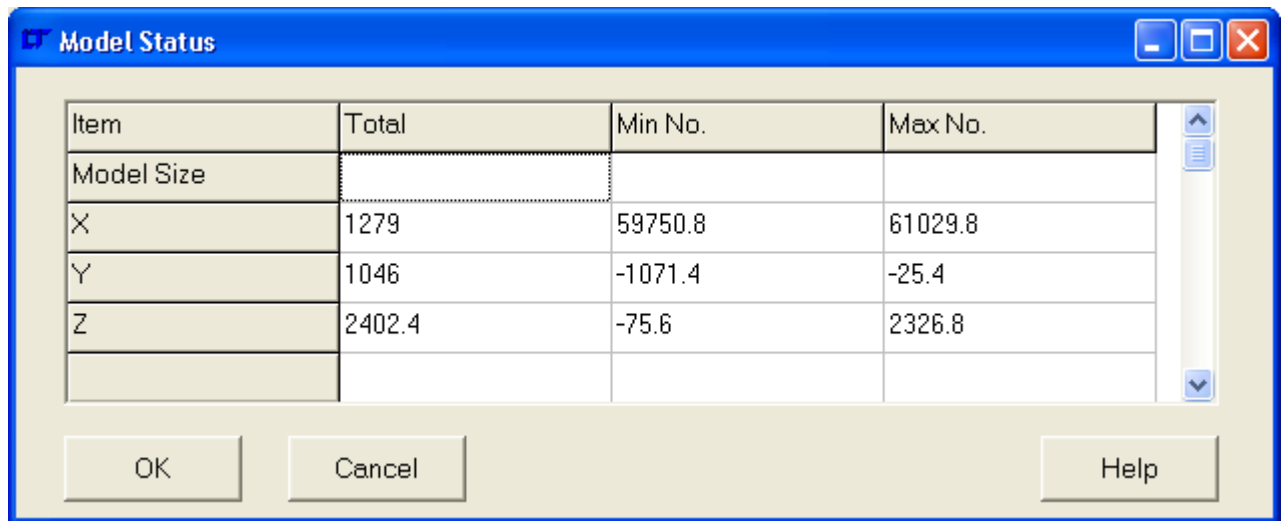
Opens a dialogue box to deselect Groups by their name.

**'View| Groups | Add to Current Group'**

Adds the currently selected elements to the current Group.

**'View| Status'**

Opens a dialogue window to dynamically inspect information about the model.



The dialogue data may be copied to the clipboard by right clicking over the data grid and selecting pop-up menu option **'Copy to Clipboard'**. The data is Tab separated on the clipboard.



### 3.2.4 Selection Menu

The following menu commands may be found in the selection menu.

**'Selection| All'**

Select all Points, Lines, Surfaces, Solids, Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'Selection| All Geometry'**

Select all Points, Lines, Surfaces and Solids.

**'Selection| All FE Model'**

Select all Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'Selection| All Boundary Conditions'**

**'Selection| All Boundary Conditions| All'**

Select all Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'Selection| All Boundary Conditions| Range by Case Number'**

Select all Displacement Constraints, Pressures, Nodal Pressures and Forces in a range of load cases. Typed input of the range is necessary in the command line.

**'Selection| None'**

Deselect all Points, Lines, Surfaces, Solids, Nodes, Elements, Rigid Links, Displacement Constraints, Pressures, Nodal Pressures and Forces.

**'Selection| Points'**

**'Selection| Points| All'**

Select all Points.

**'Selection| Points| None'**

Deselect all Points.

**'Selection| Points| Range'**

Select a range of points. Typed input of the range is necessary in the command line.

**'Selection| Points| Invert Selection'**

Invert the selection of points. All points that are initially invisible are made visible as they are selected.

**'Selection| Points| Selected Lines'**

Select points by association with selected lines.

**'Selection| Points| Selected Surfaces'**

Select points by association with selected surfaces.

**'Selection| Points| Selected Solids'**

Select points by association with selected solids.

**'Selection| Lines'**

**'Selection| Lines| All'**

Select all Lines.

**'Selection| Lines| None'**

Deselect all Lines.

**'Selection| Lines| Range'**

Select a range of Lines. Typed input of the range is necessary in the command line.

**'Selection| Lines| Invert Selection'**

Invert the selection of Lines. All Lines that are initially invisible are made visible as they are selected.

**'Selection| Lines| Selected Points'**

Select Lines by association with selected points.

**'Selection| Lines| Selected Surfaces'**

Select Lines by association with selected surfaces.

**'Selection| Lines| Selected Solids'**

Select Lines by association with selected solids.

**'Selection| Surfaces'****'Selection| Surfaces| All'**

Select all Surfaces.

**'Selection| Surfaces| None'**

Deselect all Surfaces.

**'Selection| Surfaces| Range'**

Select a range of Surfaces. Typed input of the range is necessary in the command Surface.

**'Selection| Surfaces| Invert Selection'**

Invert the selection of Surfaces. All Surfaces that are initially invisible are made visible as they are selected.

**'Selection| Surfaces| Selected Points'**

Select Surfaces by association with selected points.

**'Selection| Surfaces| Selected Lines'**

Select Surfaces by association with selected lines.

**'Selection| Surfaces| Selected Solids'**

Select Surfaces by association with selected solids.

**'Selection| Nodes'****'Selection| Nodes| All'**

Select all Nodes.

**'Selection| Nodes| None'**

Deselect all Nodes.

**'Selection| Nodes| Range'**

Select a range of Nodes. Typed input of the range is necessary in the command line.

**'Selection| Nodes| Invert Selection'**

Invert the selection of Nodes. All Nodes that are initially invisible are made visible as they are selected.

**'Selection| Nodes| Selected Elements'**

Select Nodes by association with selected Elements.

**'Selection| Nodes| Selected Rigid Links'**

Select Nodes by association with selected Rigid Links.

**'Selection| Nodes| Selected Displacements'**

Select Nodes by association with selected Displacement Constraints.

**'Selection| Nodes| Selected Pressures'**

Select Nodes by association with selected Pressures.

**'Selection| Nodes| Selected Nodal Pressures'**

Select Nodes by association with selected Nodal Pressures.

**'Selection| Nodes| Selected Forces'**

Select Nodes by association with selected Forces.

**'Selection| Elements'****'Selection| Elements| All'**

Select all Elements.

**'Selection| Elements| None'**

Deselect all Elements.

**'Selection| Elements| Range'**

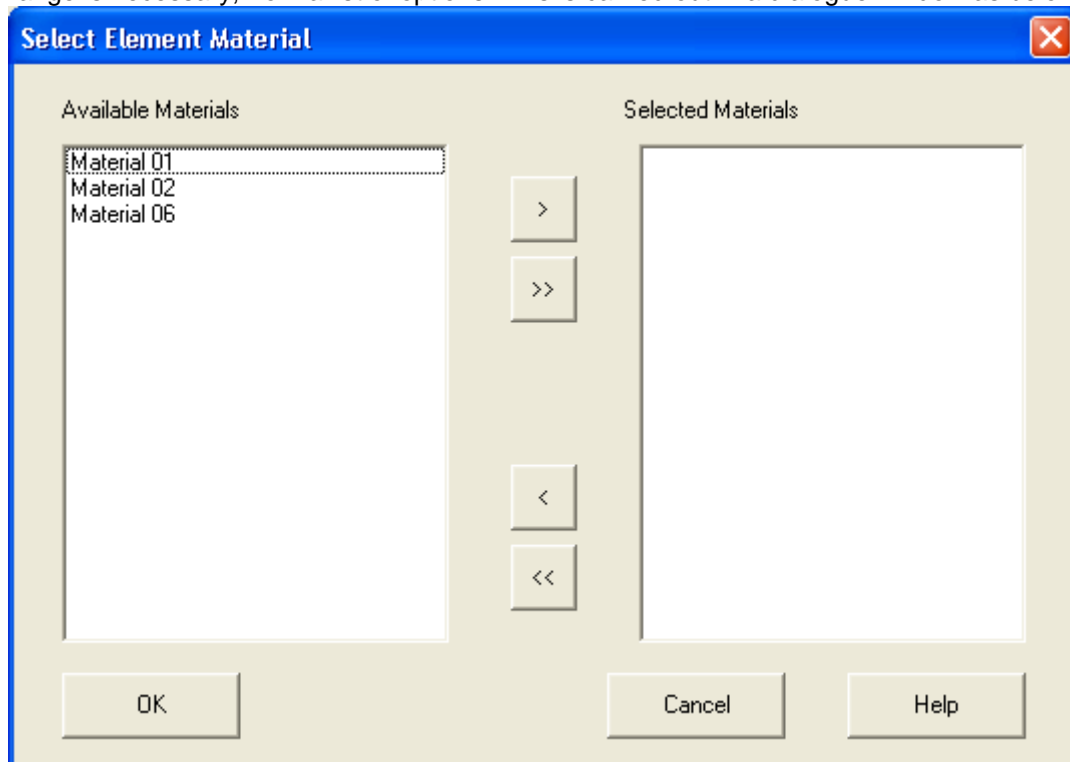
Select a range of Elements. Typed input of the range is necessary in the command line.

**'Selection| Elements| Invert Selection'**

Invert the selection of Elements. All Elements that are initially invisible are made visible as they are selected.

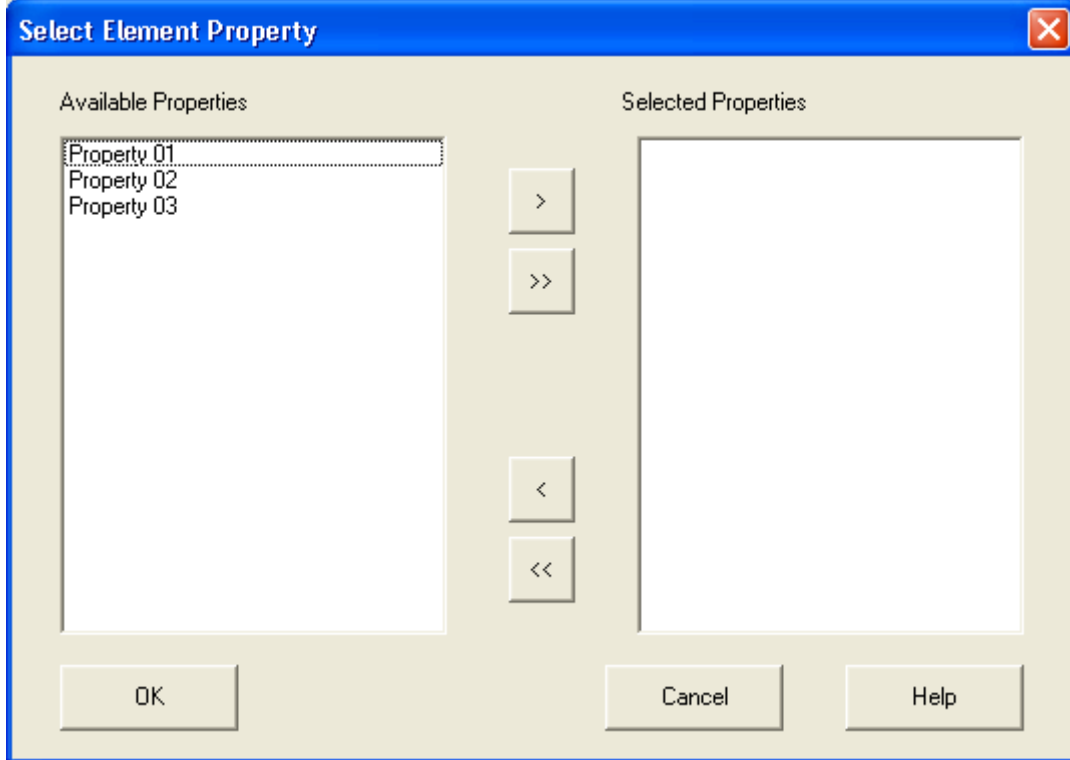
**'Selection| Elements| by Material'**

Select Elements dependant on a range of material numbers. Selection of the element material range is necessary, from a list of options. This is carried out in a dialogue window as below:-



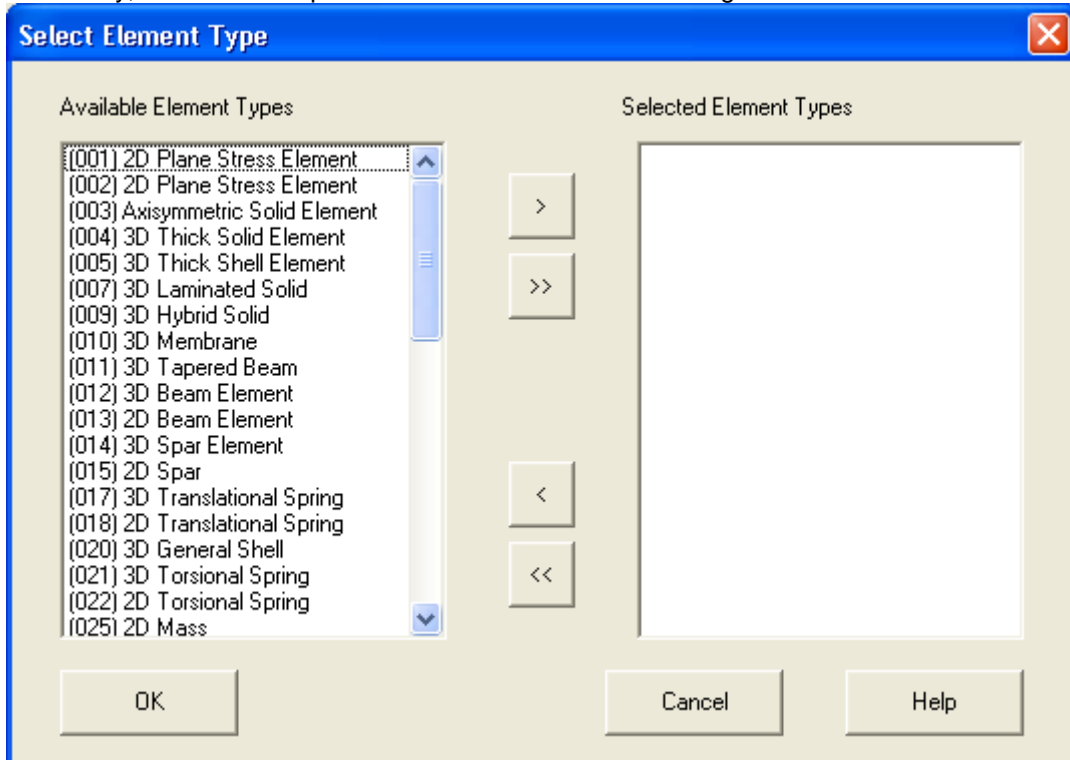
**'Selection| Elements| by Property'**

Select Elements dependant on a range of property numbers. Selection of the element property range is necessary, from a list of options. This is carried out in a dialogue window as below:-



**'Selection | Elements| by Element Type'**

Select Elements with element type number in range. Selection of the element type range is necessary, from a list of options. This is carried out in a dialogue window as below:-



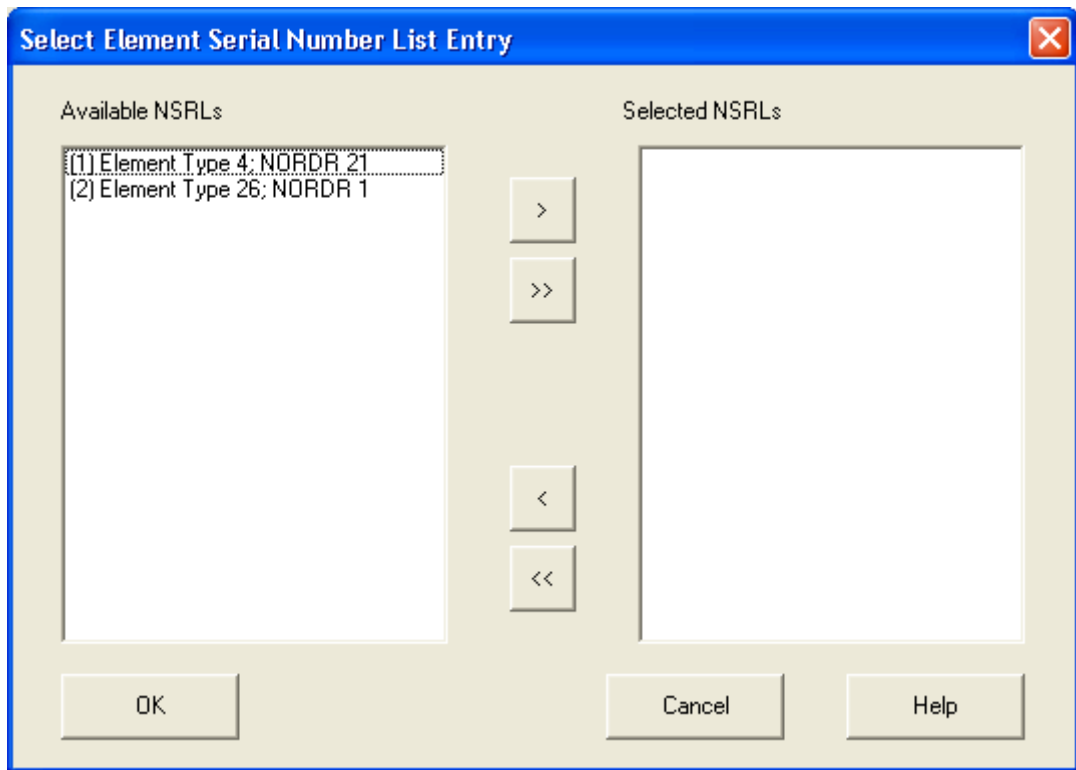
The element type number is from the list below:-

Type	Description
1	2D Plane Stress Element

2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam
40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

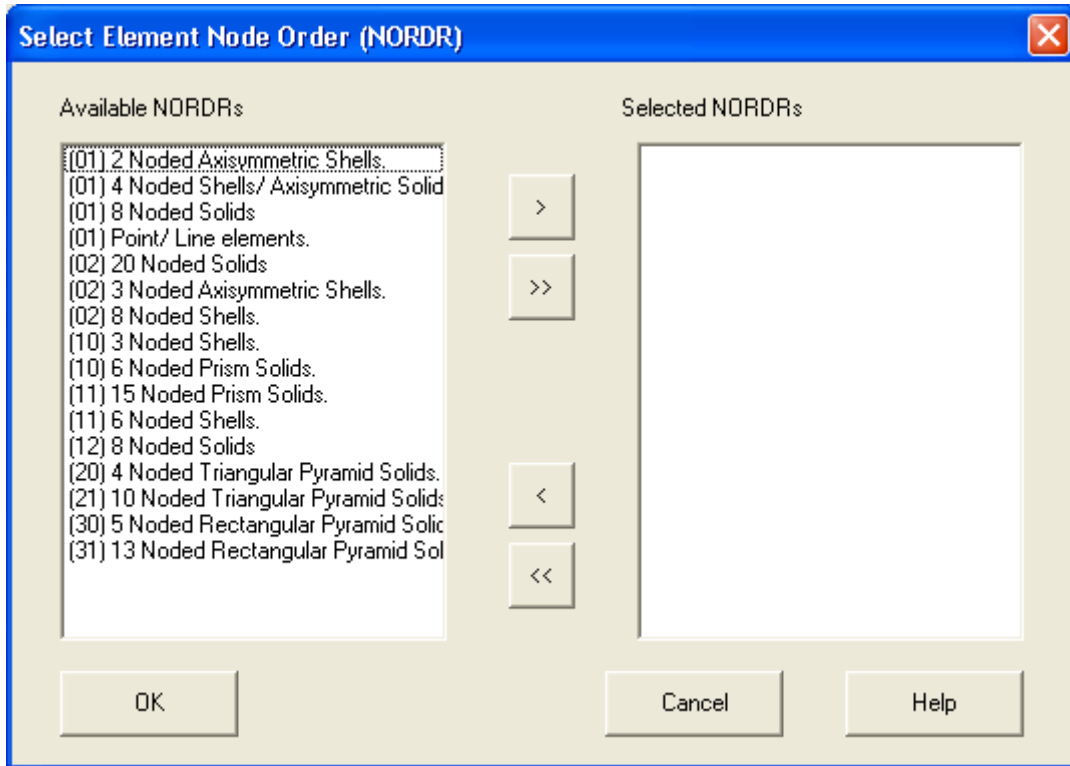
**'Selection| Elements| by NSRL'**

Select Elements with NSRL number in range. The NSRL of the elements to be erased is selected from a list of Serial Numbers generated for the current model. The selection list may be similar to the list below:-



**'Selection| Elements| by NORDR'**

Select Elements with Node Order number in range. Selection of the Node Order is from a list as below:-

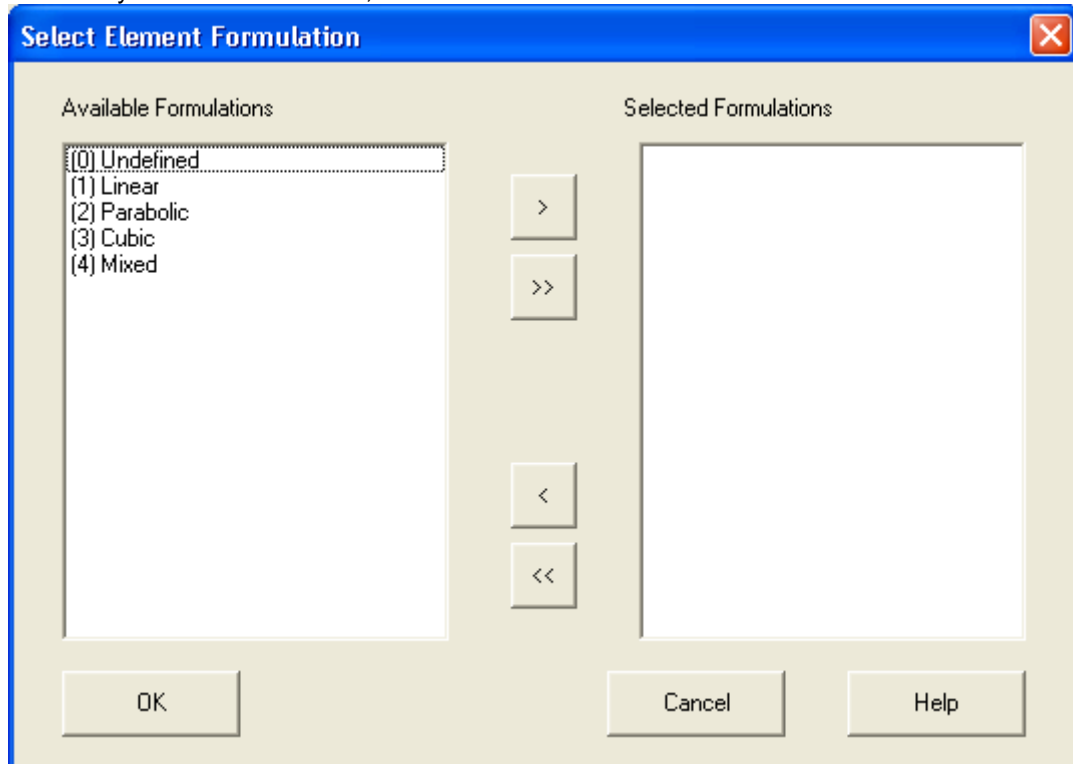


The Node Order is the node order of the element, and is defined as below:-

Node Order	Description
1	For all point or line elements
1	For all 2 noded axisymmetric shell elements
1	For all 4 noded shell elements/ 4 noded axisymmetric solid elements
1 or 12	For all 8 noded solid elements
2	For all 3 noded axisymmetric elements
2	For all 8 noded shell elements
2	For all 20 noded brick elements
10	For 3 Noded triangular shell elements
10	6 noded prism shaped brick elements
11	6 Noded triangular shell elements
11	15 noded prism brick elements
20	4 noded triangular pyramid solid elements
21	10 noded triangular pyramid solid elements
30	5 noded rectangular pyramid elements
31	13 noded rectangular pyramid elements

**'Selection| Elements| by Formulation'**

Select Elements with formulation number in range. Selection of the formulation range is necessary from a selection list, as below:-



Formulation is as below:-

Formulation	Description
1	Linear
2	Parabolic
3	Cubic
4	Mixed

**'Selection| Elements| Selected Nodes'**

Select Elements by association with selected Nodes.

**'Selection| Elements| Selected Rigid Links'**

Select Elements by association with selected Rigid Links.

**'Selection| Elements| Select by Group Name'**

Opens a dialogue window to choose the element group name to select.

**'Selection| Elements| Deselect by Group Name'**

Opens a dialogue window to choose the element group name to deselect.

**'Selection| Rigid Links'****'Selection| Rigid Links| All'**

Select all Rigid Links.

**'Selection| Rigid Links| None'**

Deselect all Rigid Links.

**'Selection| Rigid Links| Range'**

Select a range of Rigid Links. Typed input of the range is necessary in the command line.



**'Selection| Rigid Links| Invert Selection'**

Invert the selection of Rigid Links. All Rigid Links that are initially invisible are made visible as they are selected.

**'Selection| Rigid Links| Selected Nodes'**

Select Rigid Links by association with selected Nodes.

**'Selection| Rigid Links| Selected Elements'**

Select Rigid Links by association with selected Elements.

**'Selection| Displacement Constraints'****'Selection| Displacement Constraints| All'**

Select all Displacement Constraints.

**'Selection| Displacement Constraints| None'**

Deselect all Displacement Constraints.

**'Selection| Displacement Constraints| Invert Selection'**

Invert the selection of Displacement Constraints. All Displacement Constraints that are initially invisible are made visible as they are selected.

**'Selection| Displacement Constraints| by Load Case'**

Select all displacement constraints within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Displacement Constraints| by Node Number'**

Select all displacement constraints within a range of Node Numbers. Typed input of the range is necessary in the command line.

**'Selection| Displacement Constraints| Selected Nodes'**

Select all displacement constraints that are visible associated with the selected nodes.

**'Selection| Displacement Constraints| Selected Elements'**

Select all displacement constraints that are visible associated with the selected elements.

**'Selection| Pressures'****'Selection| Pressures| All'**

Select all Pressures.

**'Selection| Pressures| None'**

Deselect all Pressures.

**'Selection| Pressures| Invert Selection'**

Invert the selection of Pressures. All Pressures that are initially invisible are made visible as they are selected.

**'Selection| Pressures| Range by Element Association'**

Select all Pressures associated with selected elements.

**'Selection| Pressures| Range by Face Association'**

Select Pressures on a face associated with selected elements. Typed input of the face number is necessary in the command line.

For shell elements the face numbers are:-

- 1 Bottom Face
- 2 Top Face

For solid elements the face numbers are 1 to 4, 5 or 6 depending on the element type.

**'Selection| Pressures| by Load Case'**

Select all Pressures within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Pressures| Selected Nodes'**

Select all Pressures that are visible associated with the selected nodes.

**'Selection| Pressures| Selected Elements'**

Select all Pressures that are visible associated with the selected elements.

**'Selection| Pressures| Zero Value'**

Select all Pressures that have a pressure of zero.

**'Selection| Nodal Pressures'****'Selection| Nodal Pressures| All'**

Select all Nodal Pressures.

**'Selection| Nodal Pressures| None'**

Deselect all Nodal Pressures.

**'Selection| Nodal Pressures| Invert Selection'**

Invert the selection of Nodal Pressures. All Nodal Pressures that are initially invisible are made visible as they are selected.

**'Selection| Nodal Pressures| by Load Case'**

Select all Nodal Pressures within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Nodal Pressures| by Node Number'**

Select all Nodal Pressures within a range of Node Numbers. Typed input of the range is necessary in the command line.

**'Selection| Nodal Pressures| Selected Nodes'**

Select all Nodal Pressures that are visible associated with the selected nodes.

**'Selection| Nodal Pressures| Selected Elements'**

Select all Nodal Pressures that are visible associated with the selected elements.

**'Selection| Nodal Pressures| Zero Value'**

Select all Nodal Pressures that have a load of zero.

**'Selection| Forces'****'Selection| Forces| All'**

Select all Forces.

**'Selection| Forces| None'**

Deselect all Forces.

**'Selection| Forces| Invert Selection'**

Invert the selection of Forces. All Forces that are initially invisible are made visible as they are selected.

**'Selection| Forces| by Load Case'**

Select all Forces within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Forces| by Node Number'**

Select all Forces within a range of Node Numbers. Typed input of the range is necessary in the command line.

**'Selection| Forces| Selected Nodes'**

Select all Forces that are visible associated with the selected nodes.

**'Selection| Forces| Selected Elements'**

Select all Forces that are visible associated with the selected elements.

**'Selection| Forces| Zero Value'**

Select all Forces that have a load of zero.

**'Selection| Layers'****'Selection| Layers| Select by Name'**

Opens a dialogue window to choose the layer name to select.

**'Selection| Layers| Deselect by Name'**

Opens a dialogue window to choose the layer name to deselect.

**'Selection| Deselect After Operation'**

Specifies whether entities are to be automatically deselected after an operation occurs.

For example:-

If elements are selected and copied when 'Deselect After Operation' is checked, the original elements will be deselected once they are copied. However, if 'Deselect After Operation' is not checked, the original elements remain selected, and can be copied again.

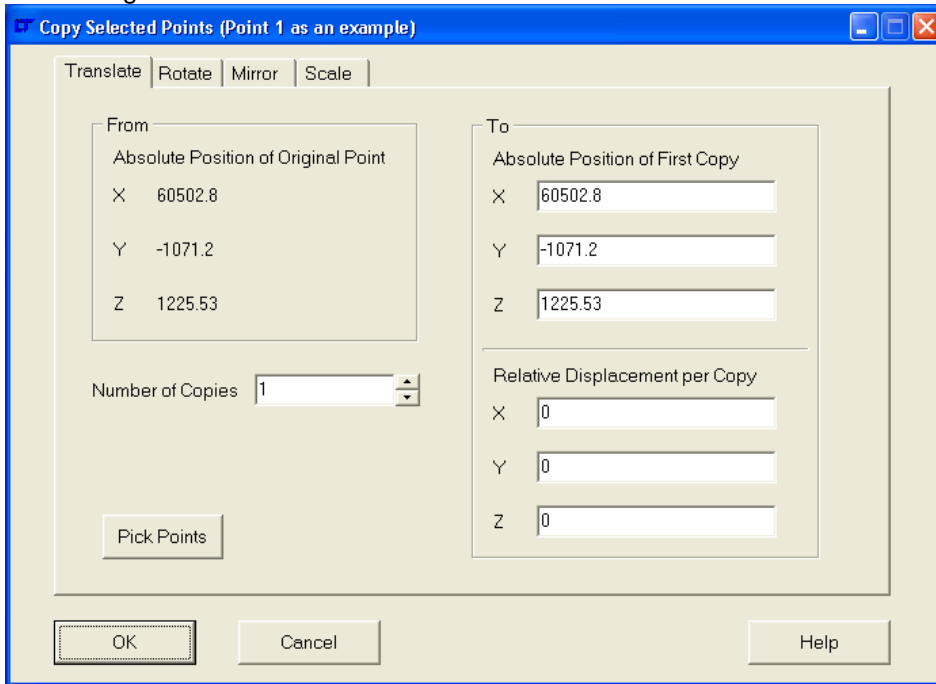
### 3.2.5 Geometry Menu

The following menu commands may be found in the geometry menu.

#### 'Geometry| Points'

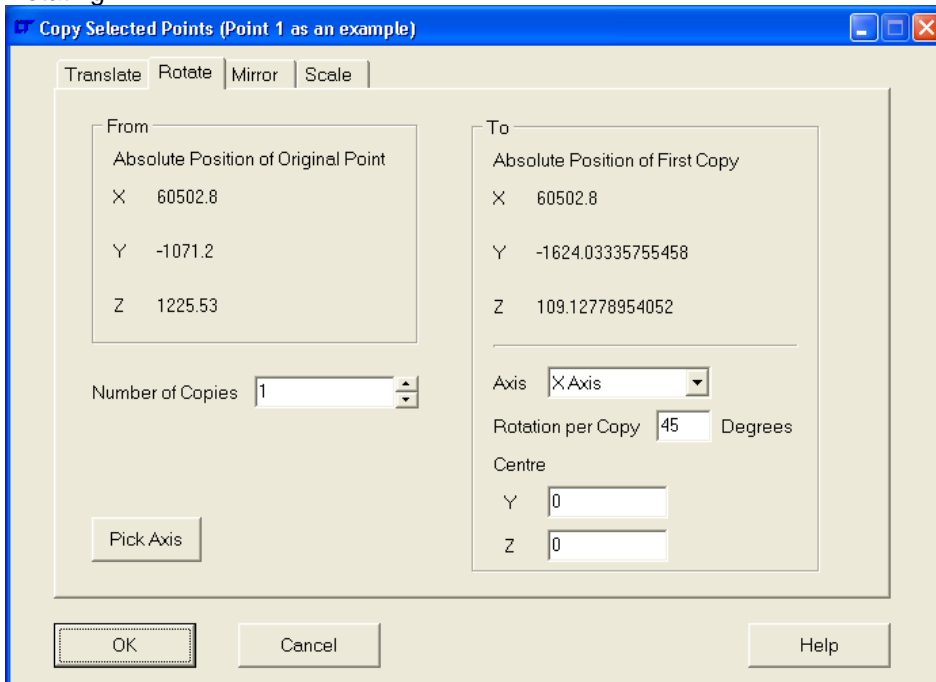
##### 'Geometry| Points| Copy'

Open a dialogue box to enter the data necessary to copy the currently selected points. Points may be translated, rotated, mirrored or scaled. The dialogue box is shown below:-  
Translating:-



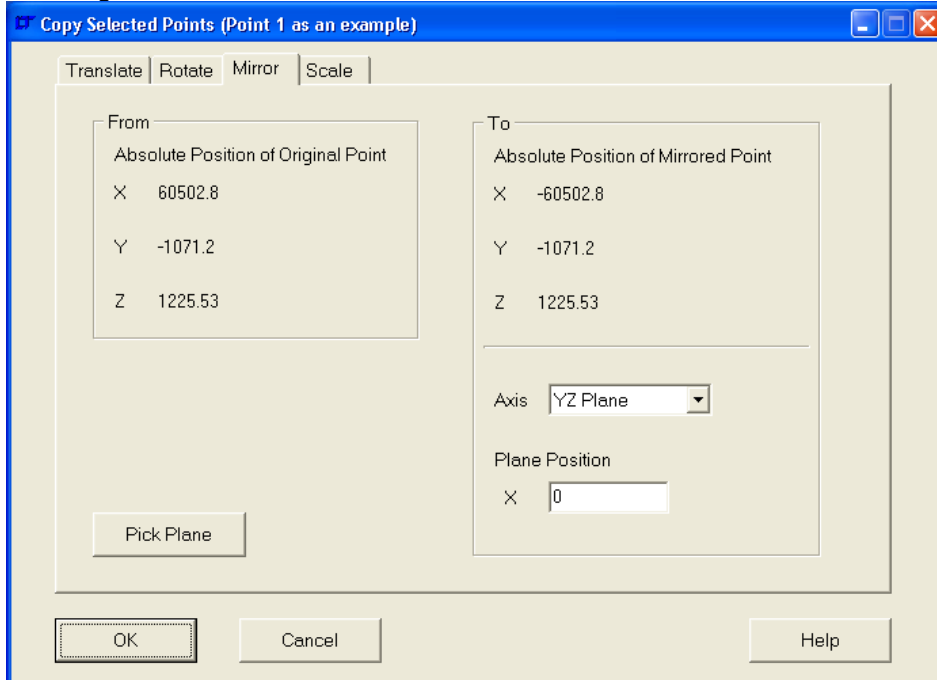
The 'Pick Points' button may be pressed to select a 'Copy From' Point and a 'Copy To' point. This will then be used as the base displacement per copy.

#### Rotating:-



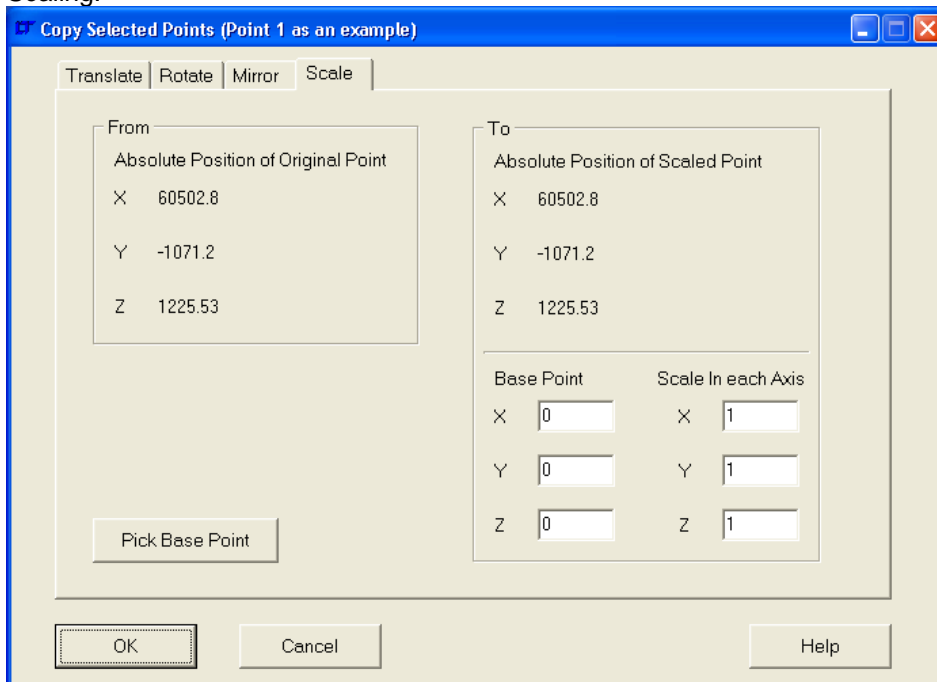
To specify an axis other than the normal axes, press the 'Pick Axis' button. It will then be possible to specify two points to describe the actual axis of rotation using the right hand rule.

Mirroring:-



To select a non-standard mirror plane press the 'Pick Plane' button and interactively select 3 points to define the mirror plane.

Scaling:-



The scale may be different in each axis to distort the copies. Note that when scaling the distance between the selected point and the base point is scaled by the axis scale factor. Consequently points at the base point will appear not to be copied.

**'Geometry| Points| Move'**

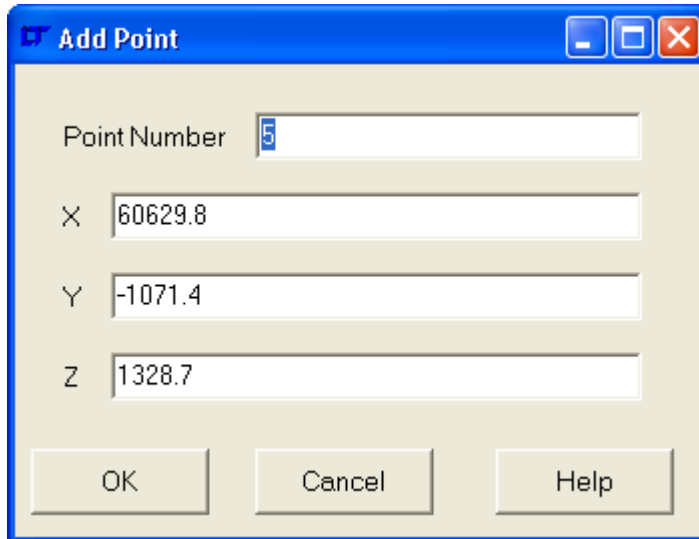
Open the same dialogue box as shown in **'Geometry| Points| Copy'** to enter the data necessary to move the currently selected points. Points may be translated, rotated, mirrored or scaled. No copies are made.

**'Geometry| Points| Add'****'Geometry| Points| Add| Pick Points'**

Starts the interactive point creation routine. Points may be added at any cursor snap location or their X,Y,Z co-ordinates may be entered in the command line.

**'Geometry| Points| Add| Enter Co-ordinates'**

Opens a dialogue box to enter the point number and location, as below:-

**'Geometry| Points| Add| Selected Nodes'**

Adds a point at the same location as every selected node.

**'Geometry| Points| Delete'****'Geometry| Points| Delete| All'**

Delete all points.

**'Geometry| Points| Delete| Selected'**

Delete all selected points.

**'Geometry| Points| Delete| Inactive'**

Delete all points that are not referenced by other entities.

**'Geometry| Points| Select'****'Geometry| Points| Select| All'**

Select all points.

**'Geometry| Points| Select| None'**

Deselect all points.

**'Geometry| Points| Select| Range'**

Select a range of points. Typed input of the range is necessary in the command line.

**'Geometry| Points| Select| Invert Selection'**

Invert the selection of points. All points that are initially invisible are made visible as they are selected.

**'Geometry| Points| Select| Selected Lines'**

Select points by association with selected lines.

**'Geometry| Points| Select| Selected Surfaces'**

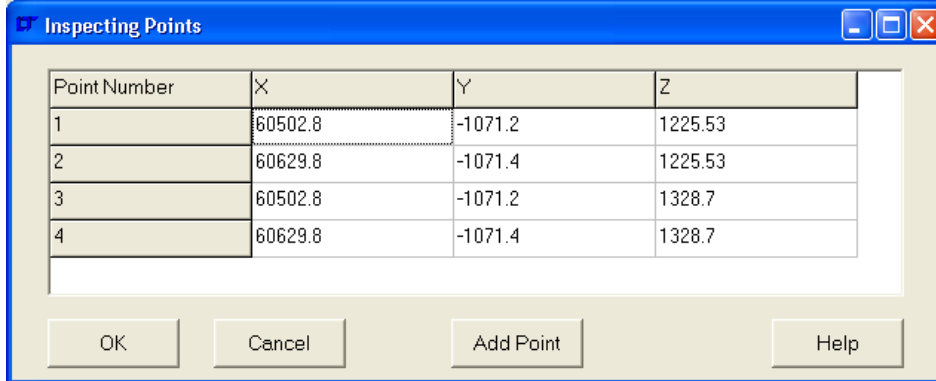
Select points by association with selected surfaces.

**'Geometry| Points| Select| Selected Solids'**

Select points by association with selected solids.

**'Geometry| Points| Inspect'**

Open a dialogue window to inspect, alter locations or add points, as below:-

**'Geometry| Points| Eliminate Duplicates'**

Checks the range of currently selected points and eliminates any duplicated points. All references to points within lines are updated too.

**'Geometry| Points| Find'**

Find a range of points. Typed input of the range is necessary in the command line. The points will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'Geometry| Lines'****'Geometry| Lines| Copy'**

Open a dialogue box to enter the data necessary to copy the currently selected lines. Lines may be translated, rotated, mirrored or scaled. The dialogue box is the same as used for points. All points necessary for the new lines are created automatically.

**'Geometry| Lines| Modify'****'Geometry| Lines| Modify| Move'**

Open the same dialogue box as shown in **'Geometry| Points| Copy'** to enter the data necessary to move the currently selected lines. Lines may be translated, rotated, mirrored or scaled. No copies are made.

**'Geometry| Lines| Modify| Break'**

Prepares to break the selected lines. Type in the number of segments to break each selected line by in the command line. The selected lines will then be hidden and de-selected as each is replaced by the required number of line segments.

**'Geometry| Lines| Add'**

Starts the interactive line creation routine. Lines may be started or finished at any cursor snap location or their X,Y,Z co-ordinates may be entered in the command line.

**'Geometry| Lines| Delete'****'Geometry| Lines| Delete| All'**

Delete all Lines.

**'Geometry| Lines| Delete| Selected'**

Delete all selected Lines.

**'Geometry| Lines| Delete| Inactive'**

Delete all Lines that are not referenced by other entities.

**'Geometry| Lines| Select'****'Geometry| Lines| Select| All'**

Select all Lines.

**'Geometry| Lines| Select| None'**

Deselect all Lines.

**'Geometry| Lines| Select| Range'**

Select a range of Lines. Typed input of the range is necessary in the command line.

**'Geometry| Lines| Select| Invert Selection'**

Invert the selection of Lines. All Lines that are initially invisible are made visible as they are selected.

**'Geometry| Lines| Select| Selected Points'**

Select Lines by association with selected points.

**'Geometry| Lines| Select| Selected Surfaces'**

Select Lines by association with selected surfaces.

**'Geometry| Lines| Select| Selected Solids'**

Select Lines by association with selected solids.

**'Geometry| Lines| Inspect'**

Starts the interactive inspection routine. Using this the line middles may be snapped to and the line information will be displayed in the **'Progress Information'** box.

**'Geometry| Lines| Eliminate Duplicates'**

Checks the range of currently selected lines and eliminates any duplicated lines. All references to lines within surfaces are updated too.

**'Geometry| Lines| Find'**

Find a range of lines. Typed input of the range is necessary in the command line. The lines will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'Geometry| Surfaces'****'Geometry| Surfaces| Copy'**

Open a dialogue box to enter the data necessary to copy the currently selected Surfaces. Surfaces may be translated, rotated, mirrored or scaled. The dialogue box is the same as used for points. All points and lines necessary for the new Surfaces are created automatically.

**'Geometry| Surfaces| Move'**

Open the same dialogue box as shown in **'Geometry| Points| Copy'** to enter the data necessary to move the currently selected Surfaces. Surfaces may be translated, rotated, mirrored or scaled. No copies are made.

**'Geometry| Surfaces| Add'****'Geometry| Surfaces| Add| 2 Lines'**

Starts the interactive Surface creation routine. Surfaces are created by snapping to the midpoints of 2 lines.

**'Geometry| Surfaces| Delete'**



**'Geometry| Surfaces| Delete| All'**

Delete all Surfaces.

**'Geometry| Surfaces| Delete| Selected'**

Delete all selected Surfaces.

**'Geometry| Surfaces| Delete| Inactive'**

Delete all Surfaces that are not referenced by other entities.

**'Geometry| Surfaces| Select'****'Geometry| Surfaces| Select| All'**

Select all Surfaces.

**'Geometry| Surfaces| Select| None'**

Deselect all Surfaces.

**'Geometry| Surfaces| Select| Range'**

Select a range of Surfaces. Typed input of the range is necessary in the command Surface.

**'Geometry| Surfaces| Select| Invert Selection'**

Invert the selection of Surfaces. All Surfaces that are initially invisible are made visible as they are selected.

**'Geometry| Surfaces| Select| Selected Points'**

Select Surfaces by association with selected points.

**'Geometry| Surfaces| Select| Selected Lines'**

Select Surfaces by association with selected lines.

**'Geometry| Surfaces| Select| Selected Solids'**

Select Surfaces by association with selected solids.

**'Geometry| Surfaces| Inspect'**

Starts the interactive inspection routine. Using this the Surface centroids may be snapped to and the Surface information will be displayed in the **'Progress Information'** box.

**'Geometry| Surfaces| Reverse Normal'**

Reverses the normals of the selected surfaces.

**'Geometry| Surfaces| Find'**

Find a range of surfaces. Typed input of the range is necessary in the command line. The surfaces will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'Geometry| Mesh'****'Geometry| Mesh| Points'****'Geometry| Mesh| Points| All'**

Mesh all points with point elements according to the current 'Point Element Mesh Settings'.

**'Geometry| Mesh| Points| Selected'**

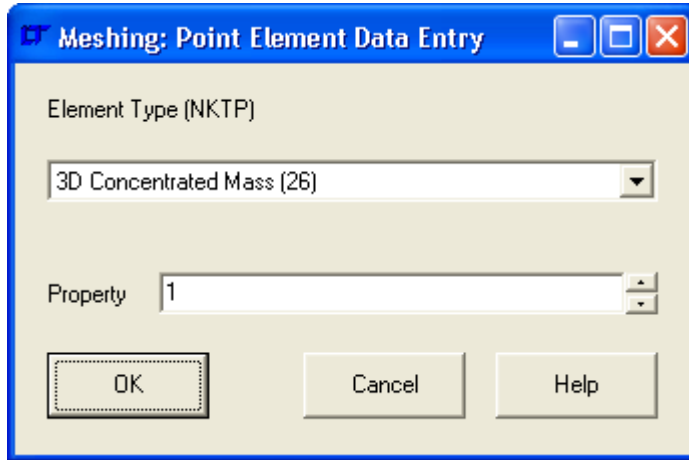
Mesh selected points with point elements according to the current 'Point Element Mesh Settings'.

**'Geometry| Mesh| Points| Pick Points'**

Start interactive point meshing routine. Points selected on screen are meshed with point elements according to the current 'Point Element Mesh Settings'.

**'Geometry| Mesh| Points| Mesh Settings'**

Open a window to specify the current 'Point Element Mesh Settings'. The window is as below:-



**'Geometry| Mesh| Lines'**

**'Geometry| Mesh| Lines| All'**

Mesh all Lines with Line elements according to the current 'Line Element Mesh Settings'.

**'Geometry| Mesh| Lines| Selected'**

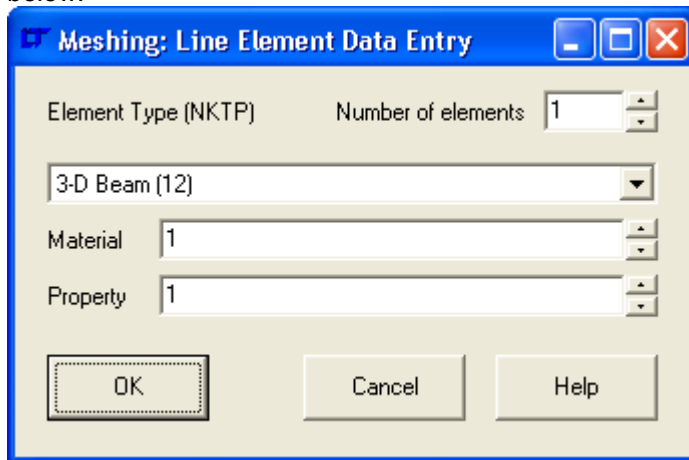
Mesh selected Lines with Line elements according to the current 'Line Element Mesh Settings'.

**'Geometry| Mesh| Lines| Pick Lines'**

Start interactive Line meshing routine. Lines selected on screen are meshed with Line elements according to the current 'Line Element Mesh Settings'. It is possible to change the numbers of elements meshed along the line by the **Command Line** while the 'Mesh Lines' function is operative. This is achieved by typing 'n=[number]'. To change the material, type 'm=[material number]', or property, 'p=[property number]'.

**'Geometry| Mesh| Lines| Mesh Settings'**

Open a window to specify the current 'Line Element Mesh Settings'. The window is as below:-



**'Geometry| Mesh| Surfaces'**

**'Geometry| Mesh| Surfaces| All'**

Mesh all Surfaces with Surface elements according to the current 'Surface Element Mesh Settings'. Where possible the surfaces are meshed as the more accurate rectangular elements. Where the surface tapers to a corner, triangular elements are put into the corner.

**'Geometry| Mesh| Surfaces| Selected'**

Mesh selected Surfaces with Surface elements according to the current 'Surface Element Mesh Settings'.

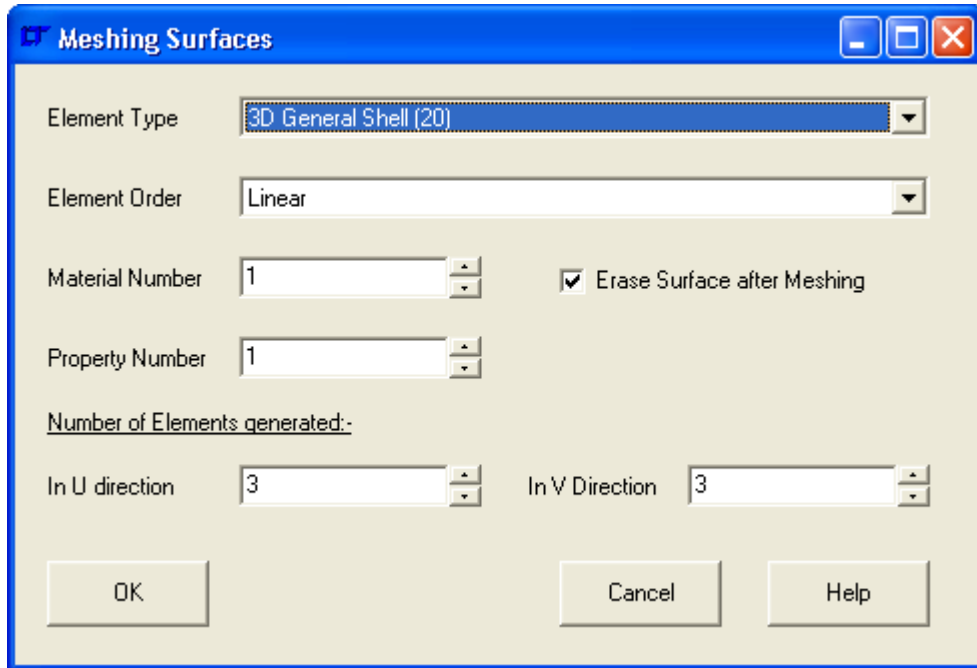
**'Geometry| Mesh| Surfaces| Pick Surfaces'**

Start interactive Surface meshing routine. Surfaces selected on screen are meshed with Surface elements according to the current 'Surface Element Mesh Settings'.

It is possible to change the numbers of elements meshed in the U and V directions by the **Command Line** while the 'Mesh Surfaces' function is operative. This is achieved by typing 'u=[number]' or 'v=[number]' as appropriate. To change the material, type 'm=[material number]', or property, 'p=[property number]'.

**‘Geometry| Mesh| Surfaces| Mesh Settings’**

Open a window to specify the current ‘Surface Element Mesh Settings’. The window is as below:-

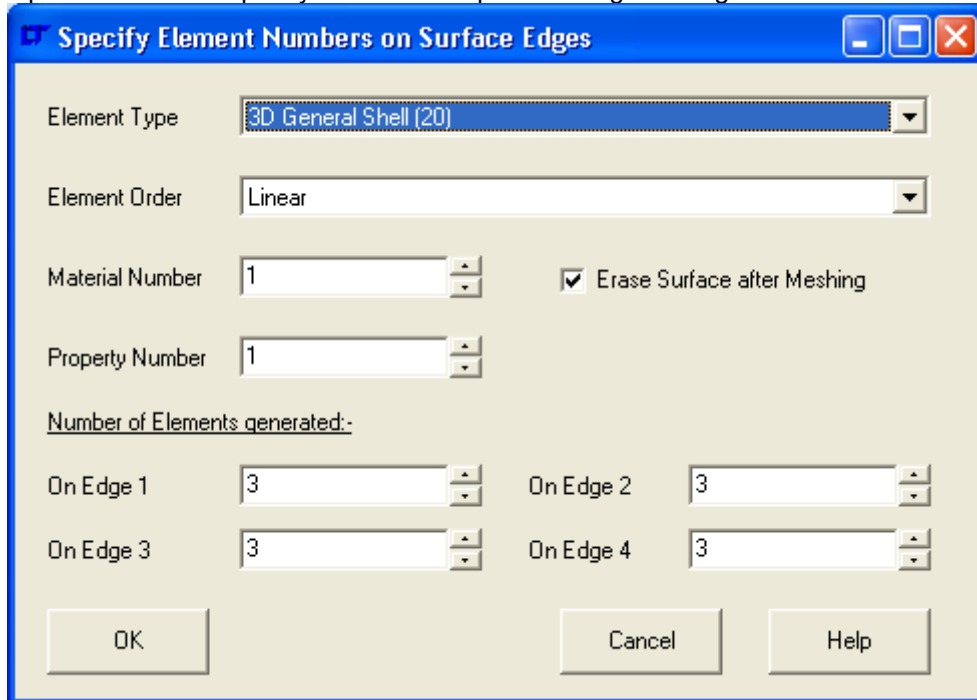


**‘Geometry| Mesh| Surfaces| Pick Specified Edges’**

Mesh Surfaces to generate numbers of elements along each edge complying with the current ‘Specified Edge Settings’. When the surface is triangular, the number specified for the elements on edge 4 is ignored.

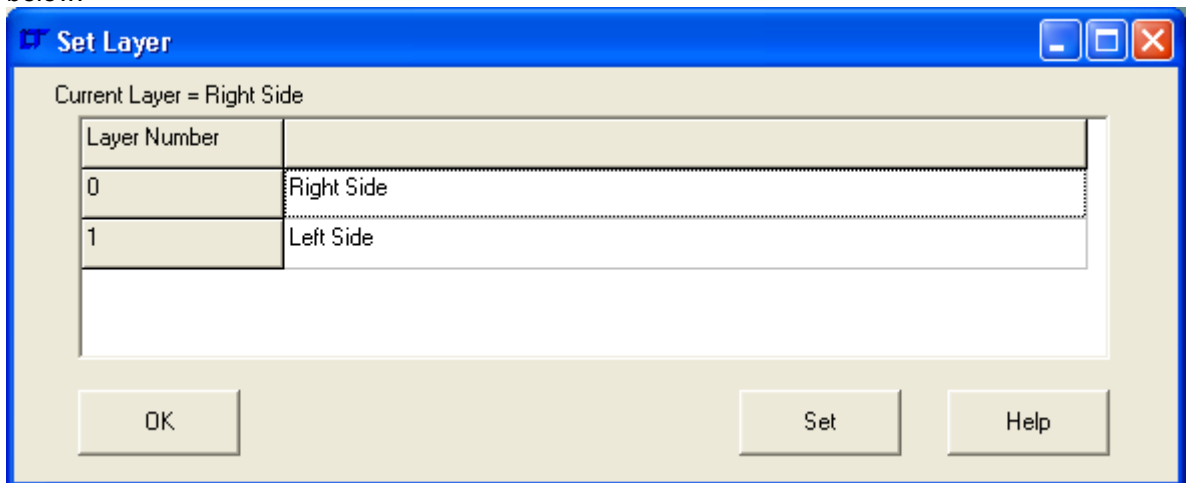
**‘Geometry| Mesh| Surfaces| Specified Edge Settings’**

Open a window to specify the current ‘Specified Edge Settings’. The window is as below:-

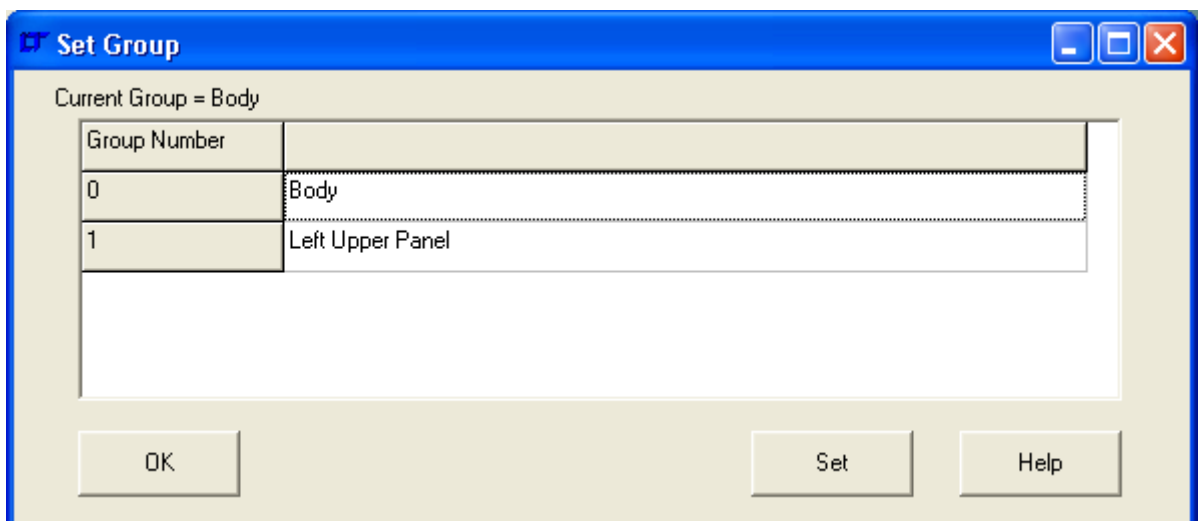


**'Geometry| Mesh| Set Current Layer'**

Opens a dialogue box to set the layer the new elements will be added to. The dialogue box is as below:-

**'Geometry| Mesh| Set Current Group'**

Opens a dialogue box to set the group the new elements will be added to. The dialogue box is as below:-



### 3.2.6 FE Model Menu

The following menu commands may be found in the FE Model menu.

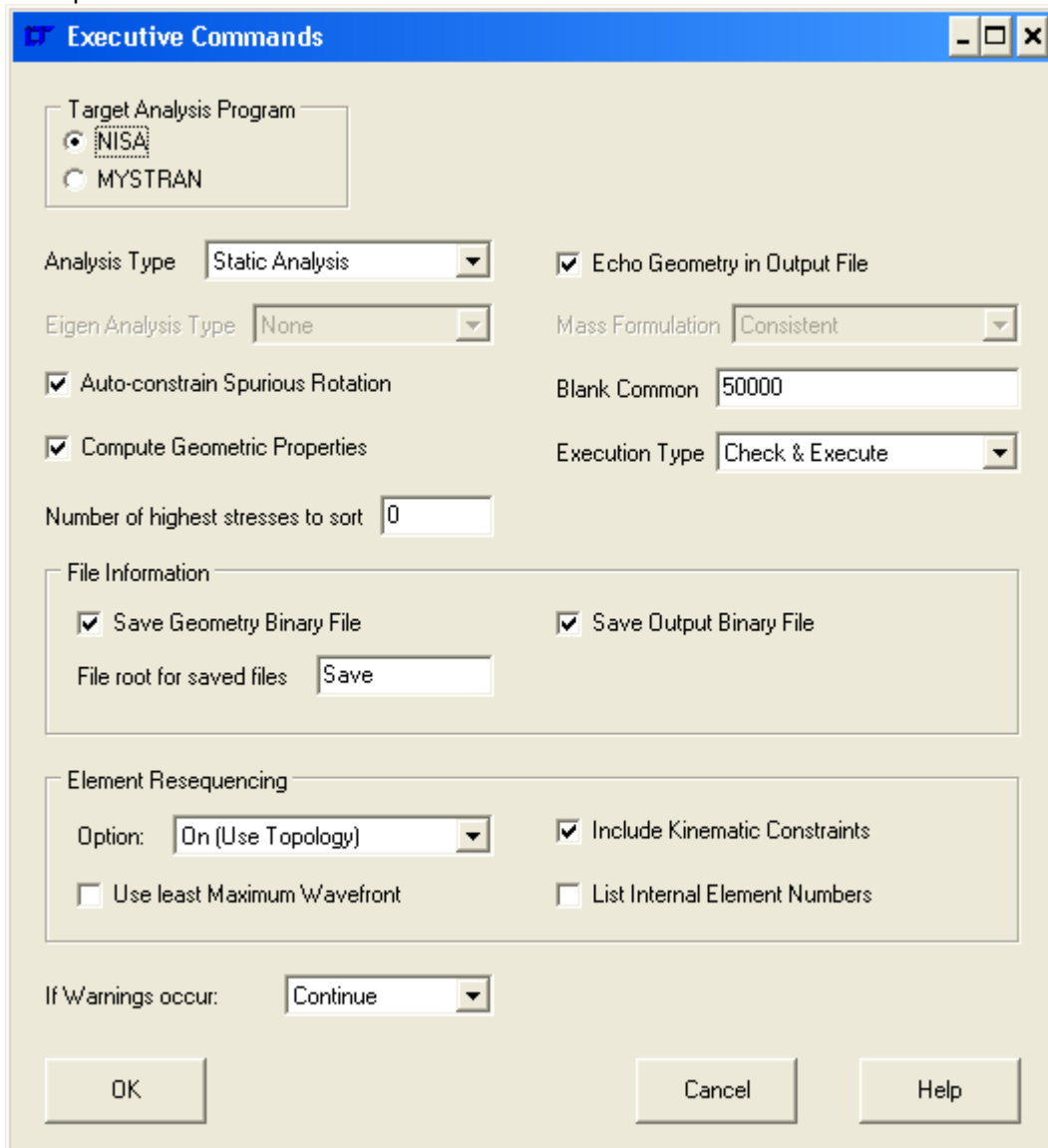
#### 'FE Model| Executive Commands'

Controls the analysis destination program and the analysis. The executive data tells the FE package what to expect and what data to operate on during the analysis. It also tells the Chestnut Pens Modelling Studio what to allow the user to specify by being aware of what the destination FE package is. This saves the user time;

For example: there is no point in specifying laminate definition for a MYSTRAN model, as MYSTRAN cannot analyse laminate elements.

Both NISA and MYSTRAN have different capabilities, so as the target analysis program is chosen, the options alter.

The options available for NISA are shown below:-



#### Analysis Type

The analysis type (at the time of writing this command reference) must be 'Static Analysis'. If you choose the Eigen analysis tag, the Eigen Analysis type must be specified and the mass formulation model. The default mass formulation model is the more accurate but more processor intensive 'Consistent' model. The alternative is the 'lumped' model which discretises the element mass at its centroid.

Echo Geometry in the output file

Echoing the geometry in the output file gives a post-processing package enough information to re-build the model without needing to open the original model file.

Auto-constrain spurious rotation

With shell elements it's possible to have spurious rotations of the elements about their normal's. To prevent this check 'Auto-constrain Spurious Rotation'.

Blank Common

For a NISA analysis the size of the Blank Common scratch area needs to be defined. This must be at least 7x the number of nodes – but experience shows that this is a minimum figure that sometimes isn't enough. When the NISA file is exported (using 'File| Export') the minimum value of Blank Common is calculated and rounded up to the next highest 50,000. The number that is exported is the higher of the figure entered here and the figure calculated.

Compute Geometric Properties

This specifies whether NISA computes and prints the mass and mass moment of inertia of the model used.

Execution Type

This specifies whether the analysis module checks, executes or checks and executes the NISA file.

Number of Highest Stresses to Sort

NISA can print a number of sorted stresses. If this is zero, no stresses are sorted.

Save Geometry/ Output Binary Files

If binary files are used to save the model data for easy access by EMRC's DISPLAY III or DISPLAY IV post processing modules. The geometry binary file saved is the 26.dat file, while the output binary file is the 27.dat file. If the user wants other binary files, then the NISA file needs to be edited in a text editor.

File root for saved files

If the binary files are saved, and the file root is 'File', then the binary geometry file saved by NISA will be 'File26.dat'.

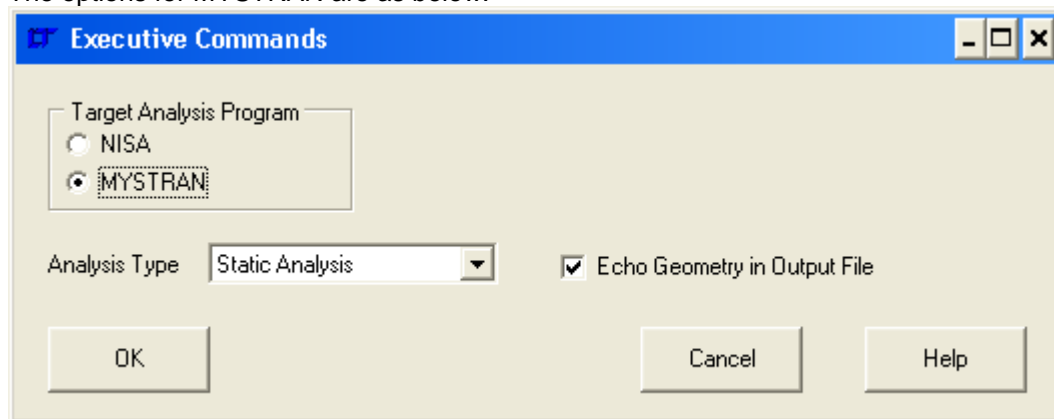
Element Resequencing Option

Select the option to specify how resequencing is conducted. The default is 'On (Use Topology)', and there is rarely any need to alter this. See the NISA user's manual for further details on resequencing options.

Warnings

Determines whether warnings permit the analysis to continue.

The options for MYSTRAN are as below:-



Analysis Type

The analysis type (at the time of writing this command reference) must be 'Static Analysis'.

Echo Geometry in the output file

Echoing the geometry in the output file gives a post-processing package enough information to rebuild the model without needing to open the original model file.



**'FE Model| Nodes'****'FE Model| Nodes| Copy'**

Open a dialogue box to enter the data necessary to copy the currently selected nodes. Nodes may be translated, rotated, mirrored or scaled. The dialogue box is the same as used for points.

**'FE Model| Nodes| Move'****'FE Model| Nodes| Move| Move'**

Open the same dialogue box as shown in **'Geometry| Points| Copy'** to enter the data necessary to move the currently selected nodes. Nodes may be translated, rotated, mirrored or scaled. No copies are made.

**'FE Model| Nodes| Move| Align X'**

Type in the new X value for the selected nodes and press 'Enter'.

**'FE Model| Nodes| Move| Align Y'**

Type in the new Y value for the selected nodes and press 'Enter'.

**'FE Model| Nodes| Move| Align Z'**

Type in the new Z value for the selected nodes and press 'Enter'.

**'FE Model| Nodes| Add'****'FE Model| Nodes| Add| Pick Points'**

Starts the interactive node creation routine. Nodes may be added at any cursor snap location or their X,Y,Z co-ordinates may be entered in the command line.

**'FE Model| Nodes| Add| Enter Co-ordinates'**

Opens a dialogue box to enter the node number and location, as below:-

**'FE Model| Nodes| Add| Selected Points'**

Adds a node at the same location as every selected point.

**'FE Model| Nodes| Delete'****'FE Model| Nodes| Delete| All'**

Delete all nodes not referenced by elements, rigid links, displacement constraints, nodal pressures or forces.

**'FE Model| Nodes| Delete| Selected'**

Delete all selected nodes not referenced by elements, rigid links, displacement constraints, nodal pressures or forces.

**'FE Model | Nodes| Select'**

**'FE Model | Nodes | Select | All'**

Select all Nodes.

**'FE Model | Nodes | Select | None'**

Deselect all Nodes.

**'FE Model | Nodes | Select | Range'**

Select a range of Nodes. Typed input of the range is necessary in the command line.

**'FE Model | Nodes | Select | Invert Selection'**

Invert the selection of Nodes. All Nodes that are initially invisible are made visible as they are selected.

**'FE Model | Nodes | Select | Selected Elements'**

Select Nodes by association with selected Elements.

**'FE Model | Nodes | Select | Selected Rigid Links'**

Select Nodes by association with selected Rigid Links.

**'FE Model | Nodes | Selected Displacements'**

Select Nodes by association with selected Displacement Constraints.

**'FE Model | Nodes | Selected Pressures'**

Select Nodes by association with selected Pressures.

**'FE Model | Nodes | Selected Nodal Pressures'**

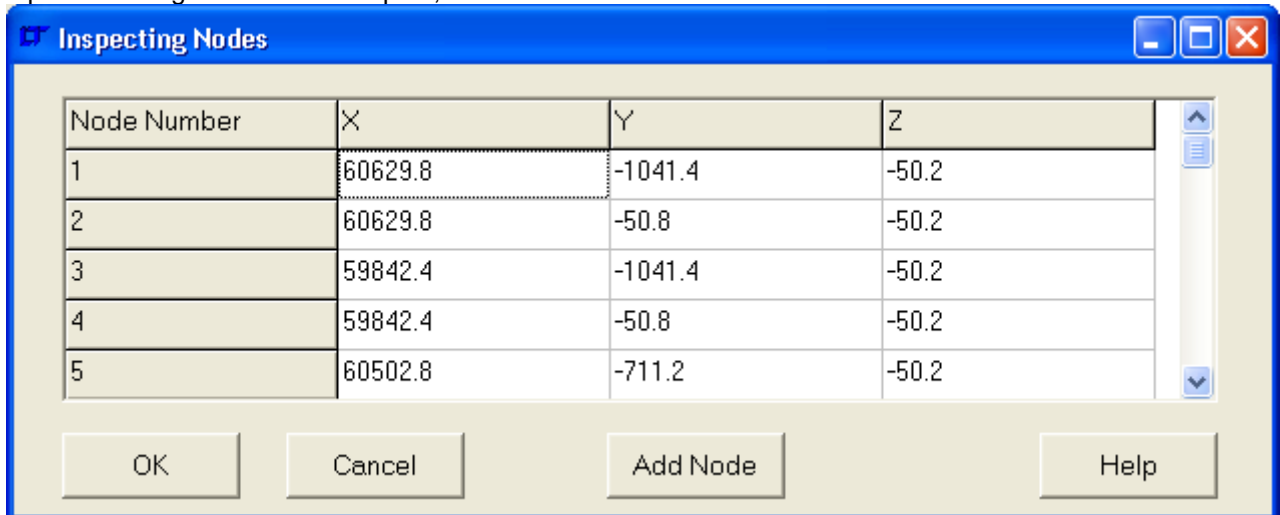
Select Nodes by association with selected Nodal Pressures.

**'FE Model | Nodes | Selected Forces'**

Select Nodes by association with selected Forces.

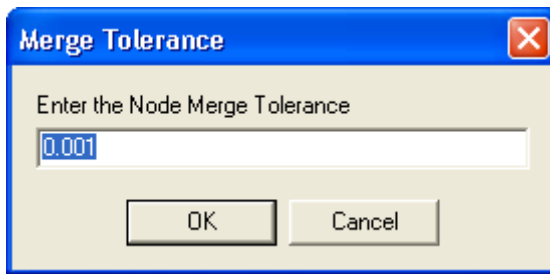
**'FE Model | Nodes | Inspect'**

Opens a dialogue window to inspect, edit or add nodes. The window is as below:-



**'FE Model | Nodes | Merge'**

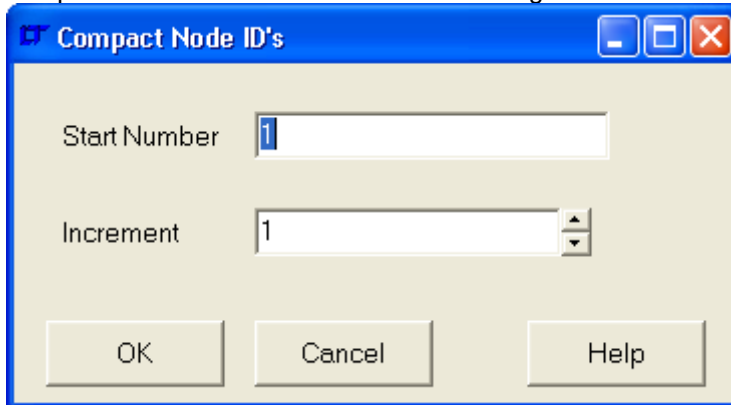
Merge the selected nodes. A dialogue window opens requesting the current merge tolerance for the nodes. This is the maximum distance at which nodes will be considered to be representing the same geometric point. The dialogue window is as below:-

**'FE Model| Nodes| Delete Inactive'**

Deletes nodes not referenced by elements, rigid links, displacement constraints, nodal pressures or forces. If any nodes are selected at the point of selecting the command, the deletion is restricted to the selected nodes. Otherwise all nodes are checked.

**'FE Model | Nodes| Compact ID's'**

Opens a dialogue box requesting the starting number and increment for the node number compaction of the selected nodes. The dialogue box is as below:-

**'FE Model | Nodes| Set Merge Tolerance'**

Sets the Node Merge Tolerance. This merge tolerance may need to be set prior to importing STL files when Nodes are automatically merged.

**'FE Model | Nodes | Find'**

Find a range of nodes. Typed input of the range is necessary in the command line. The nodes will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'FE Model| Elements'**

**'FE Model| Elements| Copy'**

**'FE Model| Elements| Copy| Copy'**

Open a dialogue box to enter the data necessary to copy the currently selected Elements. Elements may be translated, rotated, mirrored or scaled. The dialogue box is the same as used for points.

**'FE Model| Elements| Copy| Offset Selected Elements'**

The selected first order shell elements (NORDR 1 or 10 only) are copied and offset in the direction of the element normal. The user is required to type in the offset.

**'FE Model| Elements| Move'**

Open the same dialogue box as shown in **'Geometry| Points| Copy'** to enter the data necessary to move the currently selected Elements. Elements may be translated, rotated, mirrored or scaled. No copies are made.

**'FE Model| Elements| Add'**

**'FE Model| Elements| Add| Pick Nodes'**

Create Elements by picking the nodes that define the elements, element by element. The element type that is created is defined by the data entered using menu option **'FE Model| Elements| Add| Settings'**. The default settings are 'General Shell, Type 20, Linear Formulation, Material 1, Property 1. The node pick order is specified in the 'Element Library' section of the 'User Guide'. A simplified pick order guide is given below:-

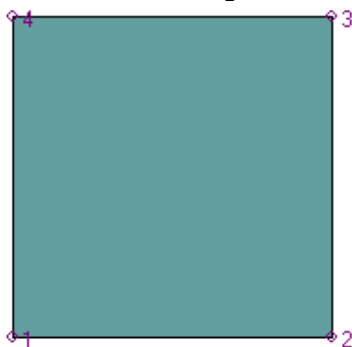
- 1) Point Elements. Pick one Node.
- 2) Line Elements. Pick two Nodes.
- 3) Shell Elements. Pick Nodes sequentially around the perimeter going anticlockwise about the intended shell normal. If midside nodes are required, pick them in the same sequence.
- 4) Solid Elements. Operation 1: Pick Nodes sequentially around the perimeter of the bottom face, going anticlockwise around the element centroid. If midside nodes are required, pick them in the same sequence. Operation 2: If midside nodes are required, pick the midside nodes on the edges linking the bottom face to the top face, going anti clockwise around the element centroid, starting on the edge linking to the first node. Operation 3: Pick Nodes sequentially around the perimeter of the bottom face, going in the same direction around the element centroid, starting on the corner nearest the first picked node. If midside nodes are required, pick them in the same sequence. If a pyramid element is required, pick only one node, then pick the first node again.

Note:- When creating Solid elements by nodal picks, only 8 Noded Bricks, 20 Noded Bricks, 4 noded Triangular Pyramids and 10 Noded Triangular Pyramid elements can be created.

The pick settings do not specify whether a rectangular element or triangular element is to be created. This is done by the sequence of nodes picked.

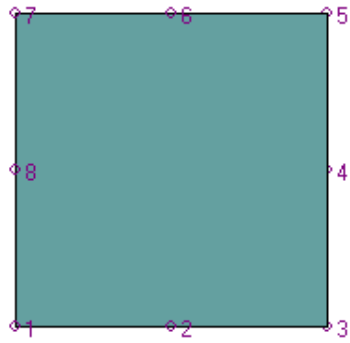
Examples:-

To create a linear rectangular shell element, pick the four corner nodes, as below:-



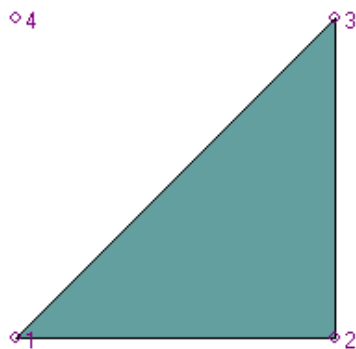
The node pick sequence would be 1,2,3,4.

To create a parabolic rectangular shell element, pick the 8 nodes below:-



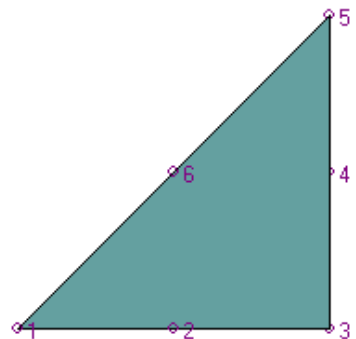
The node pick sequence would be 1,2,3,4,5,6,7,8

To create a linear triangular element, pick on the first node after picking the final node.



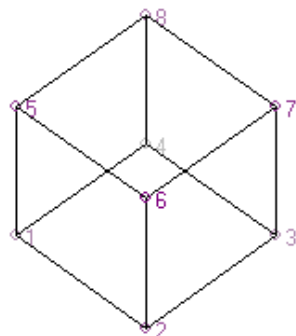
The node pick sequence would be 1,2,3,1.

To create a parabolic triangular element, pick on the first node after picking the final node.



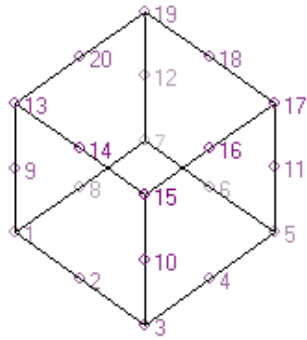
The node pick sequence would be 1,2,3,4,5,6,1.

To create a linear rectangular Brick Element, pick the nodes in the sequence shown below:-



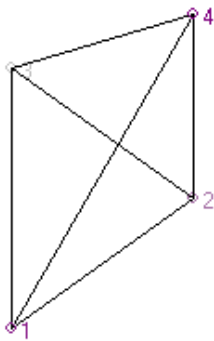
The node pick sequence is given by the node numbers.

To create a parabolic rectangular Brick Element, pick the nodes in the sequence shown below:-



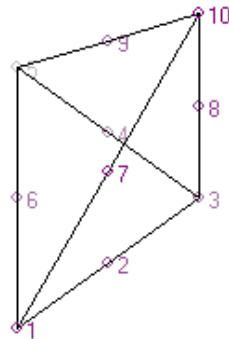
The node pick sequence is given by the node numbers.

To create a linear 4 noded pyramid element, pick the nodes in the sequence shown.



The node pick sequence is 1,2,3,4,1.

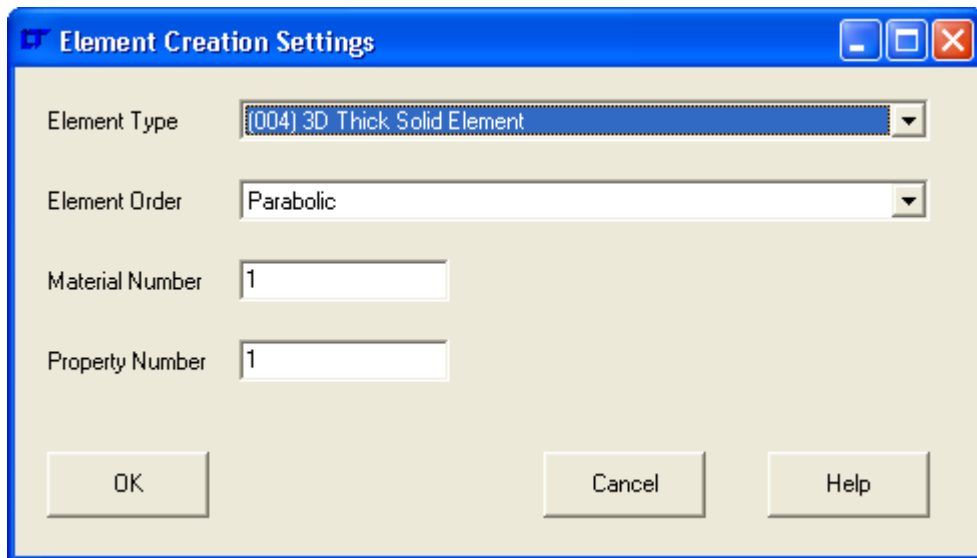
To create a parabolic 10 noded pyramid element, pick the nodes in the sequence shown.



The node pick sequence is 1,2,3,4,5,6,7,8,9,10,1.

#### **'FE Model| Elements| Add| Settings'**

Opens a window to specify the element settings for the next element created with '**FE Model| Elements| Add| Pick Nodes**'. Pick the element type, material, property and whether the element is linear or parabolic.

**'FE Model| Elements| Delete'****'FE Model| Elements| Delete| All'**

Delete all Elements not referenced by pressures.

**'FE Model| Elements| Delete| Selected'**

Delete all selected Elements not referenced by pressures.

**'FE Model| Elements| Select'****'FE Model| Elements| Select| All'**

Select all Elements.

**'FE Model| Elements| Select| None'**

Deselect all Elements.

**'FE Model| Elements| Select| Range'**

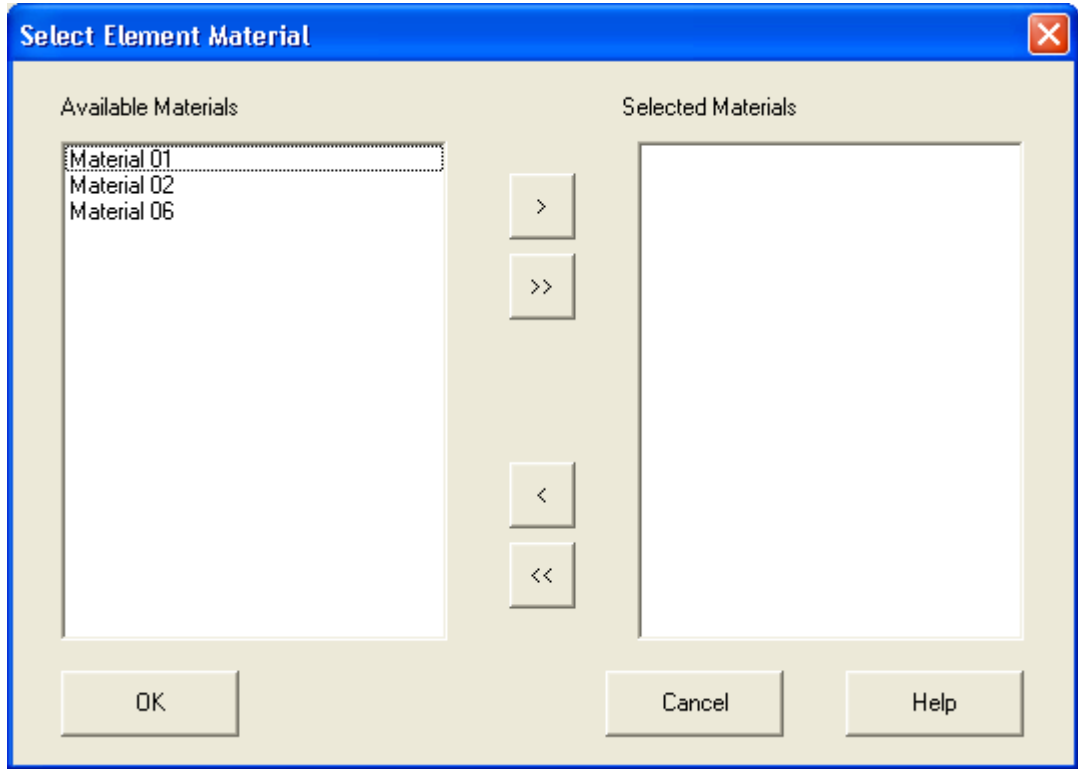
Select a range of Elements. Typed input of the range is necessary in the command line.

**'FE Model| Elements| Select| Invert Selection'**

Invert the selection of Elements. All Elements that are initially invisible are made visible as they are selected.

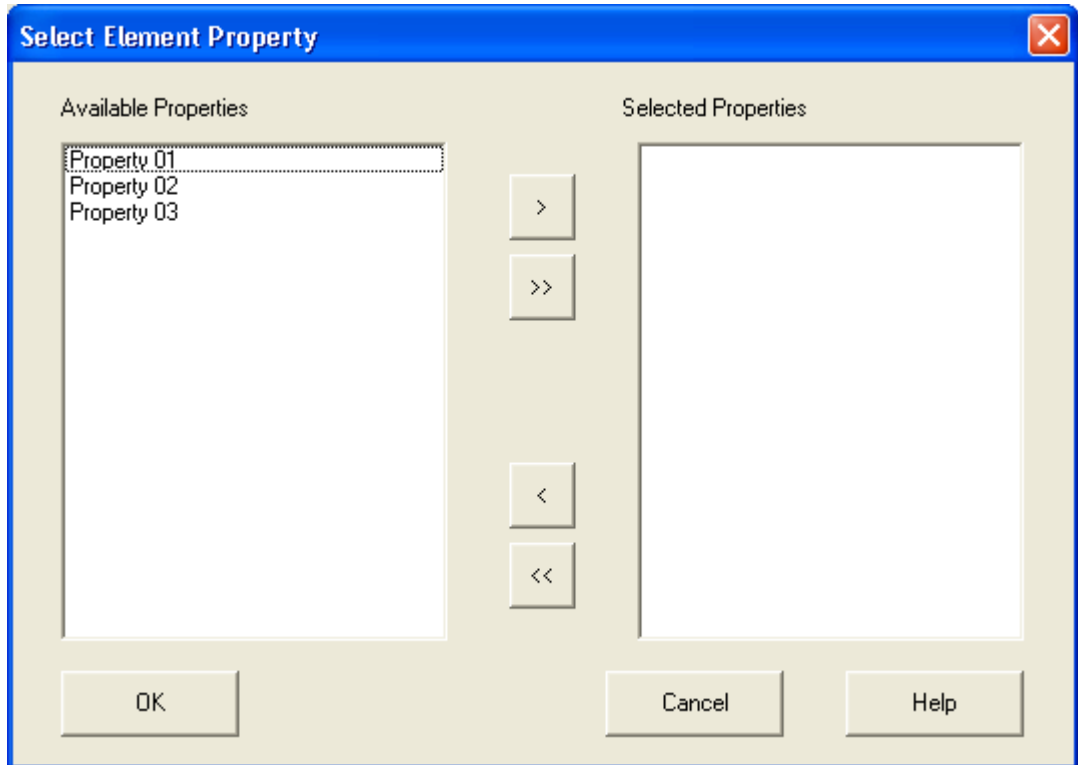
**'FE Model| Elements| Select| by Material'**

Select Elements dependant on a range of material numbers. Selection of the element material range is necessary, from a list of options. This is carried out in a dialogue window as below:-



**'FE Model| Elements| Select| by Property'**

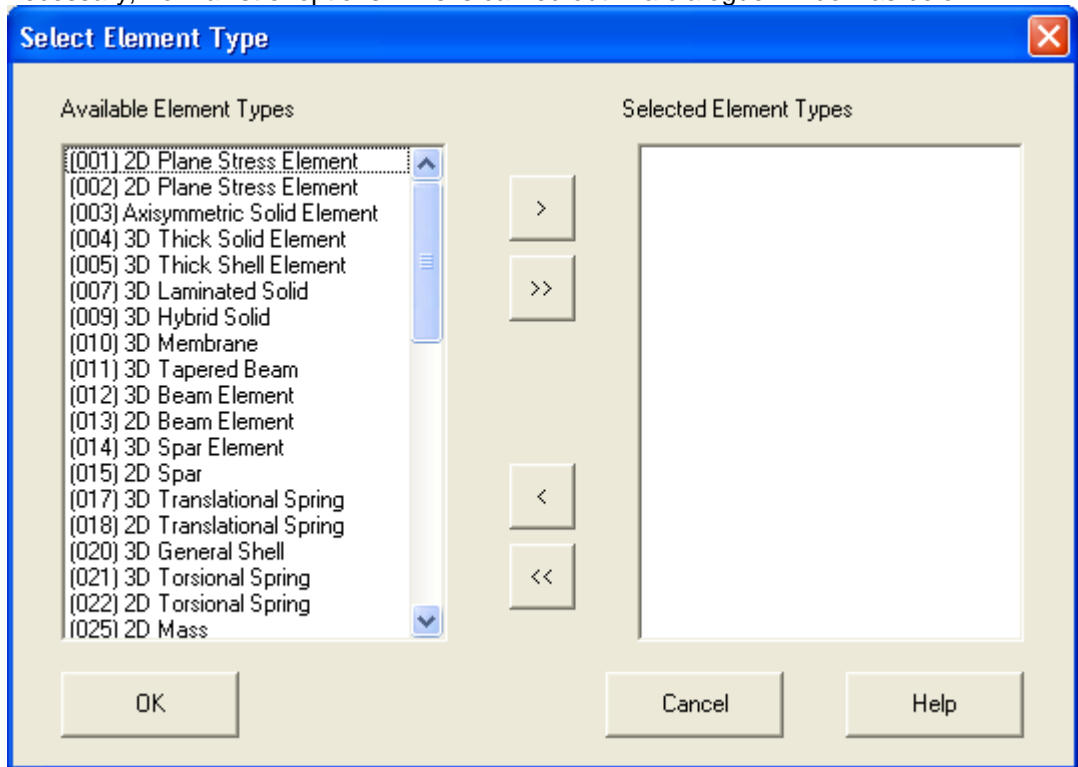
Select Elements dependant on a range of property numbers. Selection of the element property range is necessary, from a list of options. This is carried out in a dialogue window as below:-



**'FE Model| Elements| Select| by Element Type'**



Select Elements with element type number in range. Selection of the element type range is necessary, from a list of options. This is carried out in a dialogue window as below:-



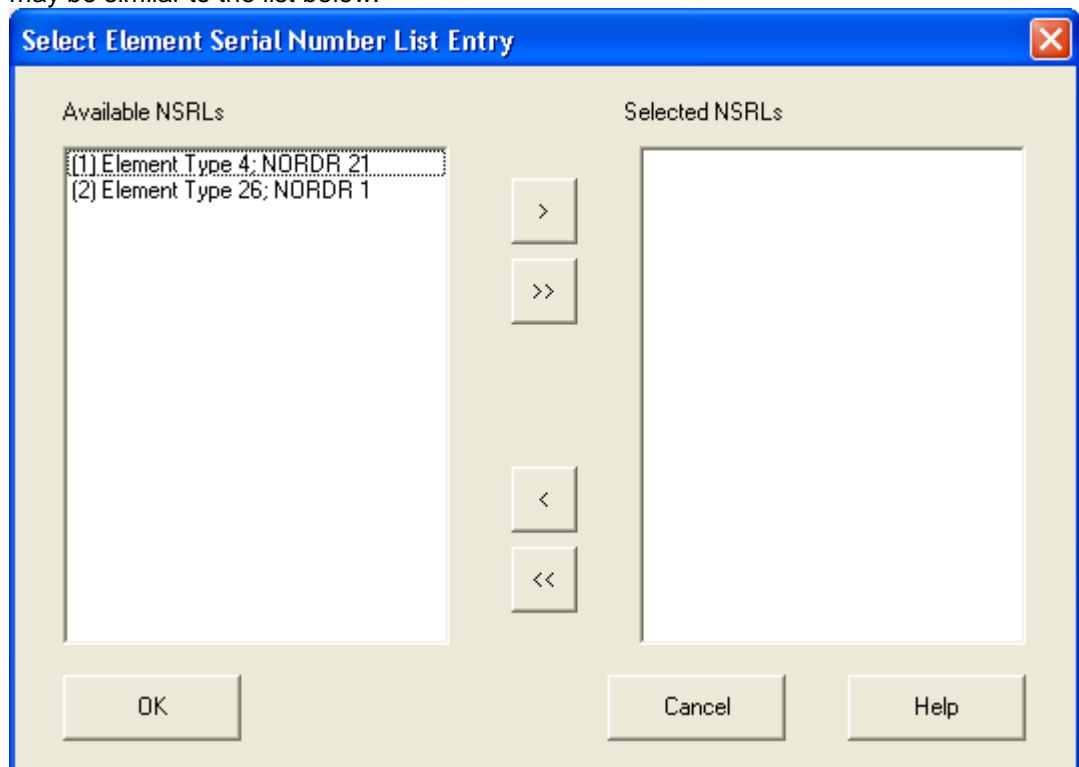
The element type number is from the list below:-

Type	Description
1	2D Plane Stress Element
2	2D Plane Stress Element
3	Axisymmetric Solid Element
4	3D Thick Solid Element
5	3D Thick Shell Element
7	3D Laminated Solid
9	3D Hybrid Solid
10	3D Membrane
11	3D Tapered Beam
12	3D Beam Element
13	2D Beam Element
14	3D Spar Element
15	2D Spar
17	3D Translational Spring
18	2D Translational Spring
20	3D General Shell
21	3D Torsional Spring
22	2D Torsional Spring
25	2D Mass
26	3D Mass
27	2D General Mass
28	3D General Mass
29	2D Mass and Inertia
30	3D Mass and Inertia
32	3D Laminated Composite
33	3D Laminated Sandwich (Honeycomb)
34	Axisymmetric Solid with Non-Axisymmetric Loading
36	Axisymmetric shell
37	Axisymmetric shell with Non-Axisymmetric Loading
38	3D General Spring
39	3D General Beam

40	3D Thin Shell
41	3D Mixed Interpolation 8 Node General Shell
42	2D or Axisymmetric Gap
43	3D Gap
45	3D Cable
46	3D Straight Pipe
47	3D Elbow
48	3D Translational Damper
49	2D Gap/Friction
50	3D Gap/Friction
104	3D Solid for Heat Transfer
105	3D Thick Shell for Heat Transfer
112	3D Bar for Heat Transfer
120	3D Thin Shell for Heat Transfer
149	3D Convection Link for Heat Transfer
150	3D Radiation Link for Heat Transfer

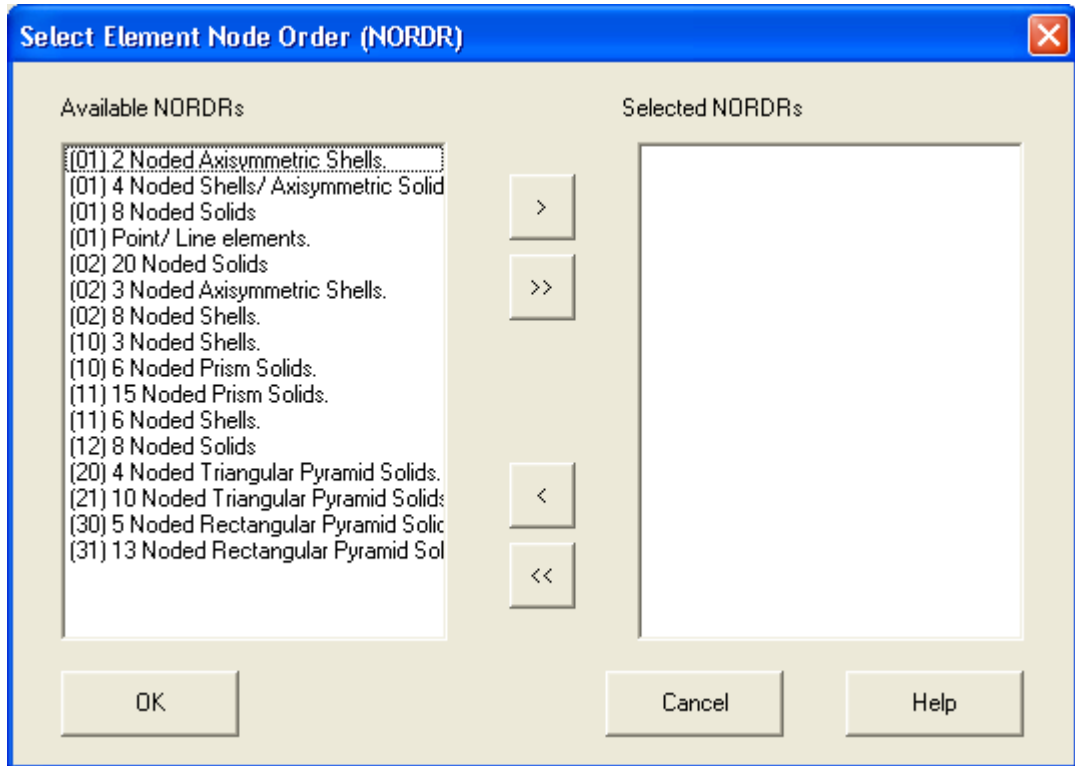
**'FE Model| Elements| Select| by NSRL'**

Select Elements with NSRL number in range. The NSRL of the elements to be erased is selected from a list of Serial Numbers generated for the current model. The selection list may be similar to the list below:-



**'FE Model| Elements| Select| by NORDR'**

Select Elements with Node Order number in range. Selection of the Node Order is from a list as below:-

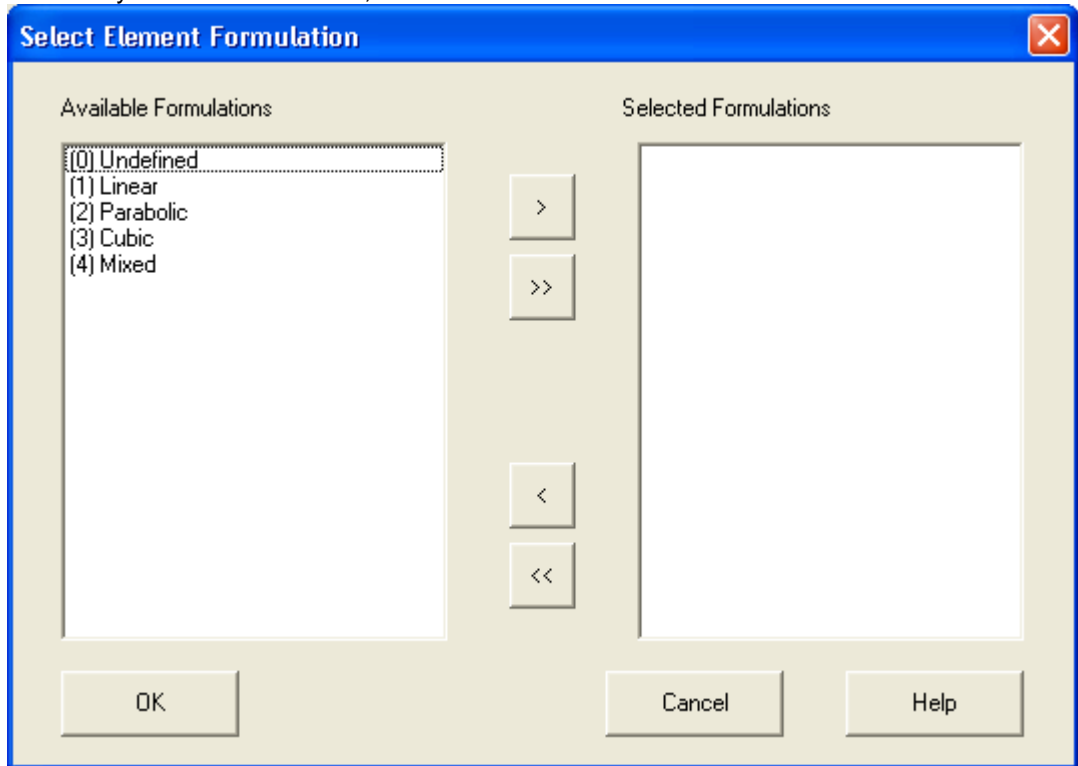


The Node Order is the node order of the element, and is defined as below:-

Node Order	Description
1	For all point or line elements
1	For all 2 noded axisymmetric shell elements
1	For all 4 noded shell elements/ 4 noded axisymmetric solid elements
1 or 12	For all 8 noded solid elements
2	For all 3 noded axisymmetric elements
2	For all 8 noded shell elements
2	For all 20 noded brick elements
10	For 3 Noded triangular shell elements
10	6 noded prism shaped brick elements
11	6 Noded triangular shell elements
11	15 noded prism brick elements
20	4 noded triangular pyramid solid elements
21	10 noded triangular pyramid solid elements
30	5 noded rectangular pyramid elements
31	13 noded rectangular pyramid elements

**‘FE Model| Elements| Select| by Formulation’**

Select Elements with formulation number in range. Selection of the formulation range is necessary from a selection list, as below:-



Formulation is as below:-

- 1      Linear
- 2      Parabolic
- 3      Cubic
- 4      Mixed

**‘FE Model| Elements| Select| Selected Nodes’**

Select Elements by association with selected Nodes.

**‘FE Model| Elements| Select| Selected Rigid Links’**

Select Elements by association with selected Rigid Links.

**‘FE Model| Elements| Select| Select by Group Name’**

Opens a dialogue window to choose the element group name to select.

**‘FE Model| Elements| Select| Deselect by Group Name’**

Opens a dialogue window to choose the element group name to deselect.

**‘FE Model| Elements| Modify’**

**‘FE Model| Elements| Modify| Separate’**

The currently selected elements are separated from the rest of the model by adding nodes at the boundary between the selected elements and the unselected elements. In effect it is an ‘Unmerge’ function. This is not a hugely useful function for FE analysis, however on rendering packages it can be important to have crisp edges, and element separation can force rendering packages that use surface smoothing to retain the crisp edge where normally it would be smoothed away.

**‘FE Model| Elements| Modify| Slice’**

The currently selected elements are cut along a plane that is entered interactively using snaps or typed locations from the command line. At the time of writing the command reference the element slice only works with linear formulation shell elements. It is planned that sometime line elements will be added. One day solid elements may also be added. It

is not expected that parabolic formulation elements will ever be added to this function (too difficult to cover all possible options – each edge in an element could be cut up to twice leading to a dreadful multiplicity of possible meshing shapes).

If you have a parabolic formulation shell element, convert it to a linear element, slice it, then convert it back to a parabolic element.

**‘FE Model| Elements| Modify| Order: Linear to Parabolic’**

Convert the currently selected linear formulation shell or solid elements to parabolic formulation. Remember to merge nodes after this operation as midside nodes are NOT merged automatically.

**‘FE Model| Elements| Modify| Order: Parabolic to Linear’**

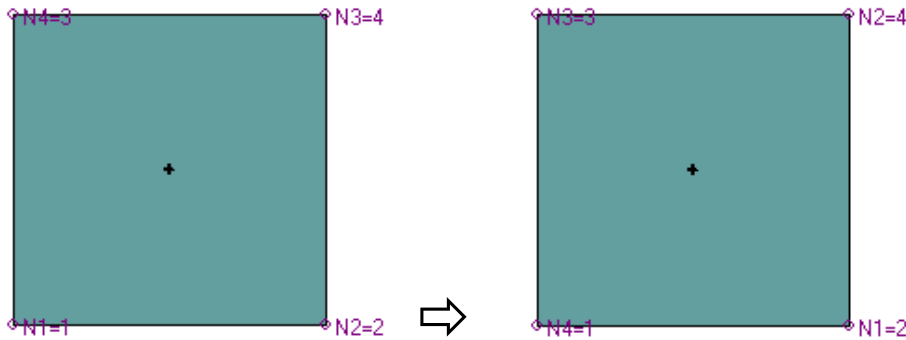
Convert the currently selected parabolic formulation shell or solid elements to linear formulation.

**‘FE Model| Elements| Modify| Reverse Normal’**

Reverse the normals of the currently selected shell elements.

**‘FE Model| Elements| Modify| Shift Element Origin’**

Shifts the origin of the selected elements one corner anti-clockwise about the normal of each currently selected shell element. This is shown below:-



This permits material orientations to be changed if they are oriented in the wrong axis.

**‘FE Model| Elements| Modify| Change Material Number’**

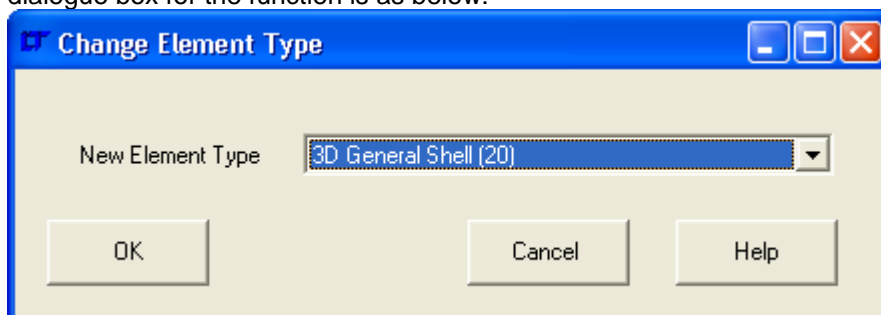
Change the material number of the currently selected shell elements. Typed input of the material number is necessary in the command line.

**‘FE Model| Elements| Modify| Change Property Number’**

Change the property number of the currently selected shell elements. Typed input of the property number is necessary in the command line.

**‘FE Model| Elements| Modify| Change Element Type’**

Change the element type number of the currently selected elements. Line elements can only be converted to other line element types, and shell elements can only be converted to other shell element types. If there is a mixture of element types selected, the Modelling Studio aborts the function and asks the user to select only one element type. A typical dialogue box for the function is as below:-



**'FE Model| Elements| Modify| Thicken Shell'**

The selected first order shell elements (NORDR 1 or 10 only) are copied and offset in the opposite direction to the element normal and edge capping is added to close off the volume created. The user is required to type in the thickness.

**'FE Model| Elements| Modify| Avoid Back Face Culling'**

The selected shell elements are copied and an element with a normal in the opposite direction is created using the same nodes. This can be a quick & dirty way of preventing some 3D packages making certain faces transparent due to back face culling.

**'FE Model| Elements| Modify| De-Triangulate'**

In the selected elements, detriangulate as many elements as possible. The function checks to see if any two triangular elements have a common long edge, and merge that edge to form a quadrilateral element. Only works with Element Type 20, Node Order 10 elements, turning them into Node Order 1 elements. Note: This function deletes the triangles. Texture mapping is preserved, but node normal directions are lost. If the operation is undone, the undeleted elements lose any texture mapping and node normal directions.

**'FE Model| Elements| Inspect'**

Starts the interactive element inspection routine. Click on an element centroid to ascertain information about the element.

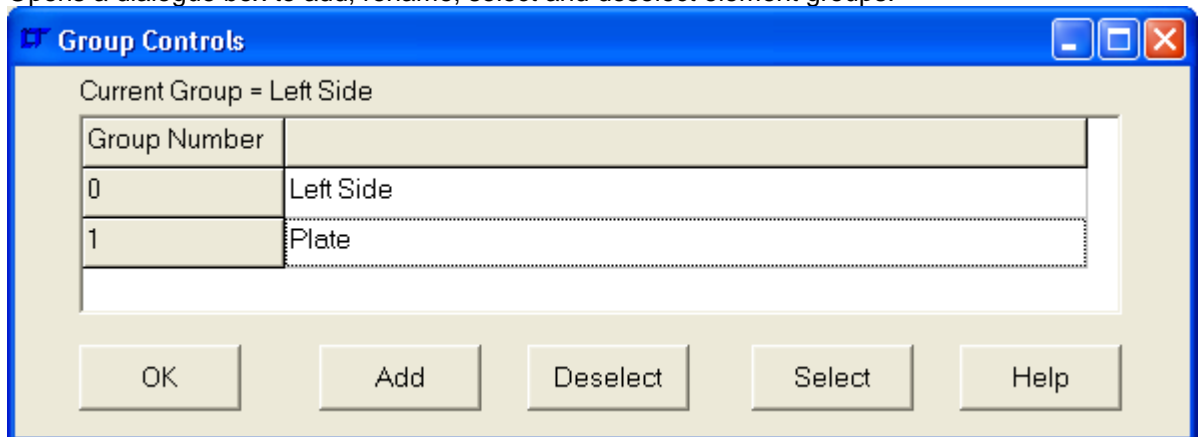
**'FE Model | Elements| Compact ID's'**

Opens a dialogue box requesting the starting number and increment for the element number compaction of the selected elements. The dialogue box is the same as shown in **'FE Model| Nodes| CompactID's'**.

**'FE Model | Elements| Groups'**

**'FE Model | Elements| Groups| Group Controls'**

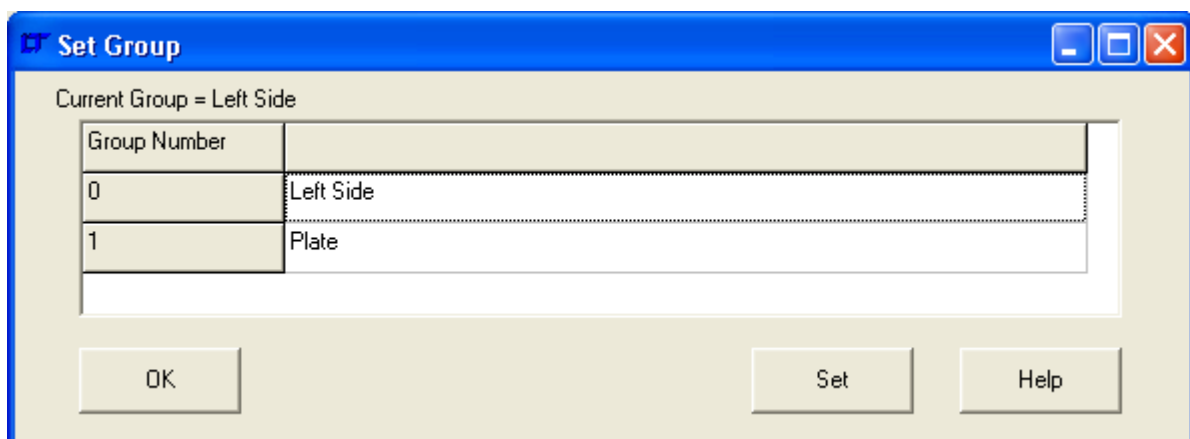
Opens a dialogue box to add, rename, select and deselect element groups.



Double click on the group name to make it the current group. Right click on the group name to get a pop-up menu to rename the group.

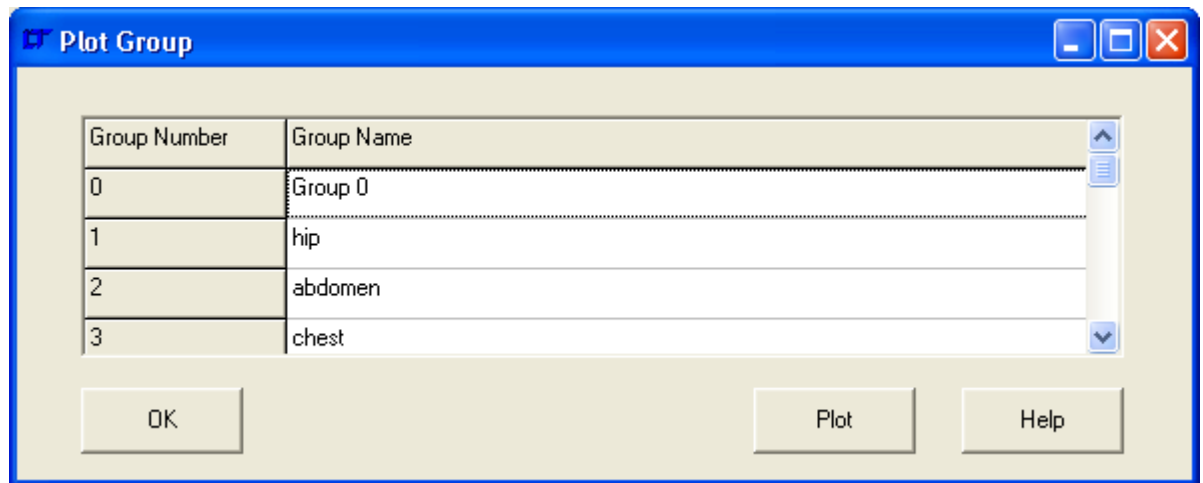
**'FE Model | Elements| Groups| Set Current Group'**

Opens a dialogue box to set the current group.

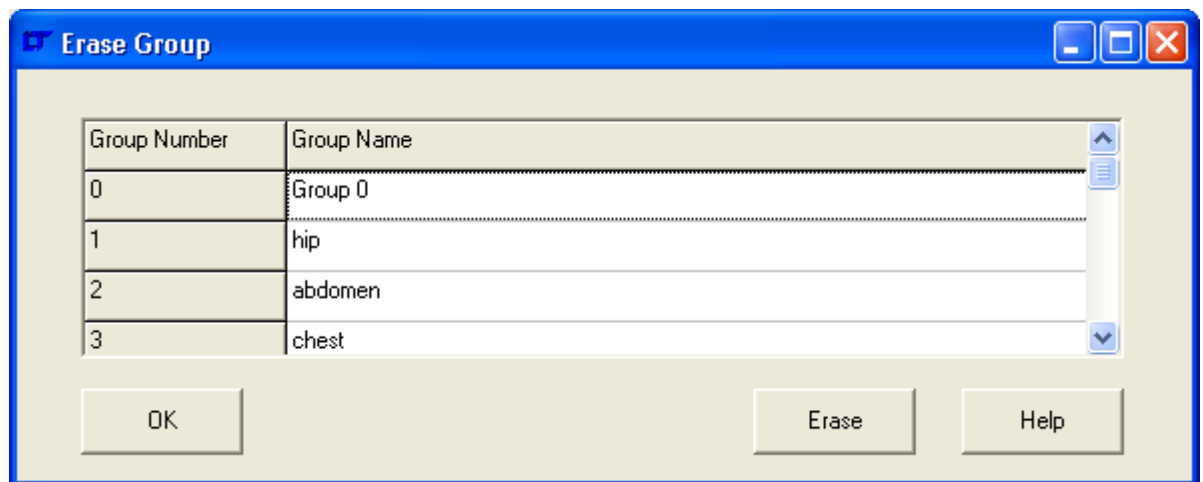


**'FE Model | Elements| Groups| Plot by Group Name'**

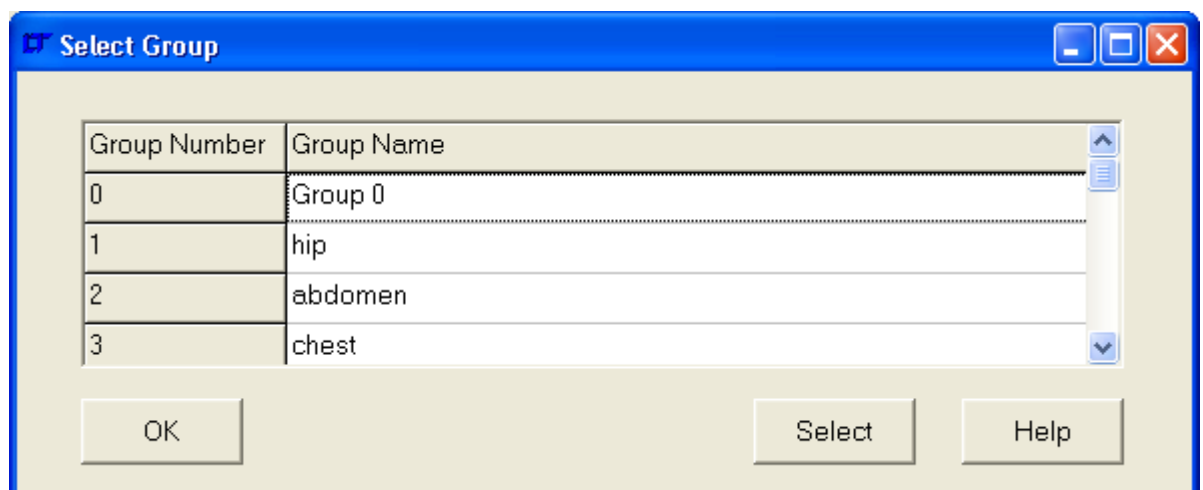
Opens a dialogue box to plot Groups by their name.

**'FE Model | Elements| Groups| Erase by Group Name'**

Opens a dialogue box to erase Groups by their name.

**'FE Model | Elements| Groups| Select by Group Name'**

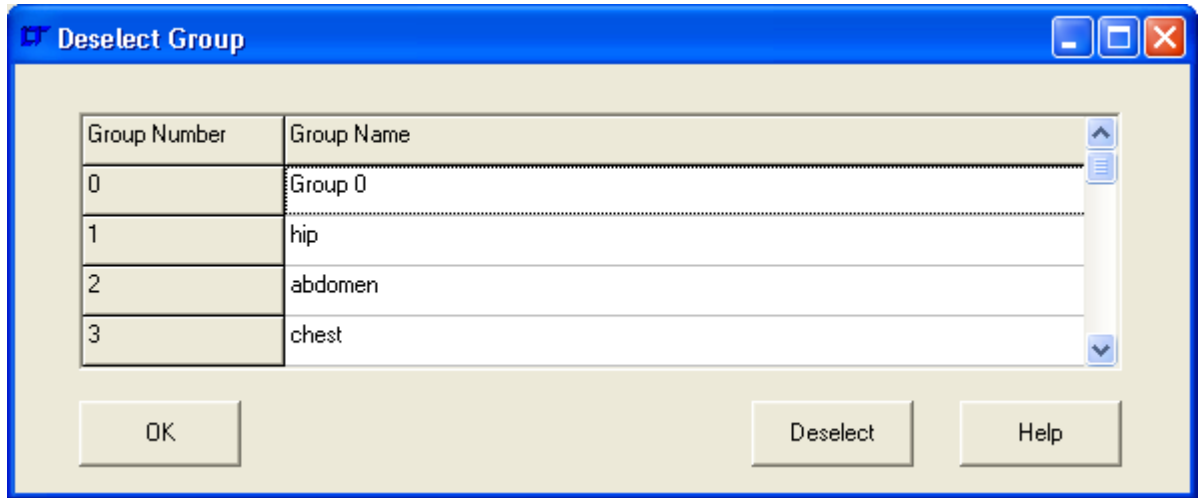
Opens a dialogue box to select Groups by their name.





**'FE Model | Elements| Groups| Deselect by Group Name'**

Opens a dialogue box to deselect Groups by their name.

**'FE Model | Elements| Groups| Add to Current Group'**

Adds the selected elements to the current group.

**'FE Model | Elements | Find'**

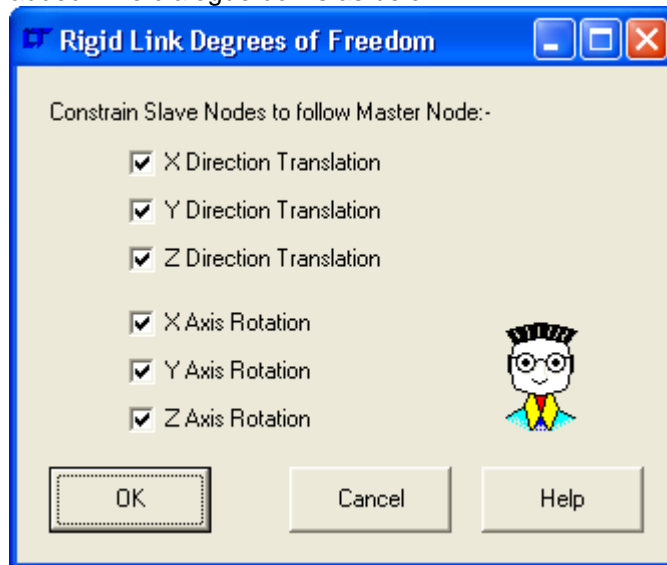
Find a range of elements. Typed input of the range is necessary in the command line. The elements will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'FE Model| Rigid Links'****'FE Model| Rigid Links| Add'****'FE Model| Rigid Links| Add| Pick Master Node Location'**

Adds a rigid link. The slave nodes are selected before choosing this menu option. The master node of the rigid link is picked by snap location or typed location in the command line. If a node is present at the pick point, it is used. Otherwise a new node is created at the pick point. The slave node degrees of freedom used are the ones defined by **'FE Model| Rigid Links| Add| Slave Node DOF'**. It is up to the user to ensure that no node is the slave of more than one Rigid Link. There are checks to prevent duplicate slavery, but it's probable that inventive users will be able to circumvent the checks.

**'FE Model| Rigid Links| Add| Slave Node DOF'**

Opens a dialogue box to set the slave node degrees of freedom when the next Rigid Link is added. The dialogue box is as below:-

**'FE Model| Rigid Links| Delete'****'FE Model| Rigid Links| Delete| All'**

Delete all Rigid Links.

**'FE Model| Rigid Links| Delete| Selected'**

Delete all selected Rigid Links.

**'FE Model | Rigid Links| Select'****'FE Model | Rigid Links| Select| All'**

Select all Rigid Links.

**'FE Model | Rigid Links| Select| None'**

Deselect all Rigid Links.

**'FE Model | Rigid Links| Select| Range'**

Select a range of Rigid Links. Typed input of the range is necessary in the command line.

**'FE Model | Rigid Links| Select| Invert Selection'**

Invert the selection of Rigid Links. All Rigid Links that are initially invisible are made visible as they are selected.

**'FE Model | Rigid Links| Select| Selected Nodes'**

Select Rigid Links by association with selected Nodes.

**'FE Model | Rigid Links| Select| Selected Elements'**

Select Rigid Links by association with selected Elements.

**'FE Model | Rigid Links| Inspect'**

Starts the interactive Rigid Link Inspection routine. Pick the master node, and the rigid link information will be printed in the **'Progress Information'** box.

**'FE Model | Rigid Links| Compact ID's'**

Opens a dialogue box requesting the starting number and increment for the rigid link number compaction of the selected rigid links. The dialogue box is the same as shown in **'FE Model| Nodes| CompactID's'**.

**'FE Model| Rigid Links| Change Slave DOF'**

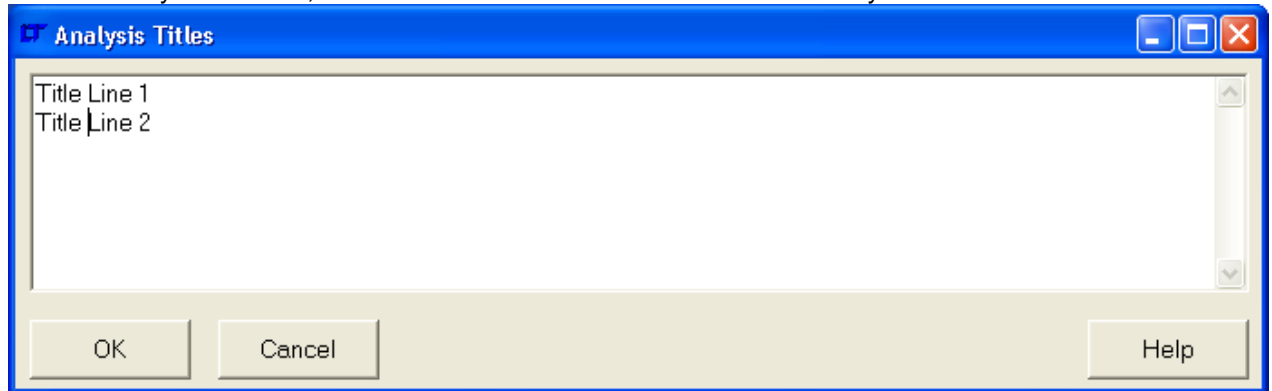
Opens a dialogue box to set the slave node degrees of freedom for the selected Rigid Links. The dialogue box is as below:-

**'FE Model | Rigid Links | Find'**

Find a range of rigid links. Typed input of the range is necessary in the command line. The rigid links will be plotted on top of the current screen view with their labels temporarily turned on, and made visible if otherwise erased.

**'FE Model| Analysis Title'**

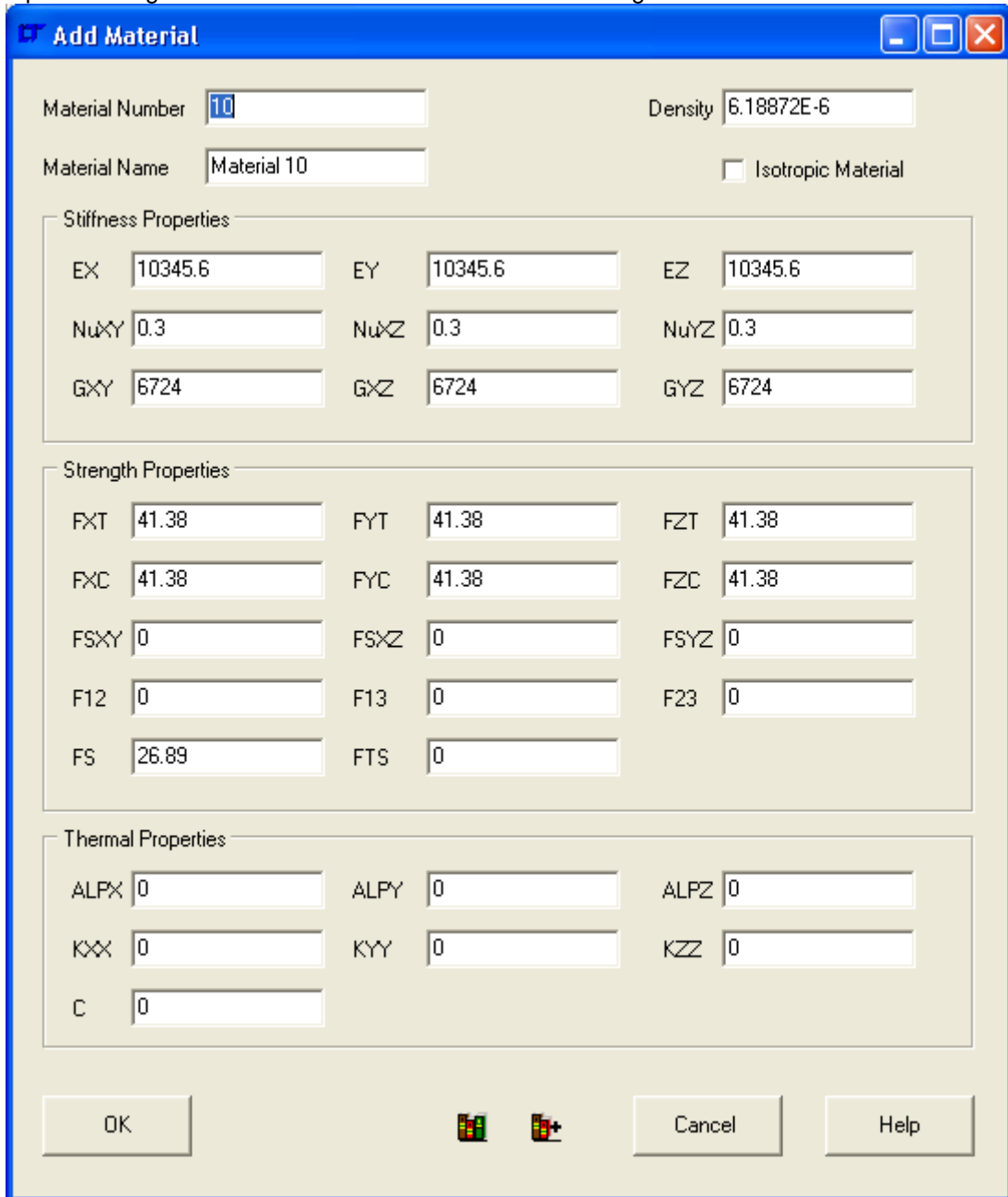
Opens a dialogue window where the titles of the model may be amended. For NISA users, a maximum of 6 lines may be entered, for MYSTRAN users a maximum of 2 lines may be entered.



**'FE Model| Material'**

**'FE Model| Material| Add'**

Open a dialogue window to add a new material. The dialogue window is as below:-



Material Number & Name

The material number is the number referred to by the element material number. The name is an aide-memoir for the user and is not used anywhere.

Density

The density specified for the material in units of mass/unit volume.

Isotropic Material

This specifies whether the material is isotropic or not. If the material is isotropic EY/EZ etc are greyed out and automatically filled when the X direction or XY plane figure is amended.

Material Data

The material data entries are as below:-

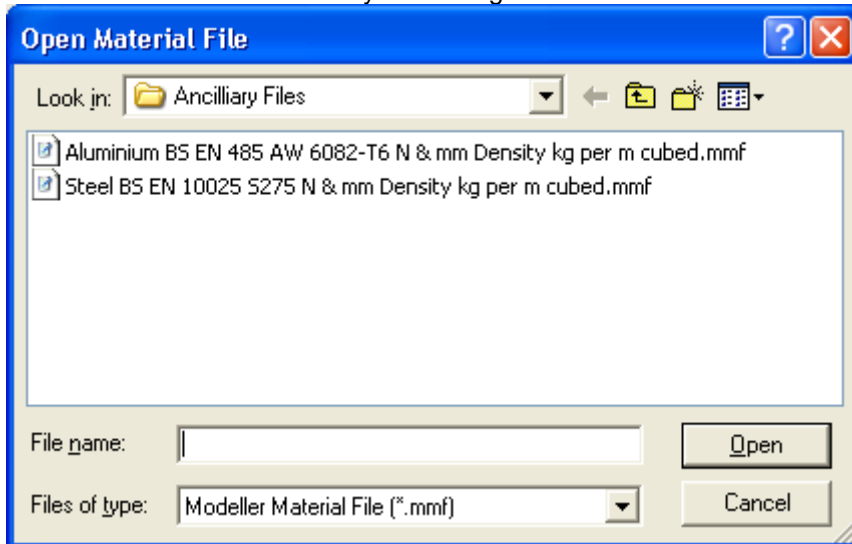
- E\* Modulus of Elasticity (force per area eg N/mm<sup>2</sup> ). Note for NISA Users: With Honeycomb/ sandwich panels (element type 33) the material defining the foam/ honeycomb core MUST have EX = 0.
- G\*\* Shear Modulus (force per area eg N/mm<sup>2</sup> )
- Nu\*\* Poisson's ratio (unitless)
- F\*T Tensile strength (force per area eg N/mm<sup>2</sup> )
- F\*C Compressive strength (force per area eg N/mm<sup>2</sup> )
- FS\*\* Shear strength (length per area eg N/mm<sup>2</sup> )
- F×× Principal axis shear strength (force per area eg N/mm<sup>2</sup> )
- FS, FTS Shear strengths. (force per area eg N/mm<sup>2</sup> )
- ALPA\* Co-efficient of thermal expansion (per temperature rise eg /°C)
- K\* Thermal conductivity (heat flux per length eg W/m)
- C Specific Heat (energy per mass eg J/kg)

Where

- \* Local element axis (X, Y or Z).
- \*\* Local element plane (XY, XZ or YZ plane).
- ×× Principal axis (12, 13, 23).

Load From Library Button

Load a material from the library of existing materials. As below:-

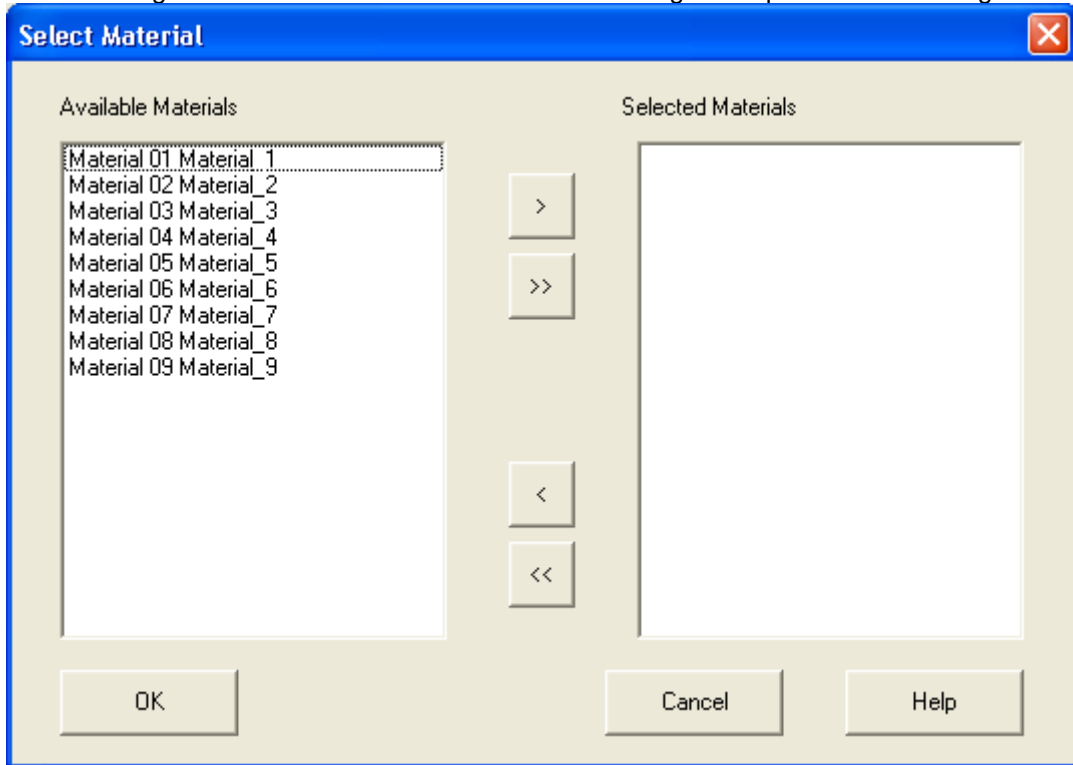


Add Material To Library

Save a commonly used material to the library of existing materials.

**'FE Model| Material| Delete'**

Delete a range of material numbers. Selection of the range is required from a dialogue box.

**'FE Model| Material| Delete All'**

Delete all materials.

**'FE Model| Material| Inspect'**

Open a dialogue box similar to the **'FE Model| Material| Add'** dialogue box, with the capability to scroll through the materials in the model as well as add and delete materials.

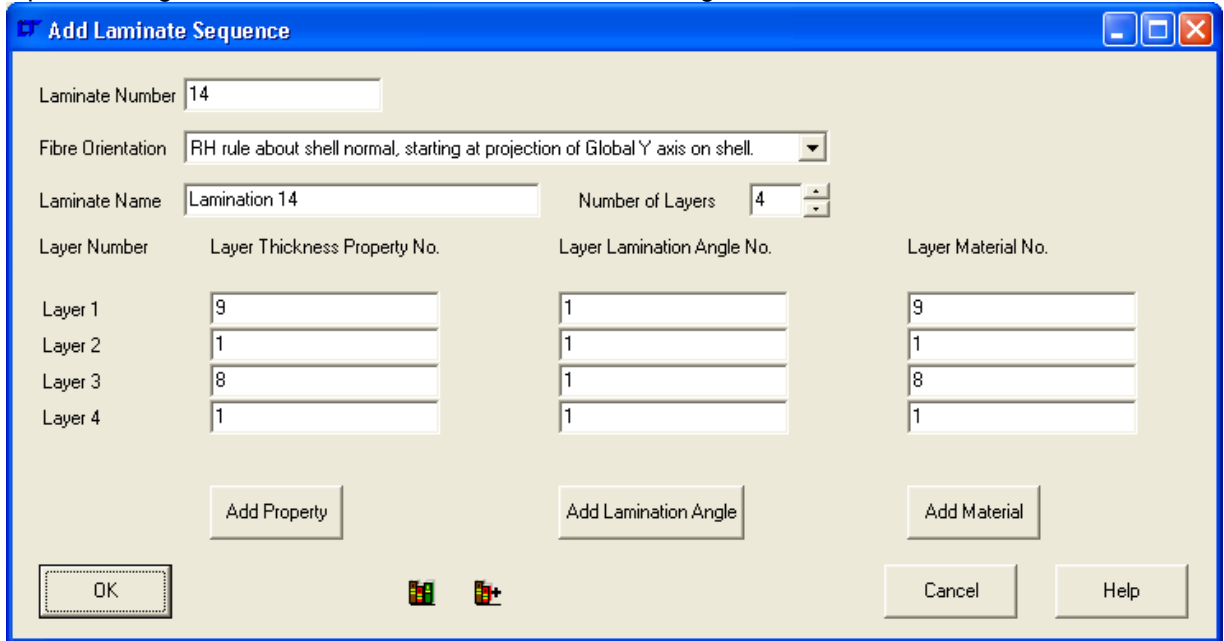
**'FE Model| Material| Change Model Density'**

Open a dialogue box to permit all material densities to be multiplied by a factor entered by the user. If a nonsense number is entered, the material density is multiplied by 1.0.

**'FE Model| Laminations'**

**'FE Model| Laminations| Add'**

Open a dialogue window to add a new Laminate. The dialogue window is as below:-



Laminate Number & Name

The Laminate number is the number referred to by the element material number for elements of type 32 & 33 (Laminate & Sandwich elements). The name is an aide-memoir for the user and is not used anywhere.

Fibre Orientation

The 0° axis for the fibre orientation is defined here. The 0° axis is in element local axes or global axes.

If any of the following definitions are used, then the element must not be exactly in the plane formed by the two axes mentioned or the NISA analysis will abort without any indication partway through the stiffness calculation.

- RH Rule about Global Z axis, starting at global X axis.
- RH Rule about Global X axis, starting at global Y axis.
- RH Rule about Global Y axis, starting at global Z axis.

If any of the following are used, then the element must not be exactly perpendicular to the specified global axis or the NISA analysis will abort without any indication partway through the stiffness calculation.

- RH Rule about shell normal starting at projection of Global X axis on shell.
- RH Rule about shell normal starting at projection of Global Y axis on shell.
- RH Rule about shell normal starting at projection of Global Z axis on shell.

Problems caused by fibre orientations are very difficult to trace as NISA aborts without any message to indicate a problem. Get them right first time!

See the 'User Guide' on 'How to Create Laminates' for further advice.

Number of Layers

Use the up & down arrows to change how many layers there are in the laminate. NISA can accept a maximum of 50, though there is no such limitation for the Modelling Studio.

Layer Data

In the next section enter the property number for each layer in order to define the thickness of that layer. Also define the fibre orientation angle set, as each layer may be rotated relative to the 0° fibre orientation defined for the element. Finally, specify the material number for each layer.



Add Property/ Lamination Angle/ Material Buttons

The buttons enable the user to add properties, Lamination Angles or Materials without using the normal Menu commands of :-

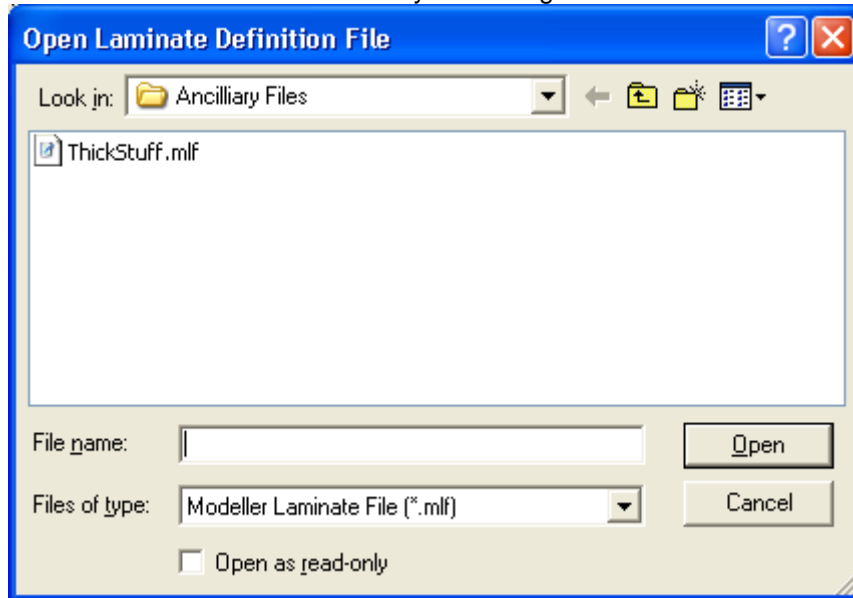
**'FE Model| Property| Add'**

**'FE Model| Lamination Angles| Add'**

**'FE Model| Material| Add'**

Load From Library Button

Load a Laminations from the library of existing Laminations. As below:-



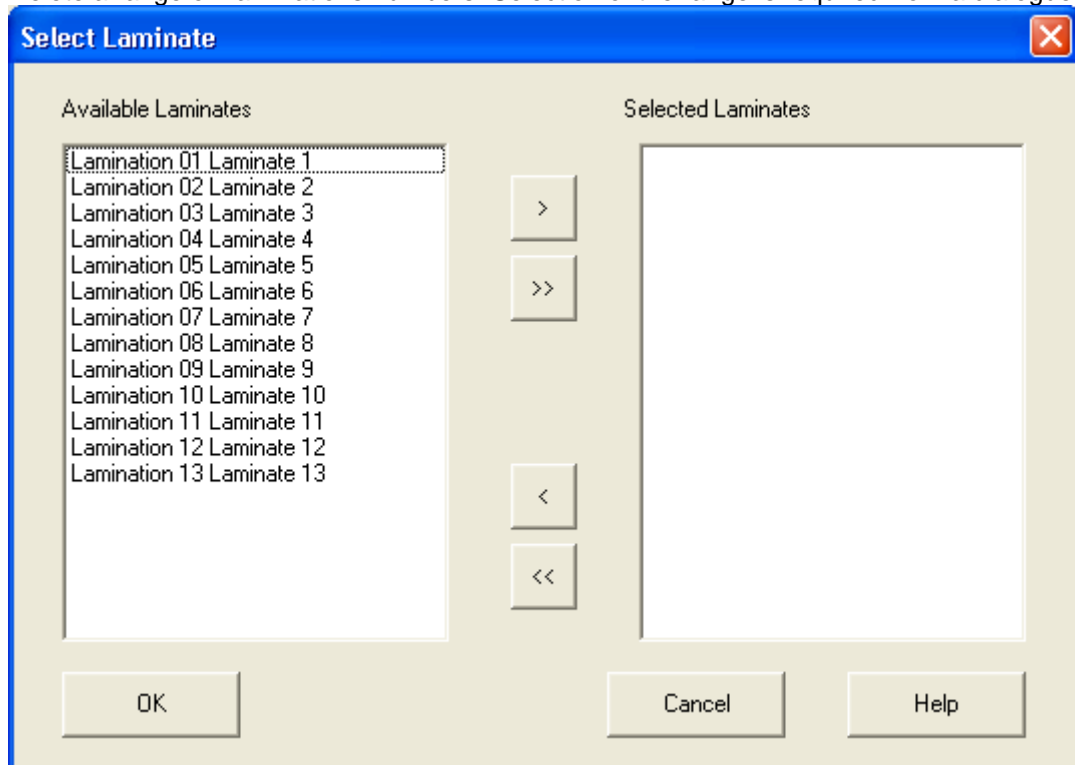
Note:- Each Laminate file contains the Property, Material and Angle data within the file so that a consistent laminate may be loaded regardless of existing properties.

Add Laminations To Library

Save a commonly used Laminations to the library of existing Laminations.

**'FE Model| Laminations| Delete'**

Delete a range of Laminations numbers. Selection of the range is required from a dialogue box.

**'FE Model| Laminations| Delete All'**

Delete all Laminations.

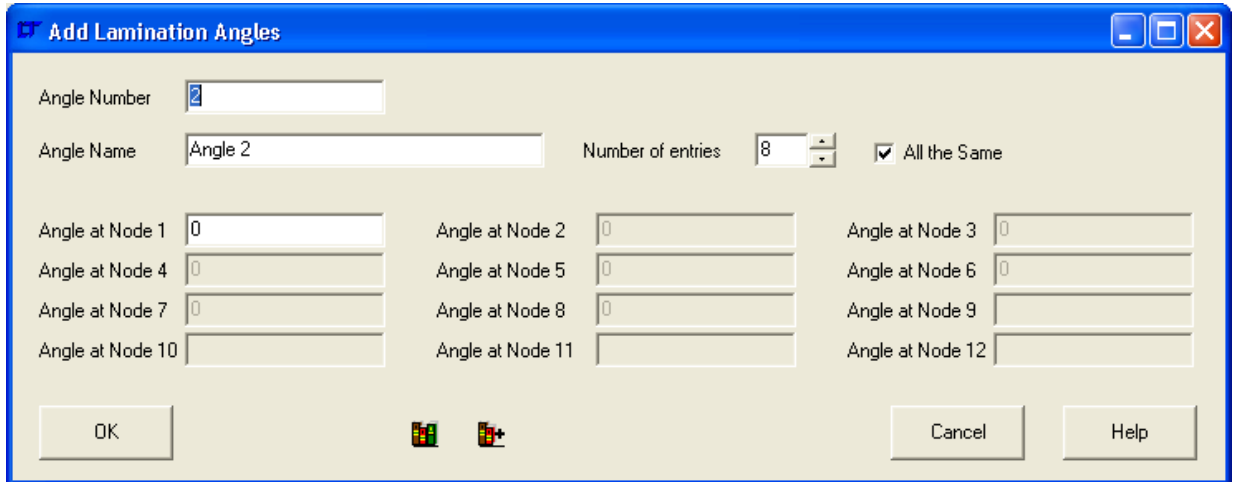
**'FE Model| Laminations| Inspect'**

Open a dialogue box similar to the **'FE Model| Laminations| Add'** dialogue box, with the capability to scroll through the Laminations in the model as well as add and delete Laminations.

**'FE Model| Lamination Angles'**

**'FE Model| Lamination Angles| Add'**

Open a dialogue window to add a new Lamination Angle Set. The dialogue window is as below:-



Angle Number & Name

The Angle number is the number referred to by the Lamination data set for elements of type 32 & 33 (Laminate & Sandwich elements). The name is an aide-memoir for the user and is not used anywhere.

Number of Entries & All the same

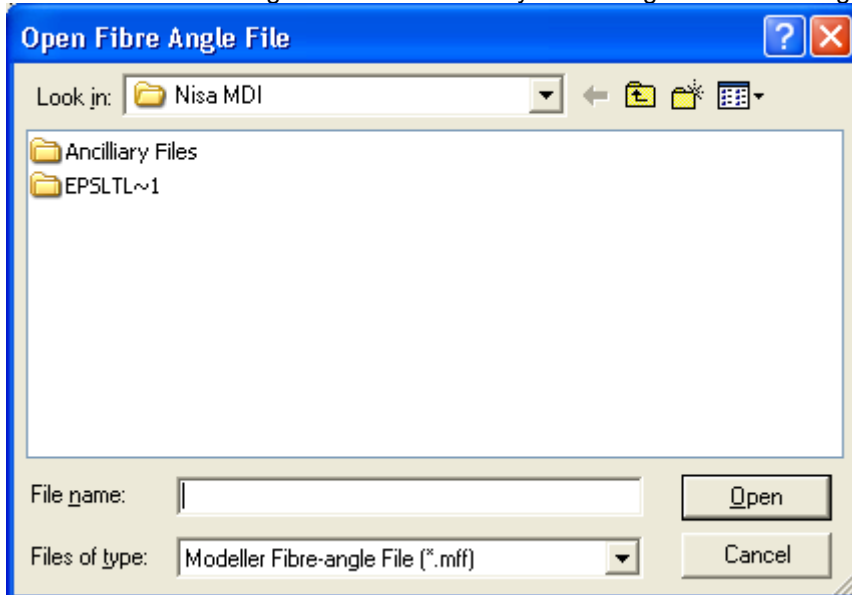
Use the up & down arrows to change how many entries there are. There must be one for each node. If all the entries are the same, tick the 'All the Same' checkbox and only enter one piece of data

Angle Data

Specify the angle at each node.

Load From Library Button

Load a Lamination Angle set from the library of existing Lamination Angles. As below:-

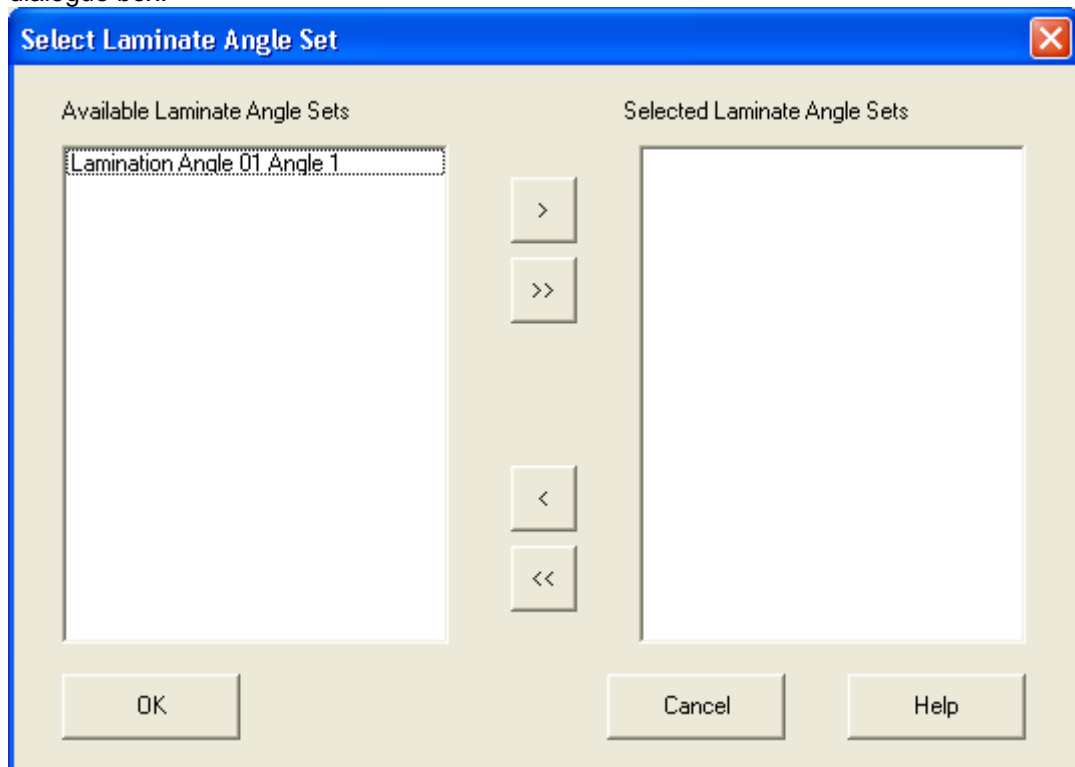


Add Lamination Angle Set To Library

Save a commonly used Lamination Angles to the library of existing Lamination Angles.

**'FE Model| Lamination Angles| Delete'**

Delete a range of Lamination Angle set numbers. Selection of the range is required from a dialogue box.

**'FE Model| Lamination Angles| Delete All'**

Delete all Lamination Angles.

**'FE Model| Lamination Angles| Inspect'**

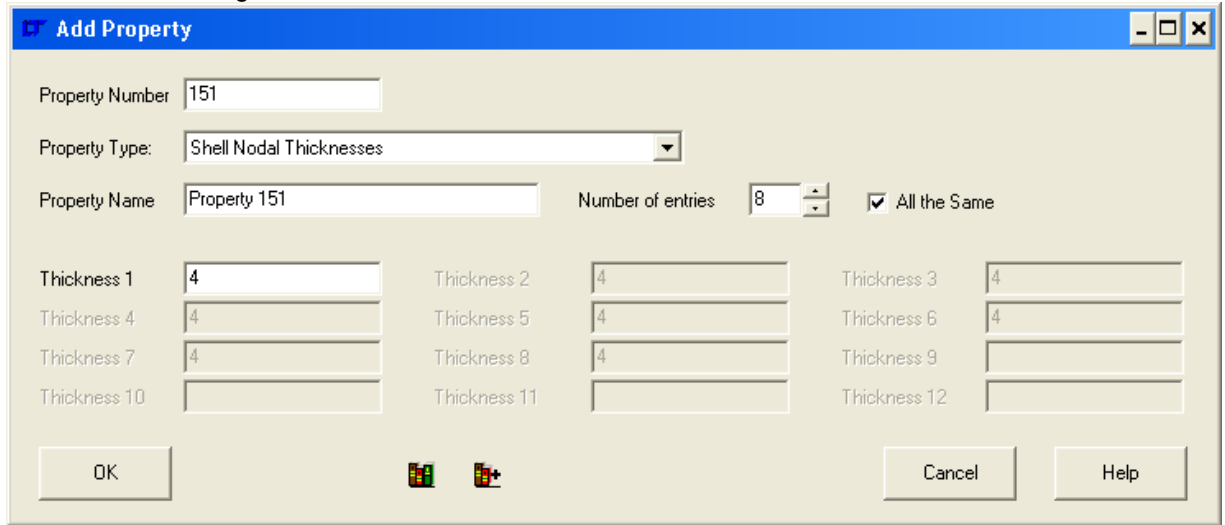
Open a dialogue box similar to the **'FE Model| Lamination Angles| Add'** dialogue box, with the capability to scroll through the Lamination Angles in the model as well as add and delete Lamination Angles.

**'FE Model| Properties'**

**'FE Model| Properties| Add'**

Open a dialogue window to add a new Property Data Set. The dialogue window changes dependant on the type of property that is being specified.

For Shells the dialogue box is as below:-



Property Number & Name

The property number is the number referred to by the Lamination data set for elements of type 32 & 33 (Laminate & Sandwich elements) or the 'Property' data of every element. The name is an aide-memoir for the user and is not used anywhere.

Number of Entries & All the same

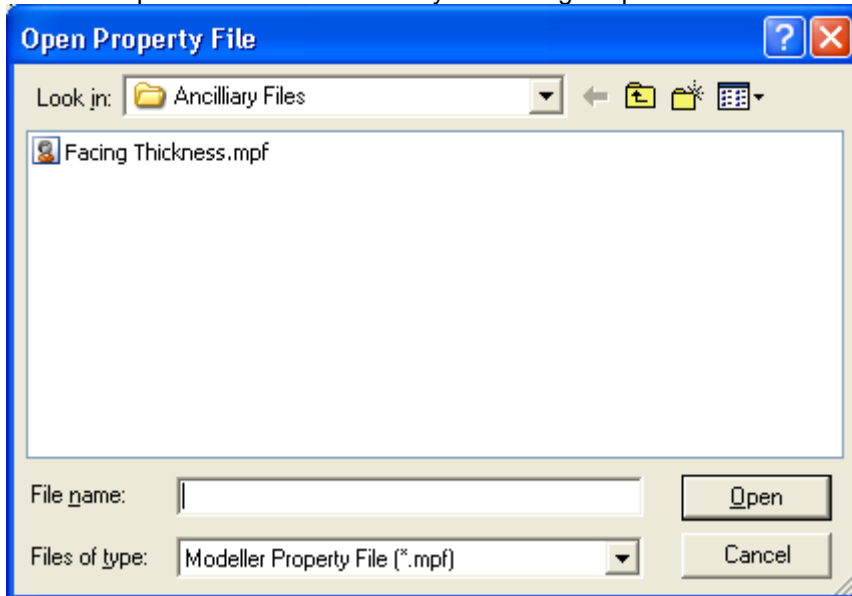
Use the up & down arrows to change how many entries there are. There must be one for each node (unless using MYSTRAN, when only one thickness needs to be specified). If all the entries are the same, tick the 'All the Same' checkbox and only enter one piece of data

Thickness Data

Specify the thickness at each node.

Load From Library Button

Load a Properties set from the library of existing Properties. As below:-



Add Property Set To Library

Save a commonly used Property to the library of existing Properties.

For 3D Beam elements the properties are:-

Assume the following Local Axes for the beam  
 X along the length of the beam  
 Y across the beam  
 Z perpendicular to X & Y axes.

The properties to enter are:-

- XSect Area      Cross sectional area of beam in YZ plane.
- Iyy              Second Moment of area about Y axis.
- Izz              Second Moment of area about Z axis
- J                 Torsional constant. If left blank = Iyy + Izz
- Ey & Ez         Offsets of beam centroid from line of element
- Dy & Dz         Depth of beam in Y & Z axes.
- Alpha Y & Z     Shear co-efficients in Y & Z axes.  
 Solid Circular section = 1.1111  
 Solid Rectangular Section = 1.2  
 Circular Tube = 1.887  
 Rectangular tube = 2.0  
 I Beam = Total Area/ Web area
- V1x,y,z         Offset distances for end 1 of beam from the node that defines the beam.
- V2x,y,z         Offset distances for end 2 of beam from the node that defines the beam.
- Okey, Theta/ X3, Y3 & Z3  
 If Okey = 0, then orientate the beam local xy plane in the global Z direction (or global Y direction if vertical) plus angle theta about the element local x axis using the right hand rule. Y3 & Z3 can be left blank.  
 If Okey = 1, then X3, Y3 & Z3 define a third point on the element local xy plane.

SKey	Number of points on cross section to calculate stresses. May be 0 to 4.
TKey	Section type = 0 for solid or thick walled section = 1 for thin walled open section = 2 for thin walled closed section.
For each stress point used the following must be entered:-	
YS	Y co-ordinate of stress point
ZS	Z co-ordinate of stress point.
TSF	Torsional Shear Factor = 0 for solid or thick walled sections. = wall thickness at stress point for thin open sections = $1/(2A_s t)$ for thin closed sections. Where $A_s$ = area enclosed by wall centreline and $t$ = wall thickness at stress point.
SBFY	Shear bending factor about Y axis – NISA only – see NISA manual.
SBFZ	Shear bending factor about Z axis – NISA only – see NISA manual.

For 2D Beam Elements:-

The screenshot shows the 'Add Property' dialog box with the following values entered:

- Property Number: 151
- Property Type: 2D Beam Section Properties
- Property Name: 50 x 25 x 3.2mm Box
- XSect Area: 434
- lyz: 128000
- Shear Coeff.: 2
- Local Y Stress Recovery Co-ordinate: 1

The properties to enter are:-

XSect Area	Cross sectional area of beam in YZ plane.
lyy	Second Moment of area about Y axis.
Shear Co-eff	Shear co-efficient. Solid Circular section = 1.1111 Solid Rectangular Section = 1.2 Circular Tube = 1.887 Rectangular tube = 2.0 I Beam = Total Area/ Web area
Local Y Stress recovery Co-ordinate	Set to 1.

For 2D Translational Springs

The properties to enter are:-  
 Spring Stiffness                      Stiffness of the spring element

For 3D Translational Springs

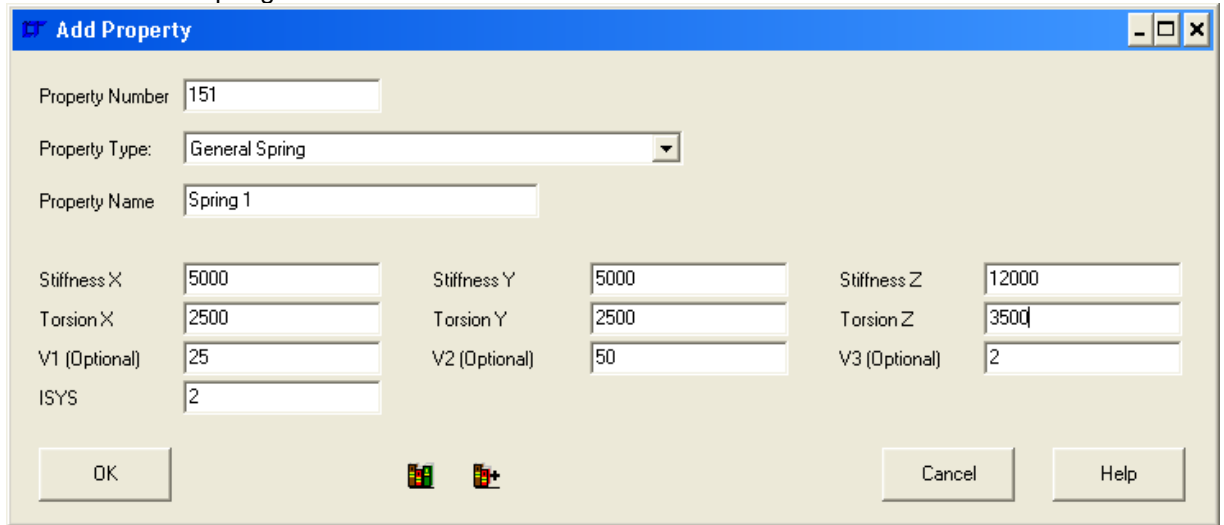
The properties to enter are:-  
 Spring Stiffness                      Stiffness of the spring element

For Torsional Springs

The properties to enter are:-  
 Spring Stiffness                      Stiffness of the spring element



For 3D General Springs



The properties to enter are:-

Stiffness X, Y, Z

Stiffness of the spring element in the X, Y & Z axes. Note to NISA users. While these axes are supposed to be local, NISA V9, 10, 11 & 12 used these as the element stiffness in global axes regardless of the actual element orientation.

Torsion X, Y, Z

Torsional Stiffness of the spring element about the X, Y & Z axes

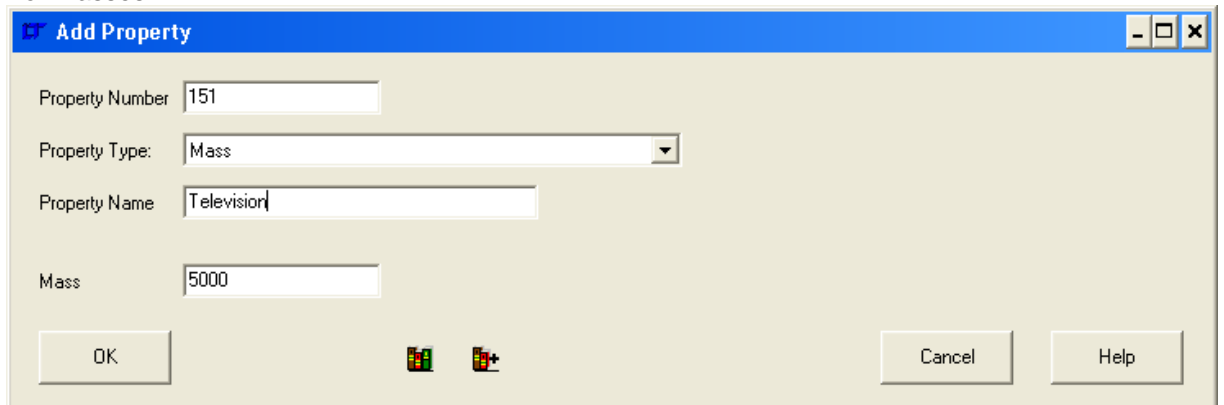
V1, V2, V3

Spring axis if end nodes are co-incident [Don't do it! This is retained for NISA import compatibility]

ISYS

Co-ordinate system to use for V1, V2 & V3.  
=0 for global Cartesian  
=1 for LCS at element node 1.  
=2 for LCS at element node 2.  
You are urged to leave this blank.

For Masses



The properties to enter are:-

Mass

Mass of the mass element referencing this property.

For 2D Mass & Inertia elements. Only for NISA.

The properties to enter are:-

Mass X, Y  
Inertia

Mass of the mass element referencing this property.  
Rotational Inertia of the mass element.

For 3D Mass & Inertia elements. Only for NISA.

The properties to enter are:-

Mass X, Y, Z  
Inertia X, Y, Z

Mass of the mass element referencing this property.  
Rotational Inertia of the mass element.

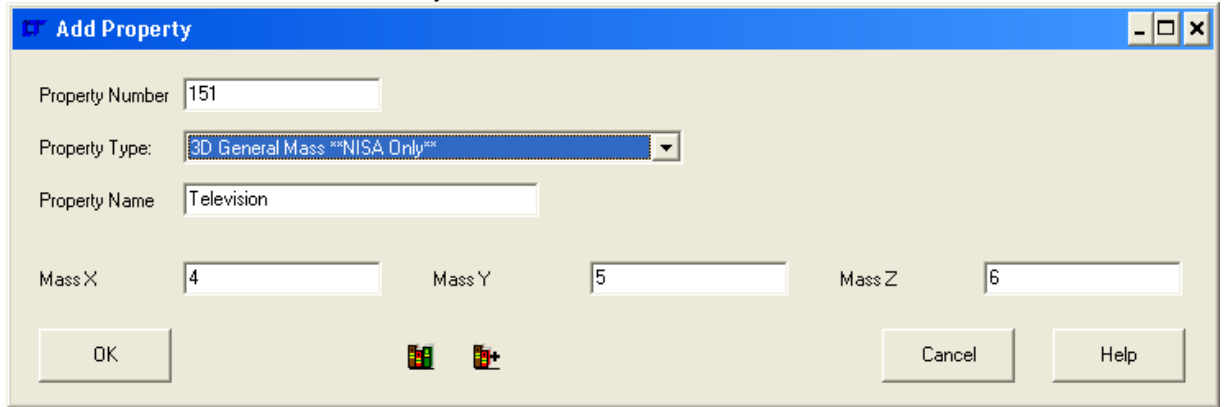
For 2D General Mass elements. Only for NISA.

The properties to enter are:-

Mass X, Y

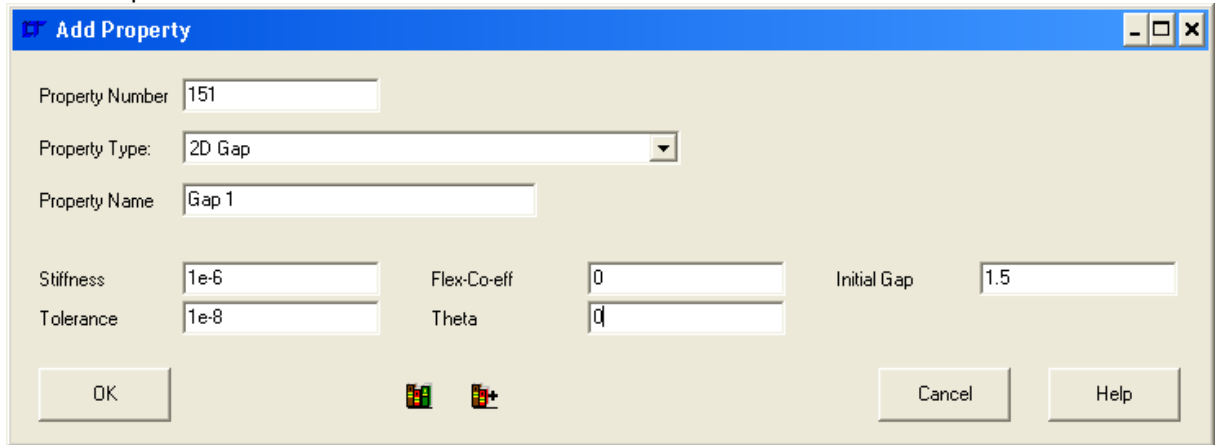
Mass of the mass element referencing this property.

For 3D General Mass elements. Only for NISA.



The properties to enter are:-  
 Mass X, Y, Z                      Mass of the mass element referencing this property.

For 2D Gap elements.



The properties to enter are:-

Stiffness	Stiffness for rigid body motion
Flex Co-eff	=0 for gap elements >0 = tension only element <0 = compression only element
Initial Gap	The initial gap width. +ve = initially open. -ve = initial interference. Note: opening is not defined by position of nodes.
Tolerance	Convergence tolerance on gap width
Theta	Gap orientation from X axis.

For 3D Gap elements.

The screenshot shows the 'Add Property' dialog box for a 3D Gap element. The fields are as follows:

- Property Number: 151
- Property Type: 3D Gap
- Property Name: Gap 1
- Stiffness: 1e-6
- Flex-Co-eff: 0
- Initial Gap: 1.5
- Tolerance: 1e-8

The properties to enter are:-

Stiffness	Stiffness for rigid body motion =0 for gap elements
Flex Co-eff	>0 = tension only element <0 = compression only element
Initial Gap	The initial gap width. +ve = initially open. -ve = initial interference. Note: opening is not defined by position of nodes.
Tolerance	Convergence tolerance on gap width. 1e-8 is a good default.

For 2D Spar elements.

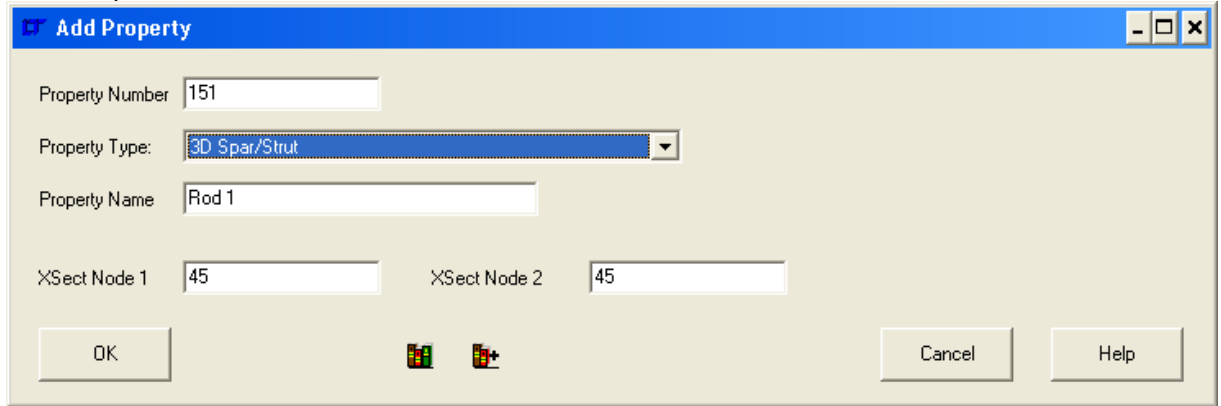
The screenshot shows the 'Add Property' dialog box for a 2D Spar element. The fields are as follows:

- Property Number: 151
- Property Type: 2D Spar
- Property Name: Rod 1
- XSect Node 1: 45
- XSect Node 2: 45

The properties to enter are:-

XSect at Node 1,2	Cross sectional area of strut
-------------------	-------------------------------

For 3D Spar /Strut elements.



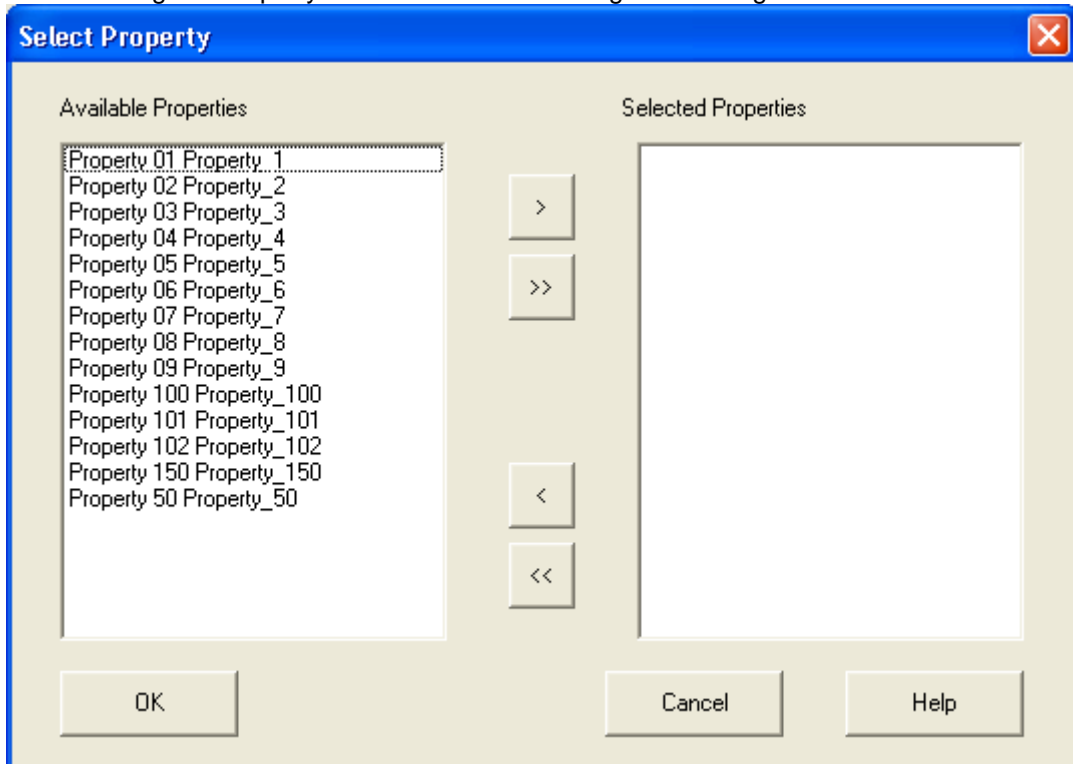
The properties to enter are:-

XSect at Node 1,2      Cross sectional area of strut

All remaining properties are for NISA only elements, and the user can look up the NISA manual for the property definitions.

**'FE Model| Properties| Delete'**

Delete a range of Property set numbers. Select range in a dialogue window.



**'FE Model| Properties| Delete All'**

Delete all Properties.

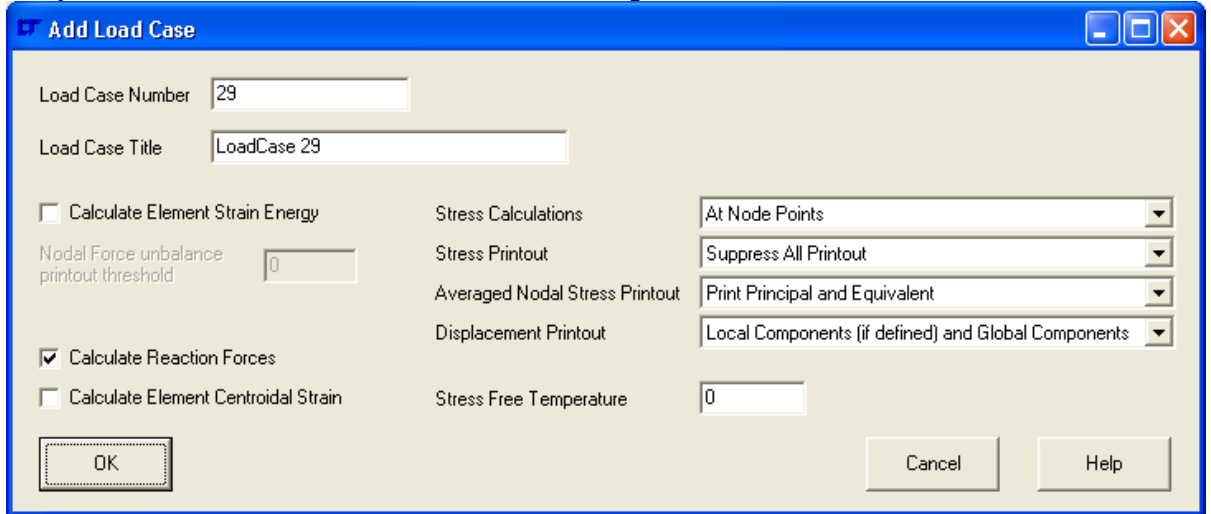
**'FE Model| Properties| Inspect'**

Open a dialogue box similar to the **'FE Model| Properties| Add'** dialogue box, with the capability to scroll through the Properties in the model as well as add and delete Properties.

**'FE Model| Load Case'**

**'FE Model| Load Case | Add'**

Open a dialogue window to add a new LoadCase Set. A load case will not be used in the analysis file if no Load Case set is defined. The dialogue window is as below:-



Load Case number

The load case number is the number referred to by all displacement constraints, pressures, nodal pressures, forces and accelerations that are applied to the model in this load case.

Load Case Title

The name of the load case. This name is passed through to the analysis package.

Calculate Element Strain Energy [applicable to NISA only]

When ticked, requires the strain energy to be calculated. When ticked, the Nodal Force unbalance threshold becomes active. Specify a suitable unbalance.

Stress Calculations

Specifies where stress calculations are carried out.

Stress Printout

Specifies what stress results are printed.

Averages Nodal Stress Printout

Specifies what averaged nodal stress results are printed.

Displacement Printout

Specifies what displacement results are printed.

Calculate Reaction Forces

Requires the analysis package to calculate reaction forces if ticked.

Calculate Element Centroidal Strain

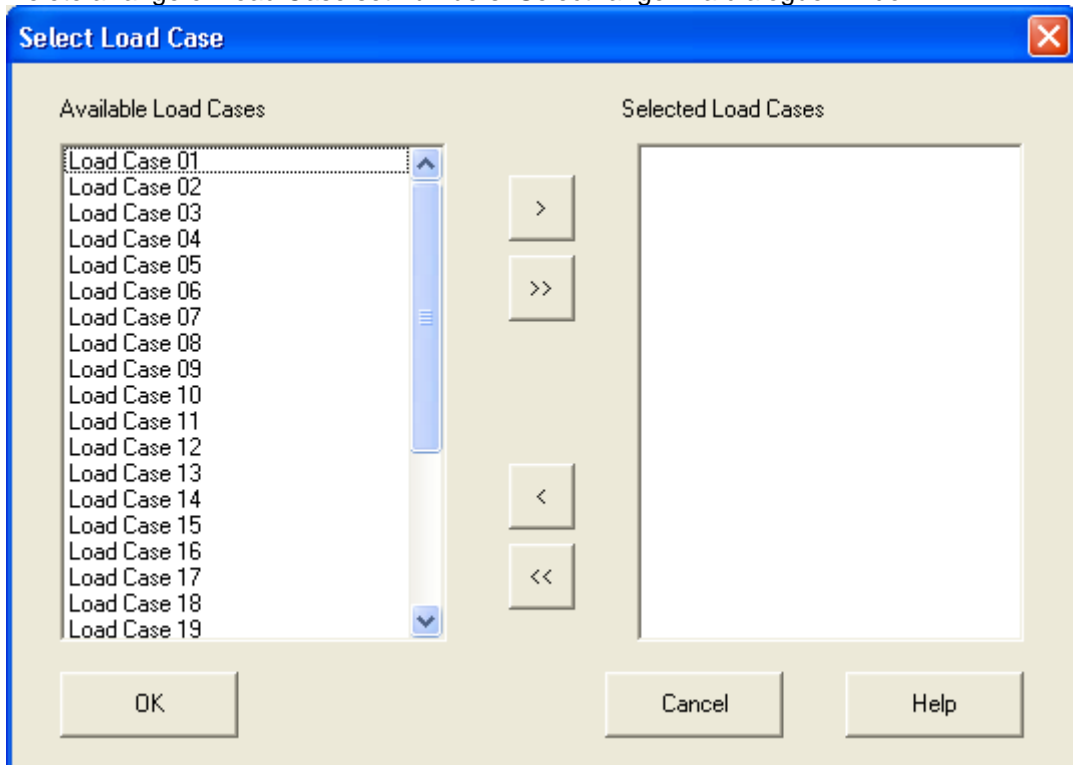
Requires the analysis package to calculate Element Centroidal Strain if ticked.

Stress Free Temperature

Specifies the temperature at which the model is stress free.

**'FE Model| Load Cases| Delete'**

Delete a range of Load Case set numbers. Select range in a dialogue window:-



Note: this command does not delete any Displacement Constraints, Pressures, Nodal Pressures, Forces or Accelerations – these have to be deleted separately.

**'FE Model| Load Cases| Delete All'**

Delete all Load Case Sets. Note: this command does not delete any Displacement Constraints, Pressures, Nodal Pressures, Forces or Accelerations.

**'FE Model| Load Cases| Inspect'**

Open a dialogue box similar to the **'FE Model| Load Cases| Add'** dialogue box, with the capability to scroll through the Load Cases in the model as well as add and delete Load Cases.

**'FE Model| Load Cases| Auto Create'**

Toggles the load case auto create setting. If a displacement constraint, pressure, nodal pressure, force or acceleration is added when **'FE Model| Load Cases| Auto Create'** is checked, then a load case will be automatically added using the settings from **'FE Model| Load Cases| Auto Create Settings'**

**'FE Model| Load Cases| Auto Create Settings'**

Opens a dialogue window to specify the load case settings to be used when a load case is automatically added. The dialogue box is as below:-

Settings for Automatically Created Load Cases

Load Case Number: 0

Load Case Title:

Calculate Element Strain Energy

Nodal Force unbalance printout threshold: 0

Calculate Reaction Forces

Calculate Element Centroidal Strain

Stress Calculations: At Node Points

Stress Printout: Suppress All Printout

Averaged Nodal Stress Printout: Print Principal and Equivalent

Displacement Printout: Local Components (if defined) and Global Components

Stress Free Temperature: 0

OK Cancel Help

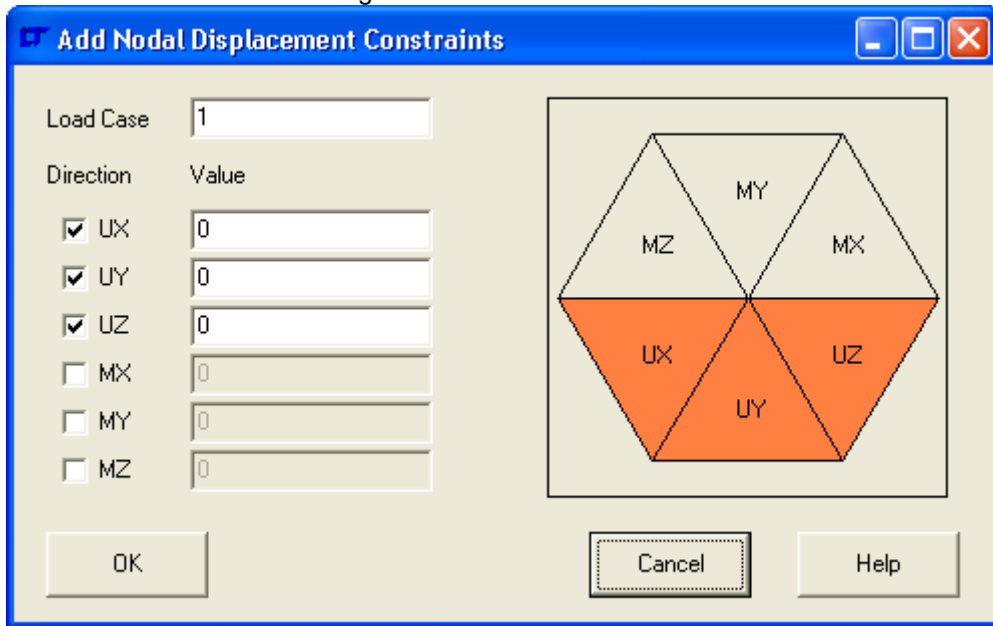
The Load Case Number and Title are blank as they will be filled automatically.



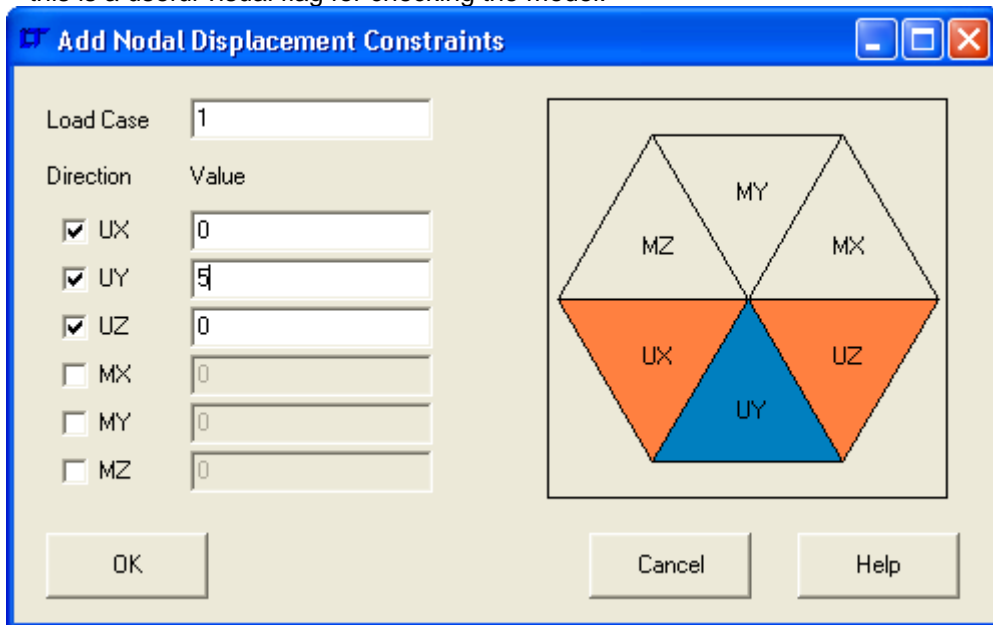
**'FE Model| Displacement Constraints'**

**'FE Model| Displacement Constraints| Add'**

Open a dialogue window to add new displacement constraints to the currently selected nodes. A new load case set will be defined if the load case does not exist and **'FE Model| Load Cases| Auto Create'** is set. The dialogue window is as below:-



To set a constraint, either click the tickbox at the left or the appropriate segment of the hexagon. When a deflection value is non-zero, the segment is highlighted in a different colour (as below) – this is a useful visual flag for checking the model.



Where a displacement constraint already exists for a node for the specified load case, the displacement constraint is over-written.

**'FE Model| Displacement Constraints| Delete'**

**'FE Model| Displacement Constraints| Delete| All'**

Delete all Displacement Constraints.

**'FE Model| Displacement Constraints| Delete| Selected'**

Delete all selected Displacement Constraints.

**'FE Model| Displacement Constraints| Select'**

**'FE Model| Displacement Constraints| Select| All'**

Select all Displacement Constraints. Any displacement constraints initially invisible will be made visible.

**'FE Model| Displacement Constraints| Select| None'**

Deselect all Displacement Constraints.

**'FE Model| Displacement Constraints| Select| Invert Selection'**

Invert the selection of all Displacement Constraints. Any displacement constraints initially invisible will be made visible.

**'FE Model| Displacement Constraints| Select| by Load Case'**

Select all displacement constraints in a range of load cases. Typed input of the range is necessary in the command line. Any displacement constraints initially invisible will be made visible.

**'FE Model| Displacement Constraints| Select| by Node Number'**

Select all displacement constraints applied to a range of node numbers. Typed input of the range is necessary in the command line. Any displacement constraints initially invisible will be made visible.

**'FE Model| Displacement Constraints| Select| Selected Nodes'**

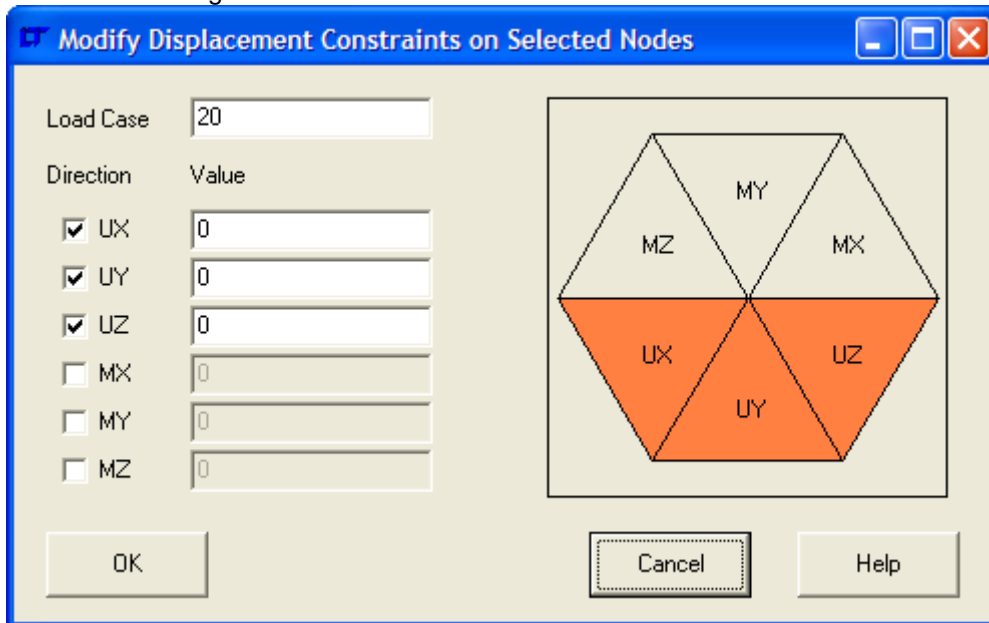
Select all visible displacement constraints associated with the currently selected nodes.

**'FE Model| Displacement Constraints| Select| Selected Elements'**

Select all visible displacement constraints associated with the currently selected elements.

**'FE Model| Displacement Constraints|Modify'**

Replaces the displacement constraints on the selected nodes for the identified load case with those in the dialogue box.



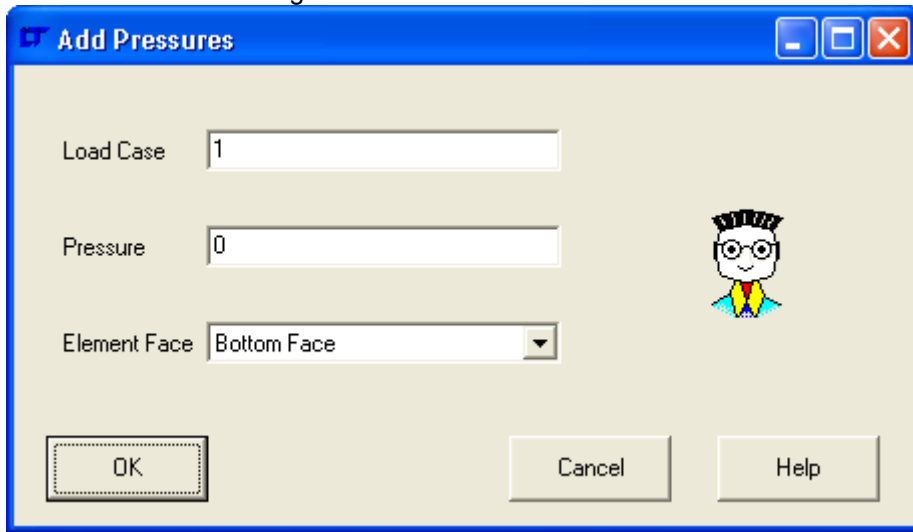
If the node does not have an existing displacement constraint, the modify command will be ignored.

**'FE Model| Displacement Constraints| Inspect'**

Starts the interactive displacement constraints inspection routine. The mouse snaps to nodes with visible displacement constraints, and all the appropriate information is listed in the 'Progress Information' box.

**'FE Model| Pressures'****'FE Model| Pressures| Add'**

Open a dialogue window to add new Pressures to the currently selected elements. A new load case set will be defined if the load case does not exist and **'FE Model| Load Cases| Auto Create'** is set. The dialogue window is as below:-



Where a Pressure already exists for a face for the specified load case, the Pressure is overwritten. When elements other than shells are selected the Element Face selection combo box switches to a range of 1 to 6 for the face number. (1 = bottom, 2 = top, 3 = front, 4 = right, 5 = back & 6 = left).

**'FE Model| Pressures| Delete'****'FE Model| Pressures| Delete| All'**

Delete all Pressures.

**'FE Model| Pressures| Delete| Selected'**

Delete all selected Pressures.

**'FE Model| Pressures| Select'****'FE Model| Pressures| Select| All'**

Select all Pressures. Any Pressures initially invisible will be made visible.

**'FE Model| Pressures| Select| None'**

Deselect all Pressures.

**'FE Model| Pressures| Select| Invert Selection'**

Invert the selection of all Pressures. Any Pressures initially invisible will be made visible.

**'FE Model| Pressures| Select| Range by Element Association'**

Select all Pressures associated with selected elements.

**'FE Model| Pressures| Select| Range by Face Association'**

Select Pressures on a face associated with selected elements. Typed input of the face number is necessary in the command line.

**'FE Model| Pressures| Select| by Load Case'**

Select all Pressures within a range of Load Cases. Typed input of the range is necessary in the command line.

**'FE Model| Pressures| Select| Selected Nodes'**

Select all Pressures that are visible associated with the selected nodes.

**'FE Model| Pressures| Select| Selected Elements'**

Select all Pressures that are visible associated with the selected elements.

**'FE Model| Pressures| Select| Zero Value'**

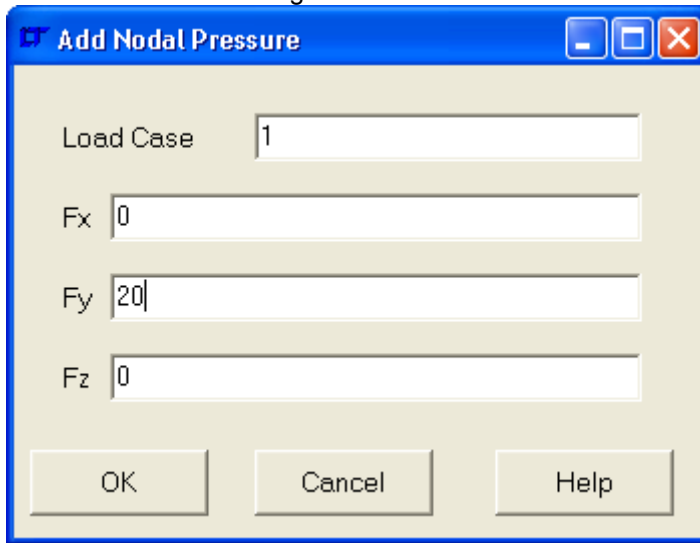
Select all Pressures that have a pressure of zero.

**'FE Model| Pressures| Inspect'**

Starts the interactive Pressures inspection routine. The mouse snaps to element face centroids with visible Pressures, and all the appropriate information is listed in the **'Progress Information'** box.

**'FE Model| Nodal Pressures'****'FE Model| Nodal Pressures| Add'**

Open a dialogue window to add new Nodal Pressures to the currently selected elements. A new load case set will be defined if the load case does not exist and **'FE Model| Load Cases| Auto Create'** is set. The dialogue window is as below:-



Where a Nodal Pressure already exists for a node for the specified load case, the Nodal Pressure is added to. This function only works with shell elements that are not loaded parallel to the shell surface.

**'FE Model| Nodal Pressures| Delete'****'FE Model| Nodal Pressures| Delete| All'**

Delete all Nodal Pressures.

**'FE Model| Nodal Pressures| Delete| Selected'**

Delete all selected Nodal Pressures.

**'FE Model| Nodal Pressures| Select'****'FE Model| Nodal Pressures| Select| All'**

Select all Nodal Pressures. Any Nodal Pressures initially invisible will be made visible.

**'FE Model| Nodal Pressures| Select| None'**

Deselect all Nodal Pressures.

**'FE Model| Nodal Pressures| Select| Invert Selection'**

Invert the selection of all Nodal Pressures. Any Nodal Pressures initially invisible will be made visible.

**'Selection| Nodal Pressures| by Load Case'**

Select all Nodal Pressures within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Nodal Pressures| by Node Number'**

Select all Nodal Pressures within a range of Node Numbers. Typed input of the range is necessary in the command line.

**'FE Model| Nodal Pressures| Select| Selected Nodes'**

Select all Nodal Pressures that are visible associated with the selected nodes.

**'FE Model| Nodal Pressures| Select| Selected Elements'**

Select all Nodal Pressures that are visible associated with the selected elements.

**'FE Model| Nodal Pressures| Select| Zero Value'**

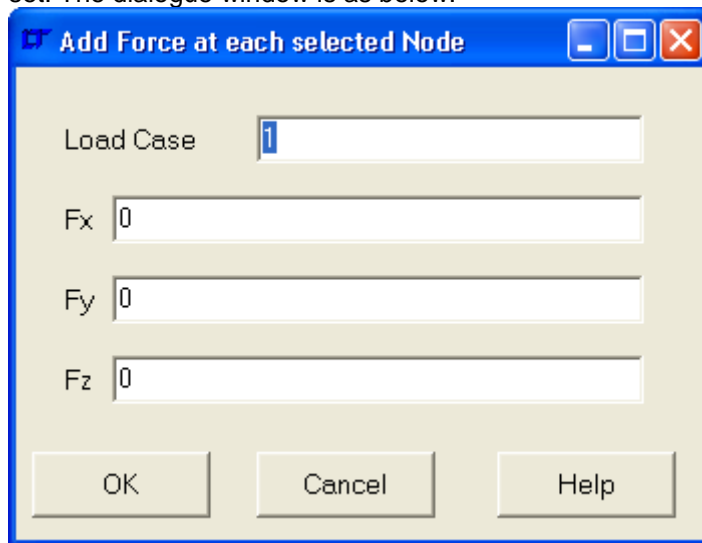
Select all Nodal Pressures that have a load of zero.

**'FE Model| Nodal Pressures| Inspect'**

Starts the interactive Nodal Pressures inspection routine. The mouse snaps to nodes with visible Nodal Pressures, and all the appropriate information is listed in the '**Progress Information**' box.

**'FE Model| Forces'****'FE Model| Forces| Add'**

Open a dialogue window to add new Forces to the currently selected nodes. A new load case set will be defined if the load case does not exist and **'FE Model| Load Cases| Auto Create'** is set. The dialogue window is as below:-



Where a Force already exists for a node for the specified load case, the Force is added to.

**'FE Model| Forces| Delete'****'FE Model| Forces| Delete| All'**

Delete all Forces.

**'FE Model| Forces| Delete| Selected'**

Delete all selected Forces.

**'FE Model| Forces| Select'****'FE Model| Forces| Select| All'**

Select all Forces. Any Forces initially invisible will be made visible.

**'FE Model| Forces| Select| None'**

Deselect all Forces.

**'FE Model| Forces| Select| Invert Selection'**

Invert the selection of all Forces. Any Forces initially invisible will be made visible.

**'Selection| Forces| by Load Case'**

Select all Forces within a range of Load Cases. Typed input of the range is necessary in the command line.

**'Selection| Forces| by Node Number'**

Select all Forces within a range of Node Numbers. Typed input of the range is necessary in the command line.

**'FE Model| Forces| Select| Selected Nodes'**

Select all Forces that are visible associated with the selected nodes.

**'FE Model| Forces| Select| Selected Elements'**

Select all Forces that are visible associated with the selected elements.

**'FE Model| Forces| Select| Zero Value'**

Select all Forces that have a load of zero.

**'FE Model| Forces|Modify'**

Replaces the forces on the selected nodes for the identified load case with those in the dialogue box.

If the node does not have an existing displacement constraint, the modify command will be ignored.

**'FE Model| Forces| Inspect'**

Starts the interactive Forces inspection routine. The mouse snaps to nodes with visible Forces, and all the appropriate information is listed in the **'Progress Information'** box.



**'FE Model| Accelerations'****'FE Model| Accelerations| Add'**

Open a dialogue window to add a new Acceleration Set. The dialogue window is as below:-

The dialog box is titled "Add Global Accelerations" and contains the following fields and buttons:

- Load Case Number: 29
- Linear Accelerations:
  - X: 0
  - Y: 0
  - Z: -1
- Centre of Rotation:
  - X: 0
  - Y: 0
  - Z: 0
- Angular Velocity:
  - X: 0
  - Y: 0
  - Z: 0
- Angular Acceleration:
  - X: 0
  - Y: 0
  - Z: 0
- Buttons: OK, Cancel, Help

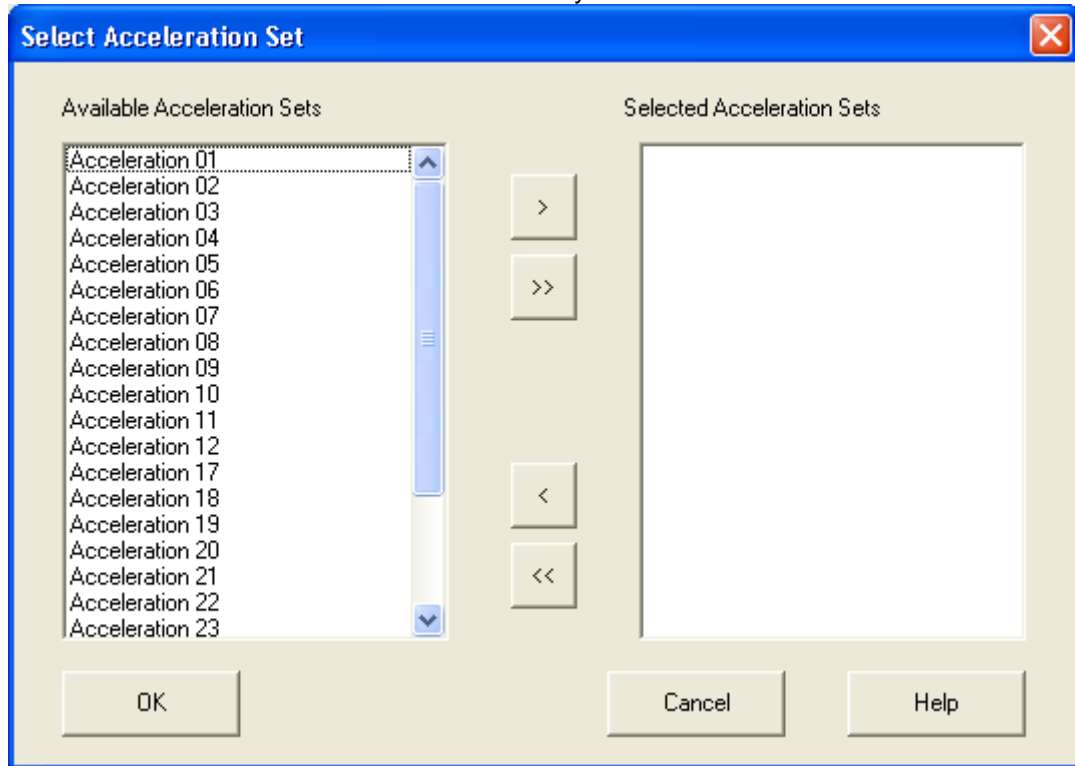
**Load Case Number**

The Load Case number is the load case that the acceleration is associated with.

The remaining data entries are self explanatory.

**'FE Model| Accelerations| Delete'**

Delete a range of Acceleration set numbers. A dialogue window will open enabling selection of the accelerations to be deleted. The window may be as below:-

**'FE Model| Accelerations| Delete All'**

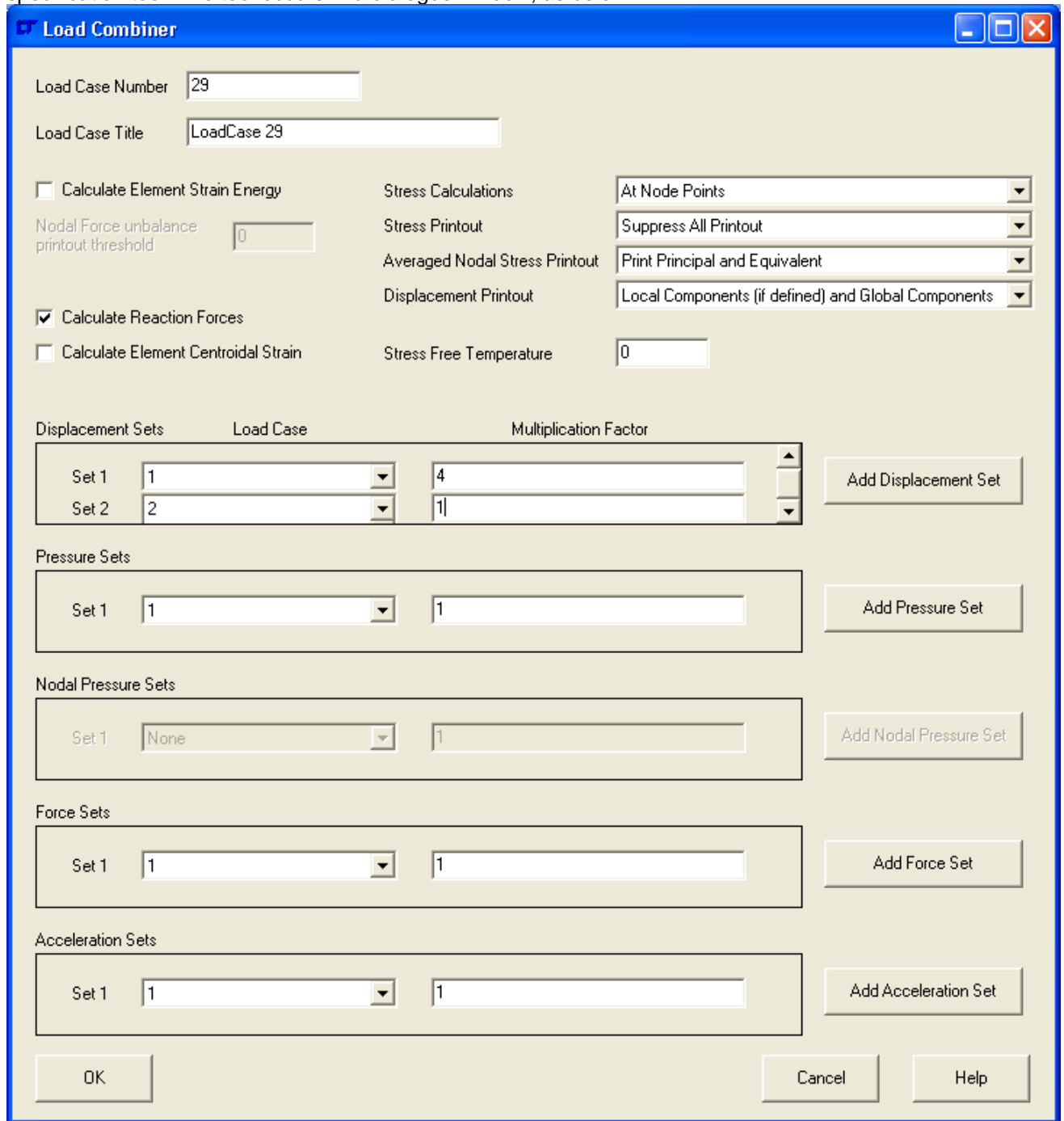
Delete all Accelerations.

**'FE Model| Accelerations| Inspect'**

Open a dialogue box similar to the '**FE Model| Accelerations| Add**' dialogue box, with the capability to scroll through the Accelerations in the model as well as add and delete Accelerations.

**'FE Model| Load Combiner'**

The load combiner is a heavy duty tool that can combine many load sets into one. It is a one stop Load Case definition, Displacement Constraints/ Pressure/ Nodal Pressure/ Force/ Acceleration specification tool. The tool occurs in a dialogue window, as below:-



The top part of the window is identical to the dialogue window that opens with menu command '**FE Model| Load Case | Add**'.

The lower part of the window specifies already defined boundary conditions that may be multiplied by a factor to get the new load case. This is best shown by example.

Consider:-

- Load case 1 = 1g forwards.
- Load case 2 = 1g starboard
- Load case 3 = 1g up.

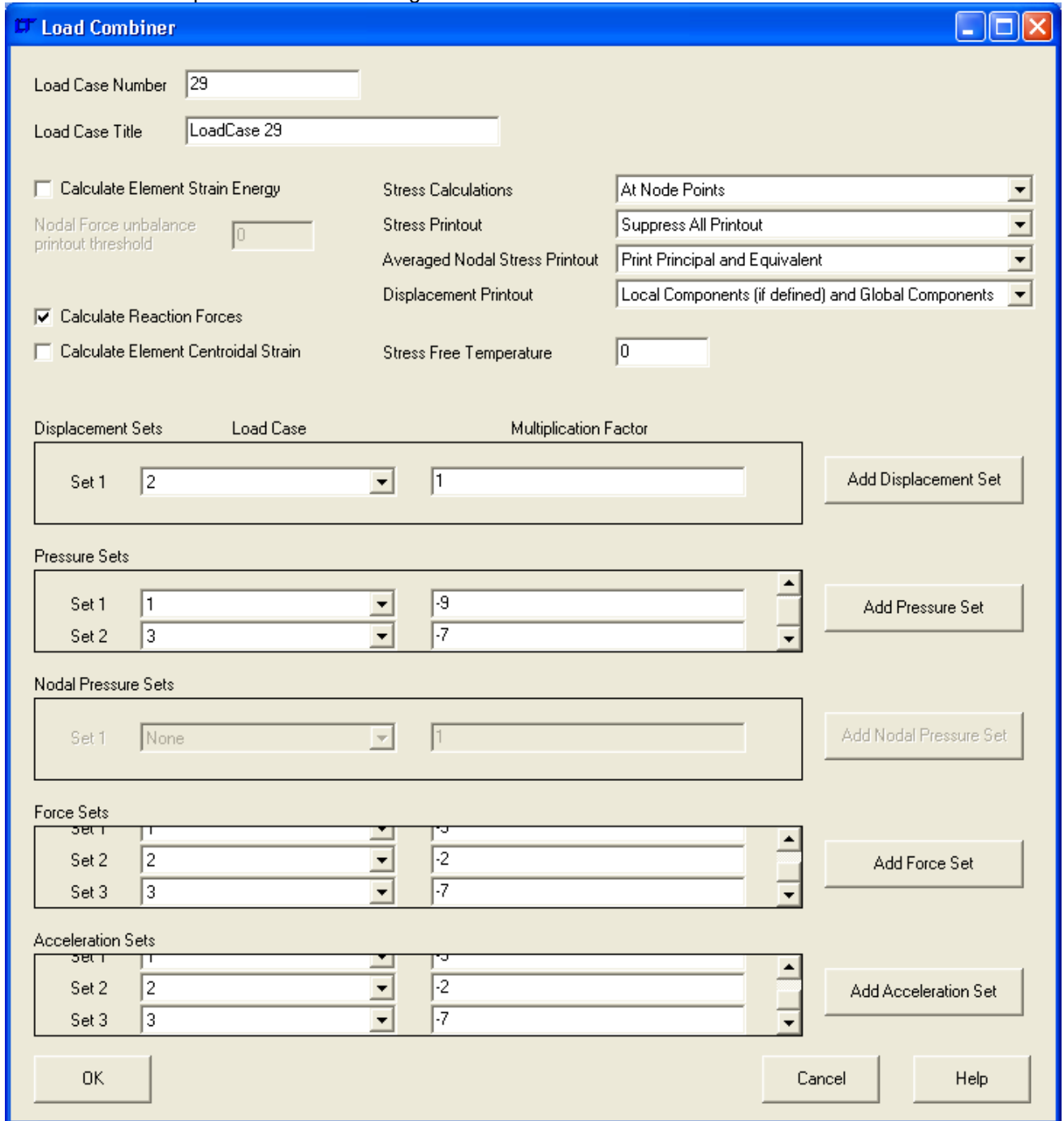
We wish to have a combination load case that is 9g aft, 2g port and 7g down, using the displacement constraints specified in load case 2. In load cases 1 & 3 part of the applied load is applied by a pressure force. There are no nodal pressures.

In the Displacements box, load set 2 would be multiplied by 1.

In the Pressures box, load set 1 would be multiplied by -9. Press the 'Add Pressure Set' button once, then set 2 would be specified as 3 and multiplied by -7.

In both the Forces and Accelerations box, multiply load set 1 by -9. Press the 'Add Force Set' and 'Add Acceleration Set' buttons twice, then fill in the additional set boxes as Set 2 x -2 and Set 3 x -7. If you make an error by pressing the 'Add \*\*\* Set' button too often, simply select load case 'None' for the load case and leave the multiplication factor as 1.

This ends up with a window looking like the one below:-

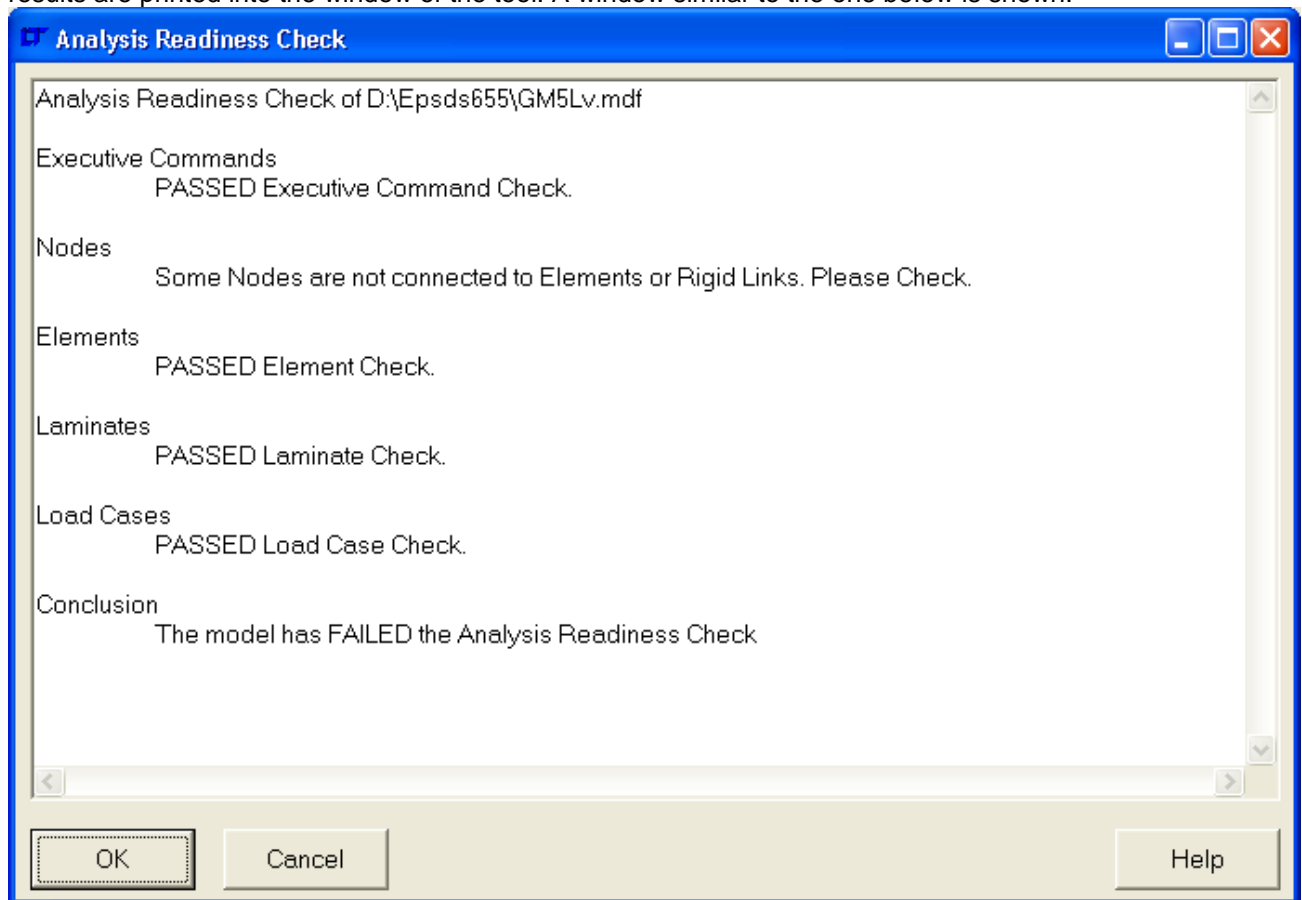


Once the 'OK' button is pressed, the tool creates a new load case with all the individual displacement constraints, pressures, nodal pressures, forces and accelerations in the same way as if they had been created manually. The multiplication factors etc cannot be amended after 'OK' is pressed.

For another example of the load combiner, see the 'User Guide'.

### 'FE Model| Analysis Readiness Check'

The 'Analysis Readiness Check' tool is another fairly heavy duty tool designed to ease the model verification burden. When the menu option is selected a window is opened and the readiness check results are printed into the window of the tool. A window similar to the one below is shown:-



The checks performed come under five categories:-

#### Executive Commands

- 1) Checks to see whether the results file name has been defined.
- 2) Whether binary geometry files are saved.
- 3) Whether binary results files are saved.

#### Nodes

- 1) Checks to see whether there are any Nodes not used by any Elements or Rigid Links.
- 2) Checks if any nodes fall within the 'Node Merge' Tolerance of each other. If they do, some nodes may need to be merged.

#### Elements

- 1) Checks to see whether all materials, properties and laminates called up by the elements are defined.
- 2) Checks whether any elements have duplicated nodes in their definitions (e.g. due to an excessive node merge tolerance).

### Laminates

- 1) Checks to see whether all materials, properties and laminate angles referenced by the laminate definitions exist.

### Load Cases

- 1) Checks to see whether each load case is adequately restrained against rigid body motion – this can get confused by many restrained nodes forming a hinge, but you wouldn't do that, would you?
- 2) Checks to see whether there is a load of some sort applied to the model for each load case.

### Conclusion

The conclusion is based on the model having passed all the checks. It is NOT definitive. Please note:-

- 1) Models will run that fail the checks, but they are more likely to run correctly first time if they pass.
- 2) Models that pass the checks can still have problems. The checks do not include duplicate element checks or whether models comprise two or more separate sections of elements.
- 3) Failing the check will not prevent the export of the analysis file.

Right clicking over the readiness check results window will activate the right click menu – enabling you to copy the results text to the clipboard should you wish to.

### 3.2.7 Mapping Menu

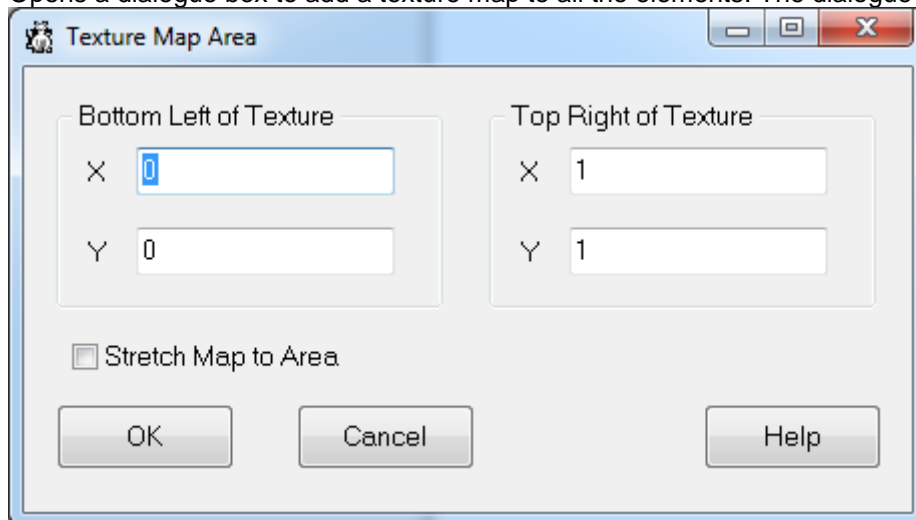
The following menu commands may be found in the Mapping menu. Texture mapping is used to wrap pictures around computer models to enable them to look more realistic in rendering packages.

#### 'Mapping| Add'

#### 'Mapping| Add| Current View Mapping'

##### 'Mapping| Add| Current View Mapping | All'

Opens a dialogue box to add a texture map to all the elements. The dialogue box is as below:-



The texture mapping on the elements is scaled to fit the current view, in the current proportions, into the rectangle specified in the dialogue box. If, however, the 'Stretch Map to Area', then the mapping will be stretched vertically or horizontally to fit the extents of the mapping area.

##### 'Mapping| Add| Current View Mapping| Selected'

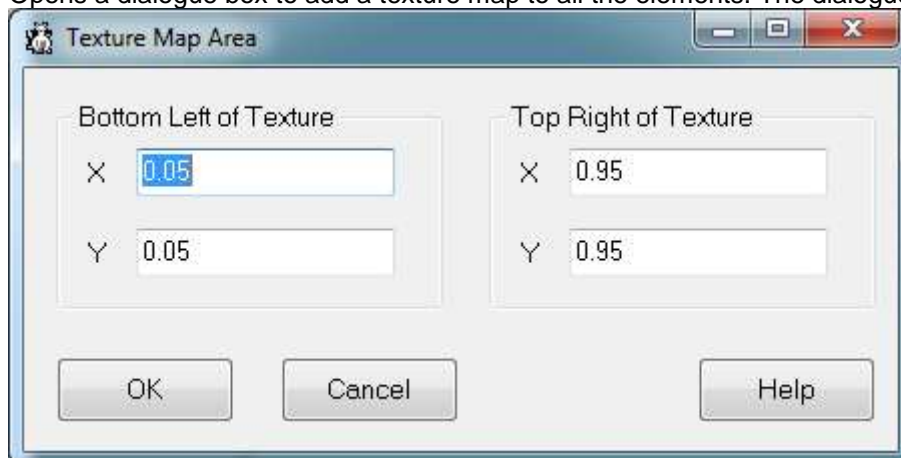
Opens a dialogue box to add a texture map to the selected elements.

The texture mapping on the elements is scaled to fit the current view, in the current proportions, into the rectangle specified in the dialogue box. If, however, the 'Stretch Map to Area', then the mapping will be stretched vertically or horizontally to fit the extents of the mapping area.

#### 'Mapping| Add| Box Mapping'

##### 'Mapping| Add| Box Mapping| All'

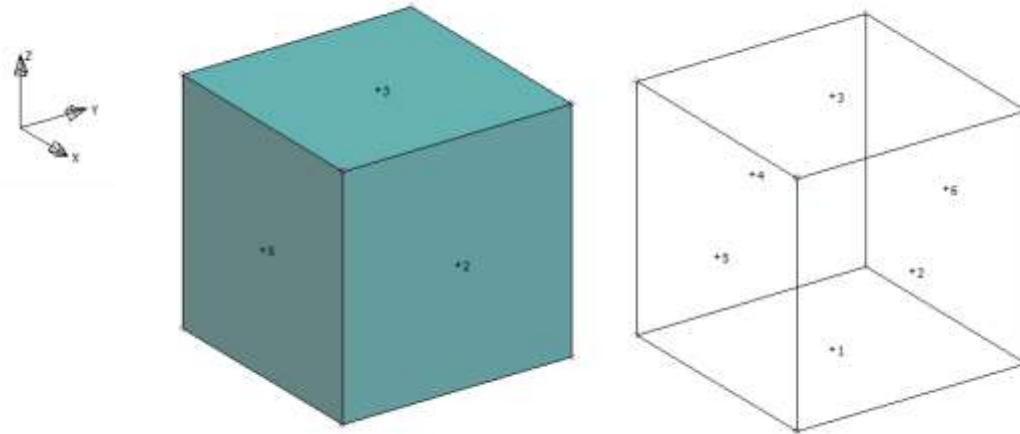
Opens a dialogue box to add a texture map to all the elements. The dialogue box is as below:-



The mapping is by means of creating an imaginary box around the model. All faces are then projected onto the appropriate face of the box, with the face normal being the vector that decides which box face the element face projects onto. The imaginary box is then unwrapped

into area defined by the rectangle left,bottom,right,top while keeping all proportions consistent. All existing texturemapping is deleted and all existing texture nodes are deleted prior to box mapping taking place.

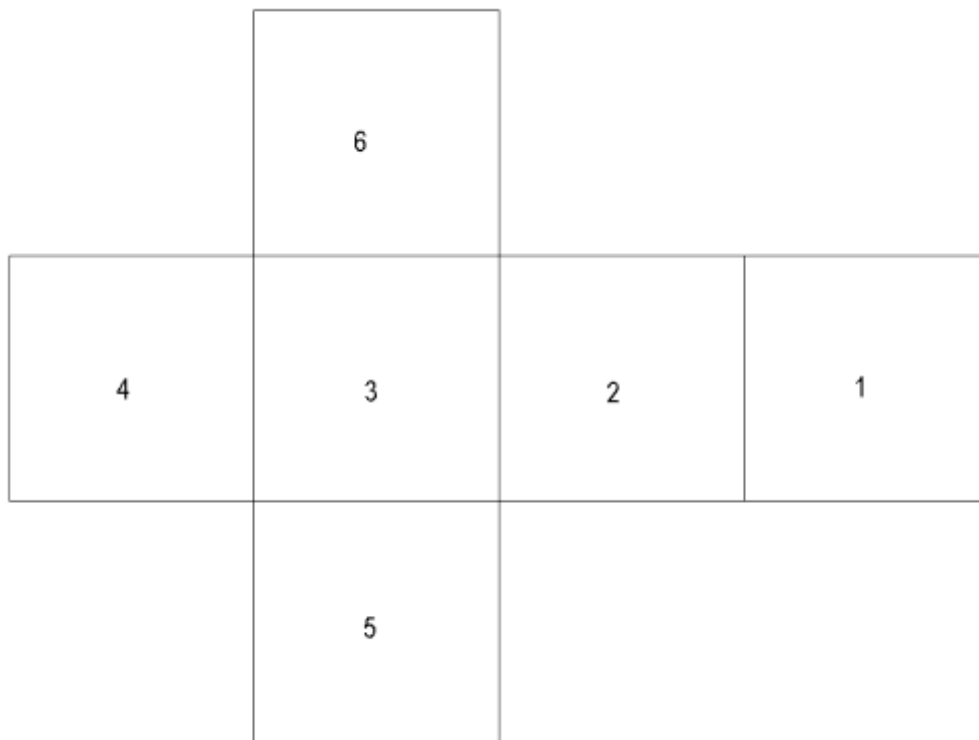
The imaginary box around the model is as shown below:



The view correspondence is:

- Area 1: Bottom
- Area 2: Right
- Area 3: Top
- Area 4: Left
- Area 5: Front
- Area 6: Back

The mapping is transformed on the texture map to the areas shown below:

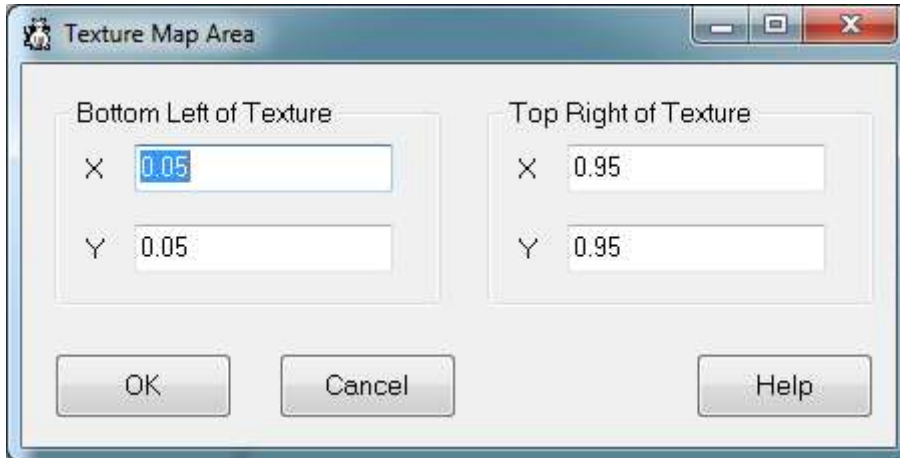


Note: these areas are not fixed between one model and the next, and adjust in height and width to match the proportions of the model extents. The texture map wrapping has been chosen to be least inconvenient for packages that use the 'Y' axis as the 'up' axis.



**'Mapping| Add| Box Mapping| Selected'**

Opens a dialogue box to add a texture map to all selected elements. The dialogue box is as below:-



The mapping is by means of creating an imaginary box around the selected model. All selected faces are then projected onto the appropriate face of the box, with the face normal being the vector that decides which box face the element face projects onto. The imaginary box is then unwrapped into area defined by the rectangle left,bottom,right,top while keeping all proportions consistent. All existing texturemapping is deleted on the selected faces. No existing texture nodes are deleted. It should be noted that the selected model is unlikely to have the same dimensions or proportions as the full model, so it is highly unlikely the the mapping of selected faces will match up with box mapping from the full model.

**'Mapping| Delete'****'Mapping| Delete| All'**

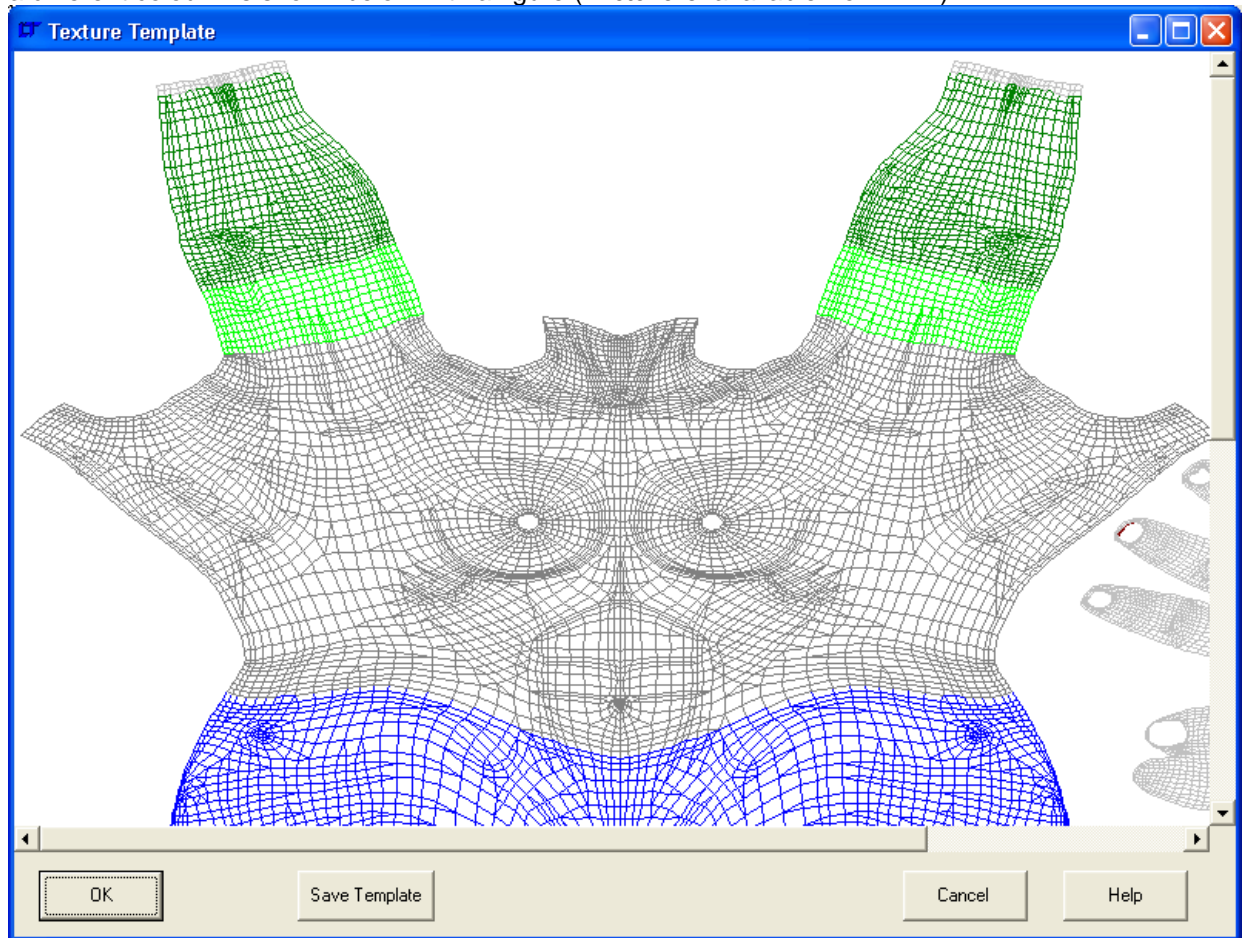
Delete all texture mapping.

**'Mapping| Delete| Selected'**

Delete texture mapping on selected elements.

**'Mapping| Templates'****'Mapping| Templates| All Layers'**

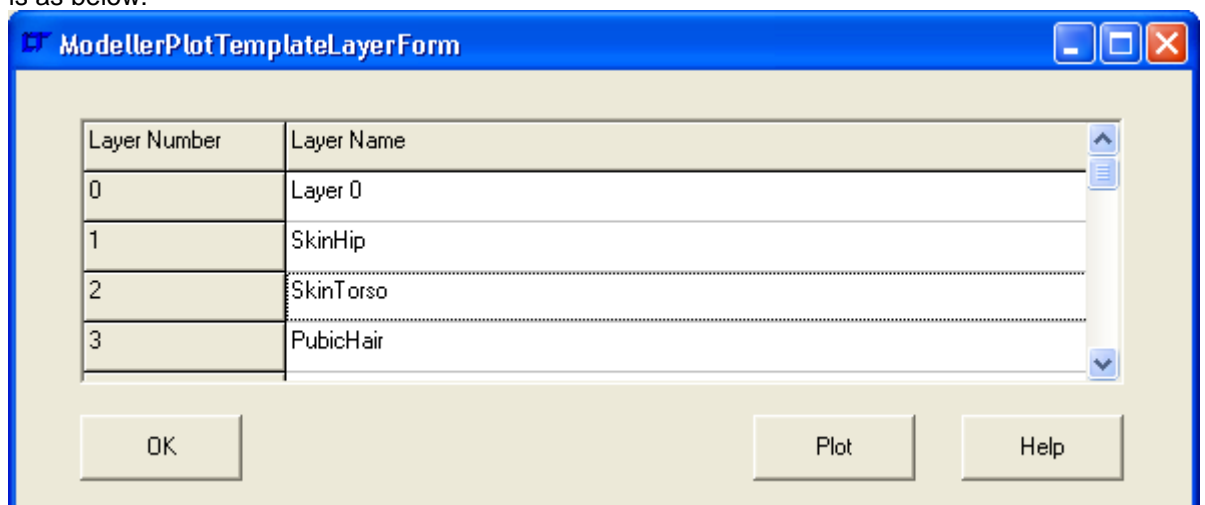
Opens a coloured texture template window to show the texture on all elements. Each layer has a different colour. As shown below with a figure ('Victoria 3' available from DAZ):



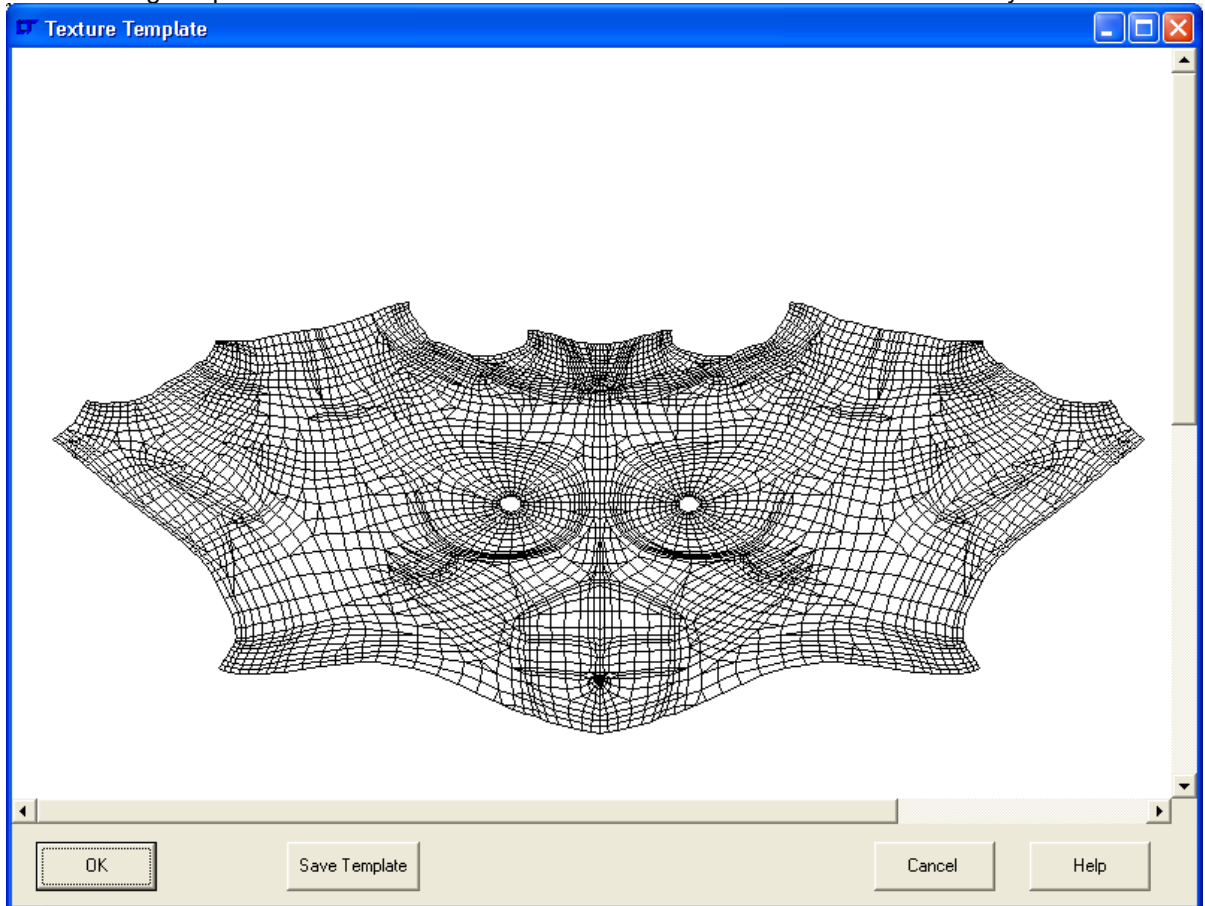
The template can be copied to the clipboard by right clicking over the template image area.

**'Mapping| Templates| Single Layer'**

Open a dialogue box to create a texture template of a single layer. The layer selection dialogue is as below:-



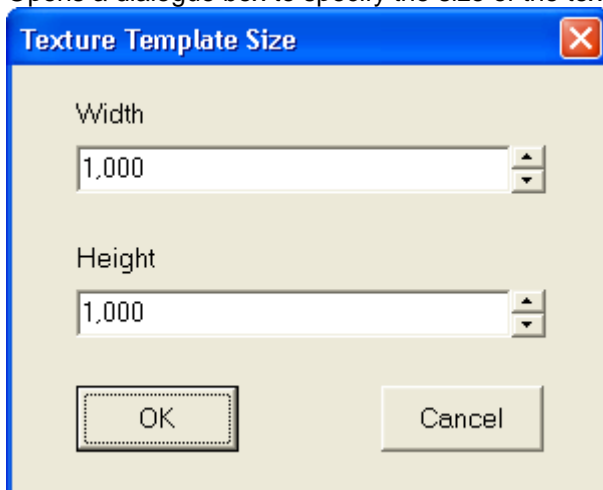
The resulting template is monochrome as there is no need to differentiate between layers:-



The template can be copied to the clipboard by right clicking over the template image area.

#### 'Mapping| Templates| Set Size'

Opens a dialogue box to specify the size of the texture template.



The template is limited to 2000 pixels in any direction.

### 3.2.8 Window Menu

The following menu commands may be found in the Window menu.

**'Window| Cascade'**

Cascade different tools within the Modelling Studio program.

**'Window| Tile Horizontal'**

Tile different tools horizontally within the Modelling Studio program.

**'Window| Tile Vertical'**

Tile different tools vertically within the Modelling Studio program.

**'Window| Next Window'**

View the next tool in the Modelling Studio program.

**'Window| Previous Window'**

View the previous tool in the Modelling Studio program.

**'Window| Arrange Icons'**

Arrange Icons of minimised tools in the Modelling Studio program.

**'Window| Maximise Window'**

Maximise tool window in the Modelling Studio program.

**'Window| Restore Window'**

Restore tool window in the Modelling Studio program.

**'Window| Minimise Window'**

Minimise tool window in the Modelling Studio program.

**'Window| Window List'**

List of open tool windows in the Modelling Studio program.

### 3.2.9 Help Menu

The following menu commands may be found in the Help menu.

**'Help| Help'**

Opens the Modelling Studio program Help file. The help file has NO information regarding the Modelling Studio.

**'Help| Modelling Studio Documentation'**

Opens all three Modelling Studio Help files (the Quick Start Guide, Command Reference and User Guide) at the same time.

**'Help| Help on Help'**

Opens the Windows 'Help on Help' file if your operating system has one. Windows XP seems to deliberately misunderstand this API call.

**'Help| Chestnut Pens Web Site'**

Sends you off to the Chestnut Pens web site [www.chestnutpens.co.uk](http://www.chestnutpens.co.uk)

**'Help| Technical Support'**

Starts an e-mail to [richard.williams@epslt.co.uk](mailto:richard.williams@epslt.co.uk) .

**'Help| About'**

Starts a window with an animated graphic giving information about EPS LTL.



### 3.2.10 Pop-up menu over model Viewport

The following menu commands may be found in the Pop-up menu over the model Viewport. Not all the commands are visible at the same time.

**'End Command'**

If carrying out an interactive command (eg **'FE Model| Node| Inspect'**) this menu item appears and can be used to stop the command.

**'Copy Model Image'**

Copies the image currently displayed of the model to the clipboard.

**'Save Image'**

Saves the model image to file.

**'Save Image As'**

Saves the model image to file, allowing the file name to be modified.

**'View'**

**'View| Front'**

View the front of the model, along the +Y axis.

**'View| Back'**

View the back of the model along the -Y axis.

**'View| Left'**

View the left side of the model along the +X axis.

**'View| Right'**

View the right side of the model along the -X axis.

**'View| Top'**

View the top of the model along the -Z axis.

**'View| Bottom'**

View the bottom of the model along the +Z axis.

**'View| Iso 1'**

View the Iso 1 view of the model.

**'View| Iso 2'**

View the Iso 2 view of the model.

**'View| Iso 3'**

View the Iso 3 view of the model.

**'View| Iso 4'**

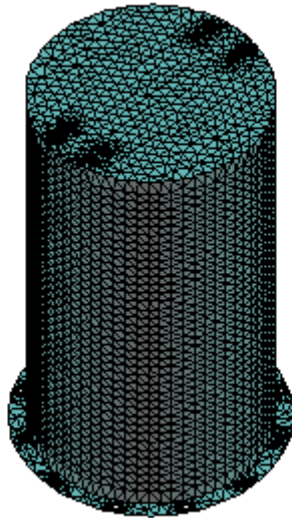
View the Iso 4 view of the model.

**'View| User Defined'**

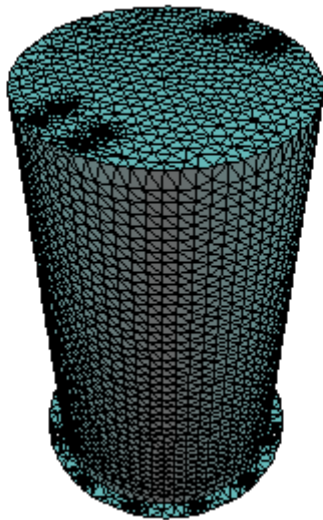
Open a dialogue box to specify the viewing angle.

**'Perspective'****'Perspective| None'**

No perspective effects are applied to the model view in the **Viewport**. As shown below:-

**'Perspective| Wide Angle Lens'**

Strong perspective effects to simulate a wide angle lens are applied to the model view in the **Viewport**. As shown below, this is from the same angle of view as the zero perspective view above.

**'Perspective| Standard Lens'**

Medium strength perspective effects to simulate a standard lens are applied to the model view in the **Viewport**.

**'Perspective| Telephoto Lens'**

Mild perspective effects to simulate a telephoto lens are applied to the model view in the **Viewport**.

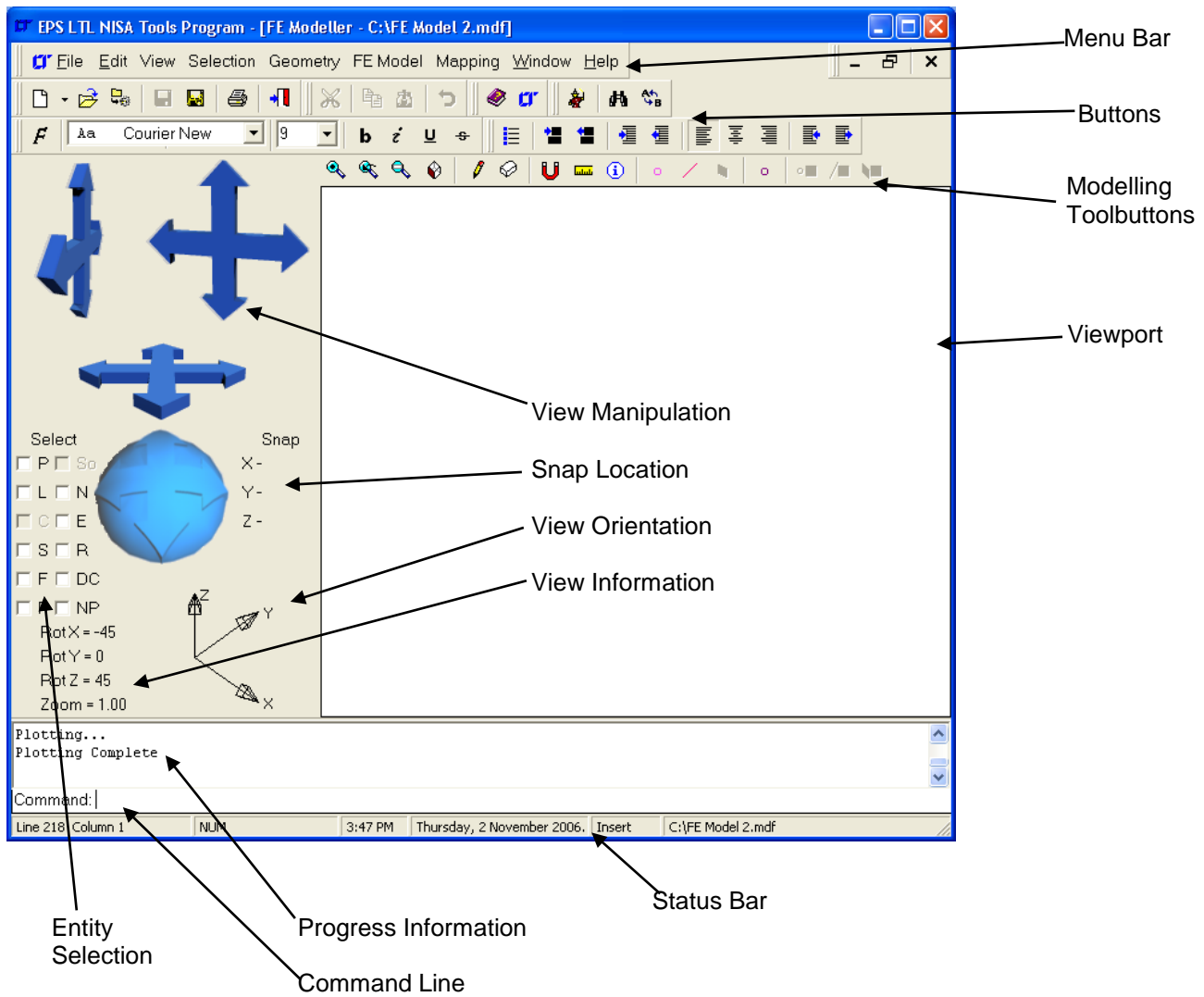
## 4. Interface Reference

### 4.1 General Information

This section identifies the active components on the Modelling Studio interface.

### 4.2 General Layout

The interface for the program is a little unusual and needs some explanation.



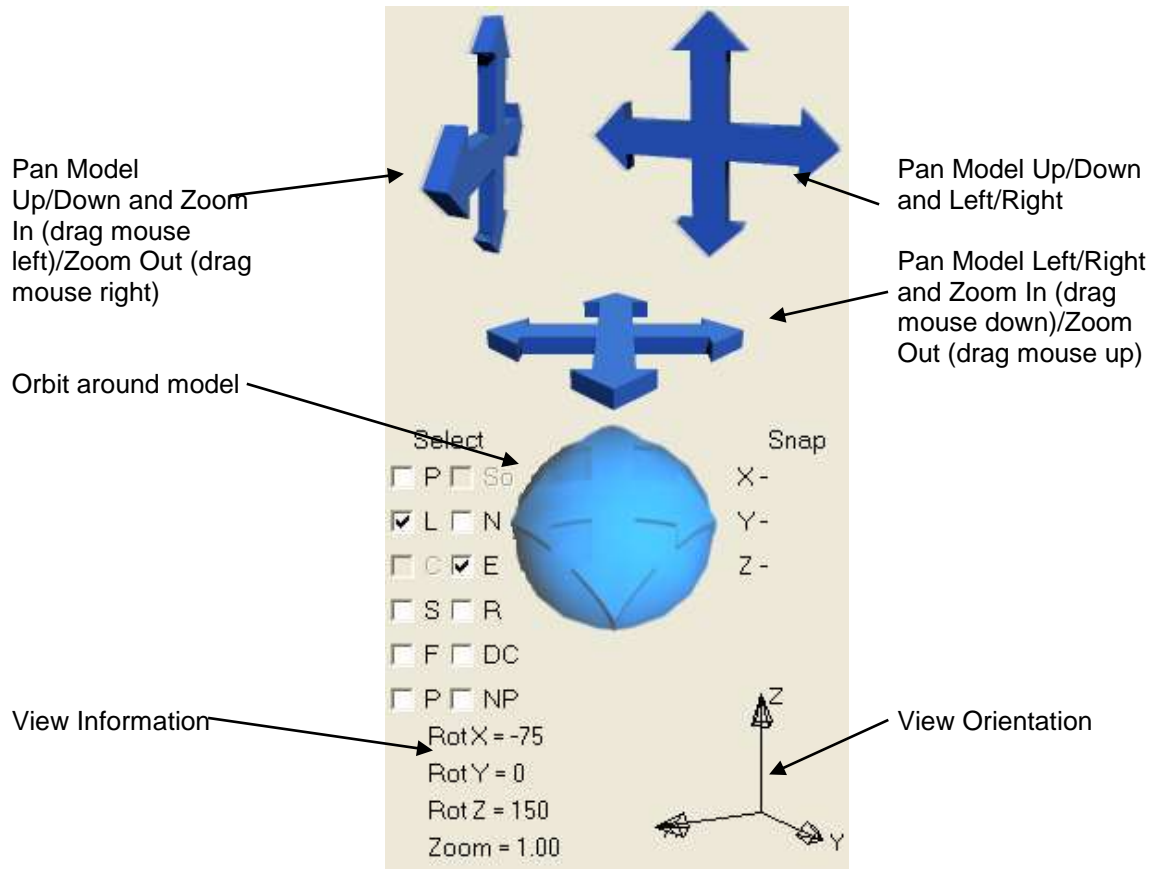
Generally when working you will be using the **Viewport**, **Entity Selection** tickboxes and the **Modelling Toolbuttons**. Typed data entry (if required) is through the **Command Line**. When more sophisticated model manipulation is necessary you will use the **Menu Bar**, where the work flow is from the top to the bottom down each menu and from left to right along the menu bar.

Now to explore the interface components in a little more detail.



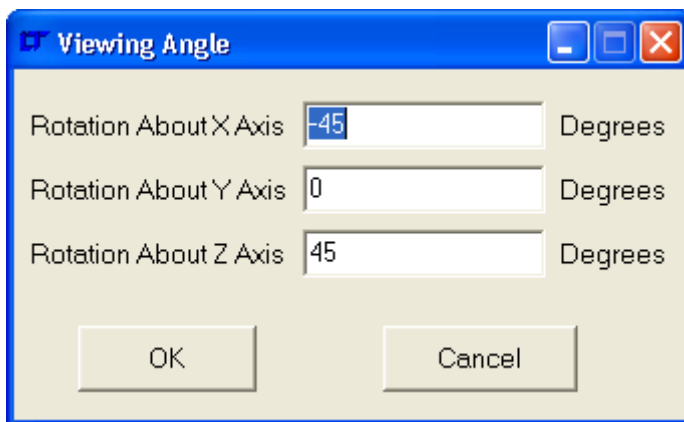
### 4.3 View Manipulation

There are four parts to the view manipulation tool, as identified below:-



As you move the view, you will see the **View Orientation** axes change, as these remain pointing in the model X, Y and Z directions. The **View Information** indicates the current angle of view and zoom level. The zoom level is 1.00 when the currently visible model is scaled to fill the **Viewport**, smaller than 1.00 when zoomed out and larger than 1.00 when zoomed in.

Double click on the **View Orientation** or **View Information** to open a text entry View angle window, (as shown below) to enter a specific viewing angle.



### 4.4 Entity Selection

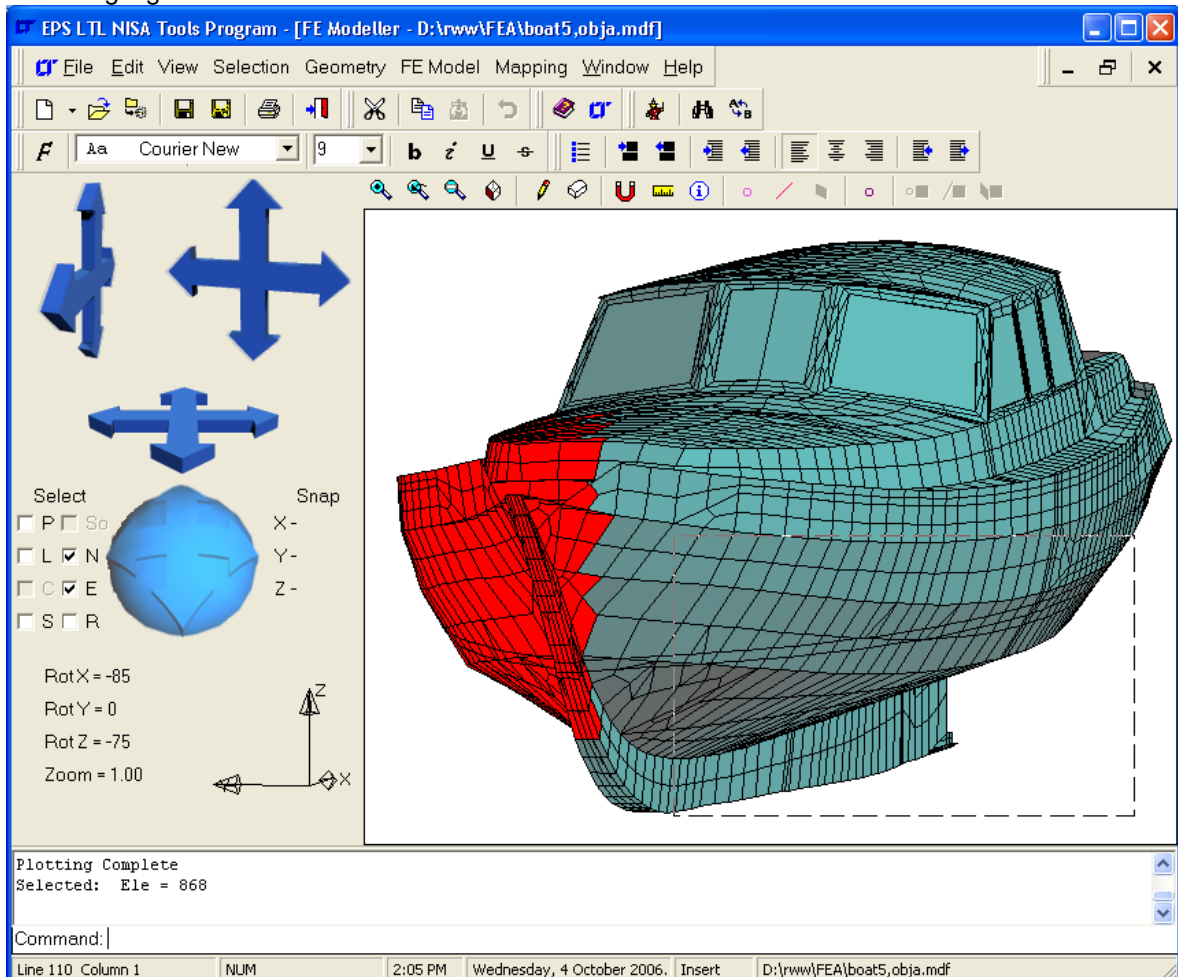
Many commands within the Chestnut Pens Modelling Studio work on selected entities. You select an entity (like a point, line or surface), and do something to it (copy, move, etc.). The selection process is done by:-

- 1) Decide which entities are to be selected in the **entity selection** tickbox area.

- P = Points
- L = Lines
- C = Curves (not implemented yet)
- S = Surfaces
- So = Solids (not implemented yet)
- N = Nodes
- E = Elements
- R = Rigid Links
- DC = Displacement Constraints
- F = Forces
- P = Pressures
- NP = Nodal Pressures



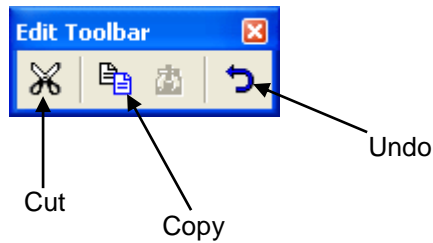
- 2) Drag a rectangle across the **Viewport** and release the mouse. The selected entities will be highlighted as shown below:-



## 4.5 Toolbuttons

Very few of the **Toolbuttons** are useful for the Modelling Studio. Most of them operate on the **Progress Information** box, which is of minor importance to the Modelling Studio. The majority of the buttons are covered in the help file.

The tool buttons that operate on the modeller are:-



### Cut

Delete selected entities

### Copy

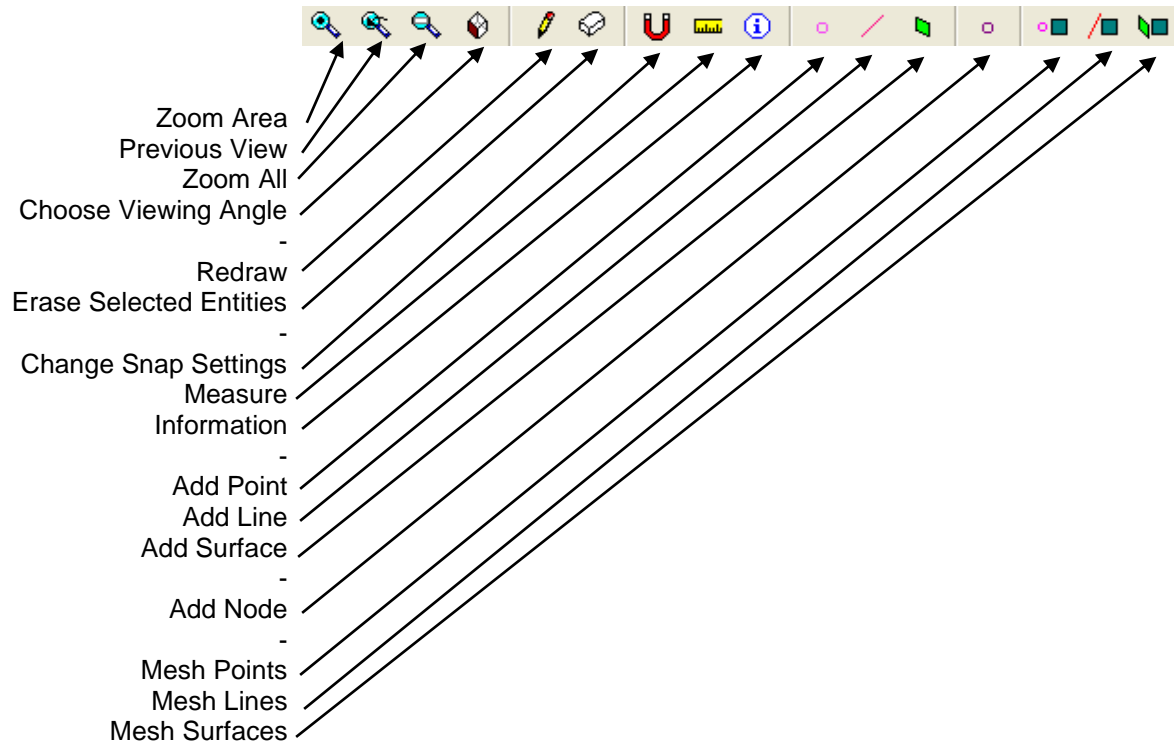
Starts the entity copy routine to copy the selected entities. Pick a 'From' point and a 'To' point to define the translation.

### Undo

Undo the last command. Look in the status bar for the command that will be undone, as, at the time of writing of this command reference, not all the commands have 'Undo' features that are implemented.

### 4.6 Modelling Toolbuttons

The **Modelling Toolbuttons** are as below:-



Not all the buttons are enabled all the time. If there are no points, then the 'Mesh Points' button is disabled etc.

Zoom Area

Zoom to a rectangular area dragged in the viewport.

Previous View

Return to the previous view in the zoom history list.

Zoom All

Zoom to the whole model.

Choose Viewing Angle

Opens the '**View| Angle**' menu.

Redraw

Redraw the screen

Erase Selected Entities

Erases all the selected entities. No deletion occurs.

Change Snap Settings

See 4.7

Measure

Measure between two pick points on screen. The total distance and X, Y and Z components of the distance are specified.

Information

Inspect the entities picked on screen.

Add Point

Add points by snap location or co-ordinate entry.

Add Line

Add lines by snap location or co-ordinate entry.

Add Surface

Add surfaces by picking lines or by typed line entry.

Add Node

Add nodes by snap location or co-ordinate entry.

Mesh Points

Start interactive point meshing routine. Points selected on screen are meshed with point elements according to the current '**Geometry| Mesh| Points| Mesh Settings**'.

Mesh Lines

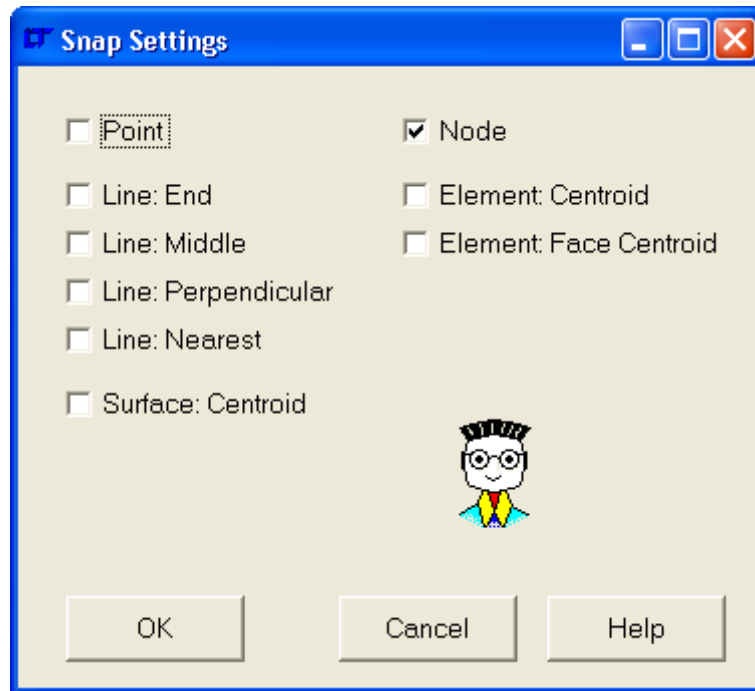
Start interactive line meshing routine. Lines selected on screen are meshed with line elements according to the current '**Geometry| Mesh| Lines| Mesh Settings**'.


Mesh Surfaces

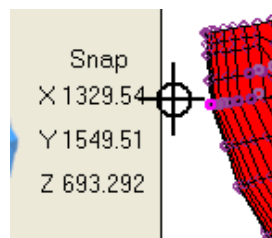
Start interactive surface meshing routine. Surfaces selected on screen are meshed with shell elements according to the current '**Geometry| Mesh| Surfaces| Mesh Settings**'.

## 4.7 Snap Location

When manipulating the model, it is often useful to have the cursor snap to particular entities within the model. For instance a user may wish to draw a line from one node to another. The Snap Settings (via the **Modelling Toolbuttons** ) would be set as below:-



Then the Line  button would be pressed. As the cursor moves over the model, possible snap sites are highlighted, as shown below:-



The current **Snap Location** is shown, to enable the user to verify whether the program is choosing the correct end location for the line.

*That's All Folks!*

\*

At last!

\*



\*



*I'm sure I've forgotten something..*

\*