EPS **Modelling Studio**

FEA models in easy steps.

User Guide

Copyright Notice

Contents

1.   Introduction

This document is the user guide reference intended to accompany the EPS Modelling Studio program.

The guide is intended to cover the ground between the basic 'Quick Start Guide' and the focused 'Command Reference'. Hopefully most questions not answered in either of the other guides are covered here.

If you are modelling to create a FE model, most of the guide will be relevant, however if you are simply modelling so that you can create a model for a rendering package, then look at the 'How to Model for Rendering' guide in section 3 before getting too bogged down with stuff you're not interested in.

It is envisaged that this guide will be updated fairly regularly as comments, new ideas or queries get incorporated.

2.   Modelling

Section 2.1 specifies the general philosophy behind creating the model and working with it. Section 2.2 gives a brief, fairly non-technical overview of how the Modelling Studio works. The remaining sections are fairly self explanatory. This section only covers items not covered in the 'Quick Start Guide', so basic FE model creation is not covered.

2.1   General Modelling Philosophy

2.1.1   The Modelling Studio is intended to be as easy to use as possible. It is intended that a majority of the most used commands and functions will require no resort to the command line or menu structure at all. Wherever possible, screen pick points are used, as this is the quickest and most efficient way of identifying entities.

2.1.2   When a command is to act on a number of entities, the entities are selected first – then the function is called and operates on them.

2.1.3   If a menu item needs to be called, generally the menus required are to the left, working to the right as the model progresses, and from top to bottom in each menu.

2.1.4   When creating a model, it is started with Lines [by preference] or Points. The Lines are used to create Surfaces. Once the Surface geometry is created it can be modified by moving Points, Lines or the Surfaces themselves. Having been satisfied with the geometry, the model is meshed. When the model is meshed to create Nodes and Elements, the link with the geometry is broken, so moving a Point will not move any Nodes or Elements. Once the elements have been created, the remaining FE information may be created, such as Rigid Links, Materials, Properties and all the Load Case information. If your workflow is such that you prefer to have properties, materials, laminates etc defined before the elements, then you can do this. The modeller tries to allow you to work the way *you* want, not to force you to work the way *it* does.

2.2   General Overview of how the Modelling Studio works

2.2.1   The Modelling Studio is a native Windows .exe program that conforms to the requirements of the Windows 'Multiple Document Interface' model and supports various child windows. All the child windows are C++ descendants of a basic word processor, and as a result some part of every child window is a word processor. In the case of the Modelling Studio 'Modeller' tool, the word processor section is the '**Progress Information**' box. Each descendant of the word processor has its own functions that modify or add to the word processor functions. Some of the tools are second or third generation descendants [like the NISA File Recovery tool], though the Modelling Studio is a first generation descendant.

2.2.2   The Modelling Studio holds the model as a series of arrays of entities, so there is an array of Points, another for Lines etc. When the Modelling Studio saves its binary native file, all that is saved is a copy of the memory storing all the data. The memory size can be quite significant – a 100,000 shell element model can take up 80-90 Mbytes of RAM.

2.2.3   Command Handling: In most cases a menu command or button press writes the command text to the '**Command Line**' and automatically triggers the command line parsing routine to carry out the action. As the parsing routine is triggered, the command is automatically added to a session file on disc (for its filename see the beginning of the text in the '**Progress Information**' box). The session file is opened and closed for each command that is added, ensuring maximum safety from corruption. The undo command is written to an equal and opposite session file held in memory.

When the [Carriage Return] key is pressed having typed in a command, the command line parsing routine is triggered, and adopts the same route to creating the action as a menu command.

When an 'Undo' command is triggered, or a session file is read in, a series of actions are directed through the command line.

The command parsing routine amounts to about one fifth of the code in the Modelling Studio – which is hardly surprising, considering what it has to do.

2.2.4   When an additional window appears with model information in it, such as the one which occurs when following the menu path '**FE Model| Materials| Inspect**', the data held in the window is a copy of the model data. This means that the user can amend and discard the amendments without detriment to the original model data. It is all kept as safe as possible until the user decides to change it.

2.3   Modelling Geometry

A majority of the information needed to model geometry is covered in the 'Quick Start Guide'. The exceptions are covered below.

2.3.1   Copying Entities

Very little information, beyond the basic 'From Here, To Here' type copying, is given in the Quick Start Guide. The description below concentrates on copying, but everything applies equally to moving.

To copy entities you need to select the entities first. Then either press the '**Copy**' button or select menu command '**Geometry| Points| Copy**', '**Geometry| Lines| Copy**' or '**Geometry| Surfaces| Copy**'. If the '**Copy**' button is pressed, then the behaviour is as specified in the 'Quick Start Guide'. Otherwise you are presented with a dialogue window with four tabs, Translate, Rotate, Mirror and Scale.
Translating:-
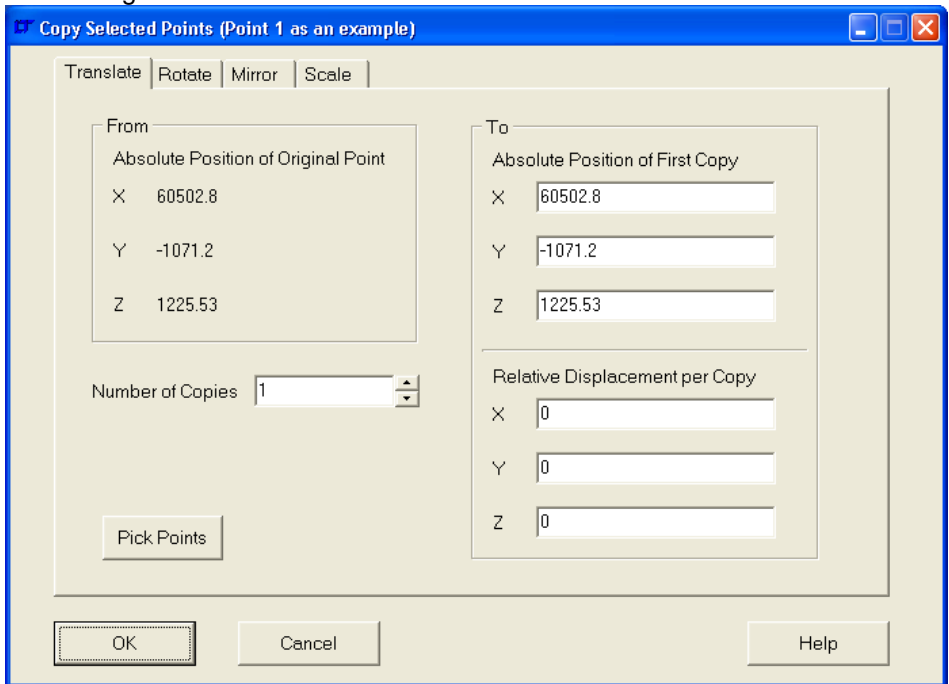
The Translate window always takes an entity key feature as an example. For a point, it is the point location. For a Line, the key feature is the line midpoint, and for a surface it's the surface centroid.

The 'From' point is the key feature of the example displayed. The 'To' point of the translation may be entered directly, or alternatively the 'Relative Displacement per Copy' may be entered. In either case, as you change one, the other alters.

If you need more than one copy, increase the 'Number of Copies'. For example, if the 'From' point is 0,0,0, the 'Number of Copies' is 2, and the 'Relative Displacement per Copy' is 1,0,0, then new points will occur at 1,0,0 and 2,0,0.

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 relative displacement per copy. The 'Pick Points' button is equivalent to pressing the '**Copy**' button, but will *only* work on the entities initially chosen. For example, if you got to the translate window by '**Geometry| Points| Copy**', then only points will be copied, whatever else was initially selected.

Rotating:-
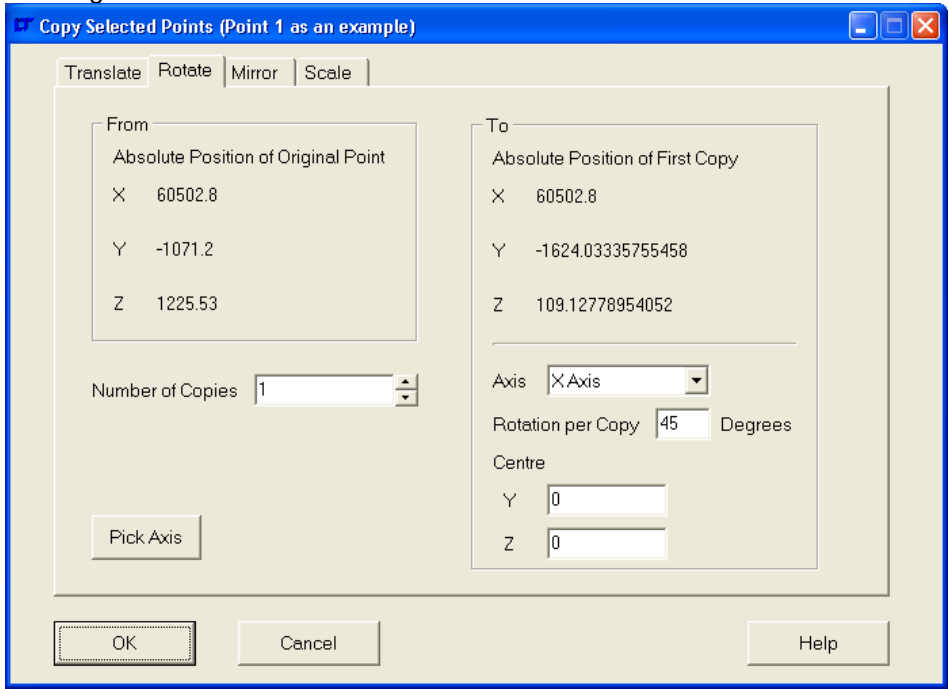
When rotating you specify the 'Number of Copies', the angle of 'Rotation per Copy' and the axis of rotation. Three axes of rotation may be entered on the form, the X, Y or Z axis. These axes may be offset to pass through a point other than the global origin. Alternatively, to specify an axis other than the principal axes, press the 'Pick Axis' button. It will then be possible to specify two pick points to orient the actual axis of rotation using the right hand rule (thumb = axis direction, curled right fingers = direction of positive rotation).

Mirroring:-
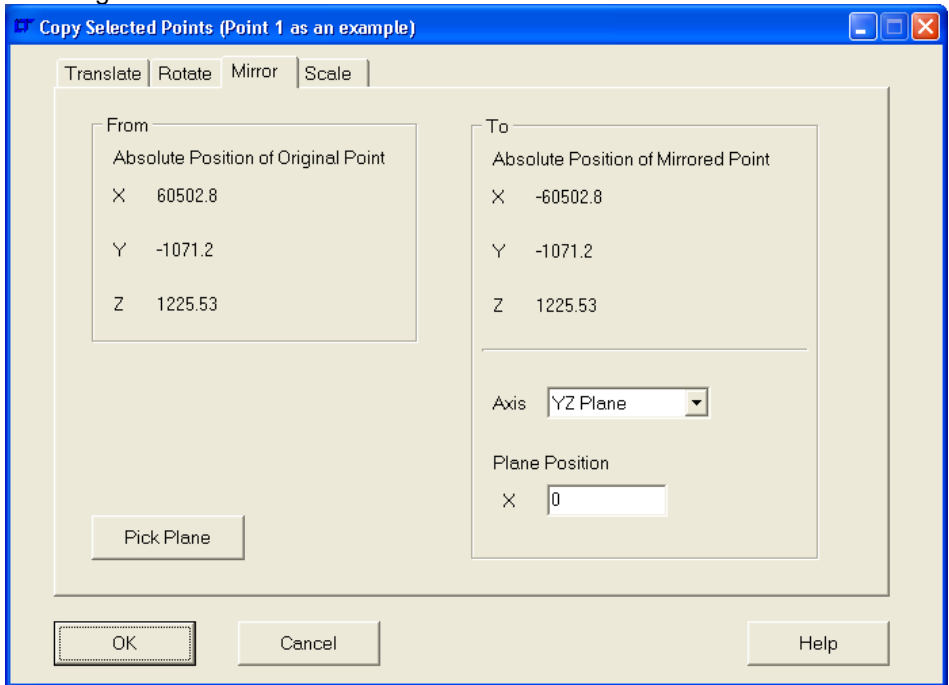
When mirroring entities, only one copy is made. The mirror plane has to be defined, and this may be done either on the form or by pressing 'Pick Plane'. Once the 'Pick Plane' button is pressed, interactively select 3 pick points to define the mirror plane [as usual, pick points can be typed in if necessary].

Scaling:-
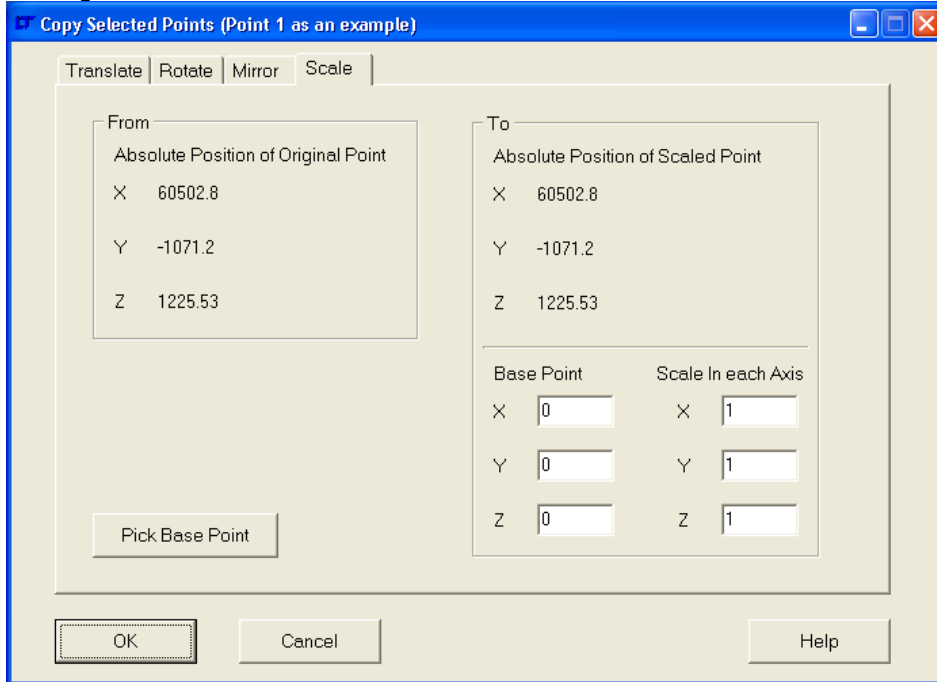
When scaling, a base point for the scale must be determined (from which all the entities are scaled), and the scale in each axis. The scale may be different in each axis to distort the copy. Note that when scaling the distance between the selected point and the base point is scaled by the axis scale factor. Consequently points at the base point will appear not to be copied.
When scaling entities, only one copy is made.

Press the 'Pick Base Point' button to pick the base point interactively.

2.3.2 Moving Entities

Entities are moved by exactly the same process as copying entities, except that no copies are created.

2.4   Modelling Finite Element Entities

The creation of finite element entities such as Nodes, Elements, Displacement Constraints, Pressures and Forces is covered in the 'Quick Start Guide'. Consequently only three finite element entities remain to be discussed; Rigid Links, Nodal Pressures and Accelerations.

2.4.1   Creating Rigid Links

Firstly, what are Rigid Links, and where should they be used?

A Rigid Link is really a modelling short cut. It is a way of transferring load from one point to another part of the model without having to model the mechanism or the actual bits that are necessary to cover the intervening space.

A rigid link can be used when part of the structure is currently un-defined or is particularly rigid compared to the remaining structure. Typical uses may be to connect a mass element at an object CG to the structure it loads (like an electric motor down to its attachment points) or to represent a heavy bracket that transfers load from one part of a structure to another. In an extreme, rigid links can be used to replace the bolts attaching two halves of a structure together, where the bolts are short enough for the stiffness difference to be negligible.

It's possible to over use rigid links and accidentally short out the structure, so do be aware of this. Take a galley that has a known contents in each aperture. The temptation is to put a mass element at the CG of every compartment and connect it to the walls of the compartment, as in the two middle compartments below:-
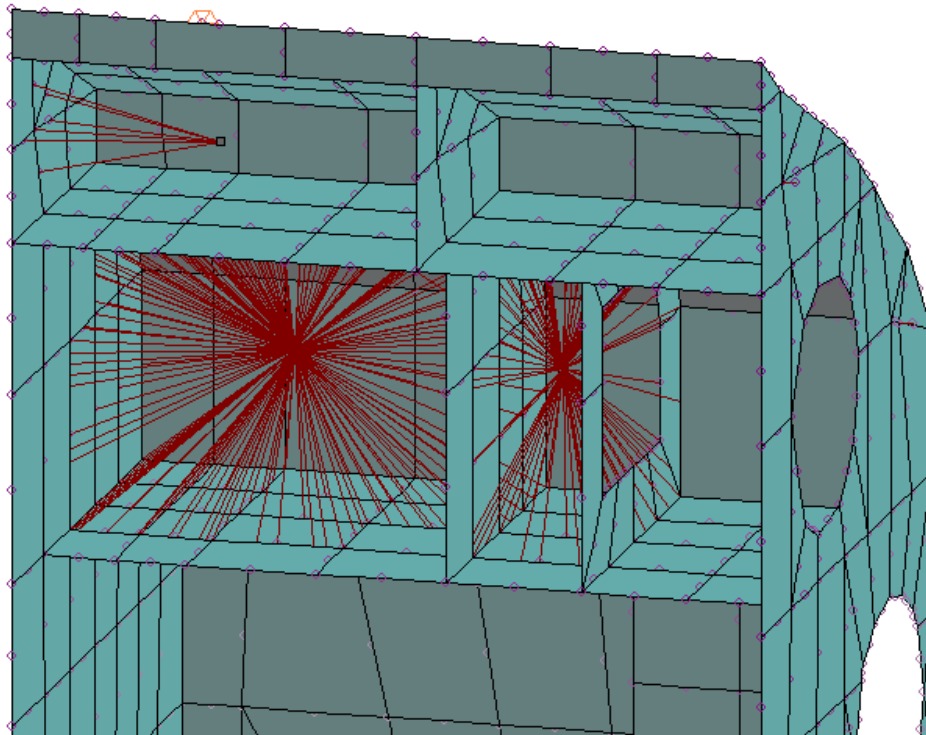
This won't work, because the structure has been short circuited, and the panels surrounding each box will behave as a single rigid lump without deforming relative to each other. This is where you need to use forces, pressures or nodal pressures. The other problem with this is that the nodes on the panel between the compartments are slave nodes of more than one Rigid Link; this is not accepted by analysis packages and will cause an error [the Modelling Studio does try to prevent this, but ingenious users may be able to get around the checks – try not to do it!].

Create a Rigid Link

The way to create a Rigid link is described below.

Firstly the model must be nearly complete with all the nodes you need for the model. Now is the time to set the degrees of freedom for the slave nodes, compelling them to mimic the movements of the master node. So select menu item '**FE Model| Rigid Links| Add| Set Slave DOF**'. A dialogue window will open as below:-

The default settings are all constrained. Set the constraints as you need. Now all future Rigid Links will be created with these slave settings until the settings are changed.

Now select the slave nodes for the Rigid Link, as shown in the example below:-

Selected Nodes (8 off)
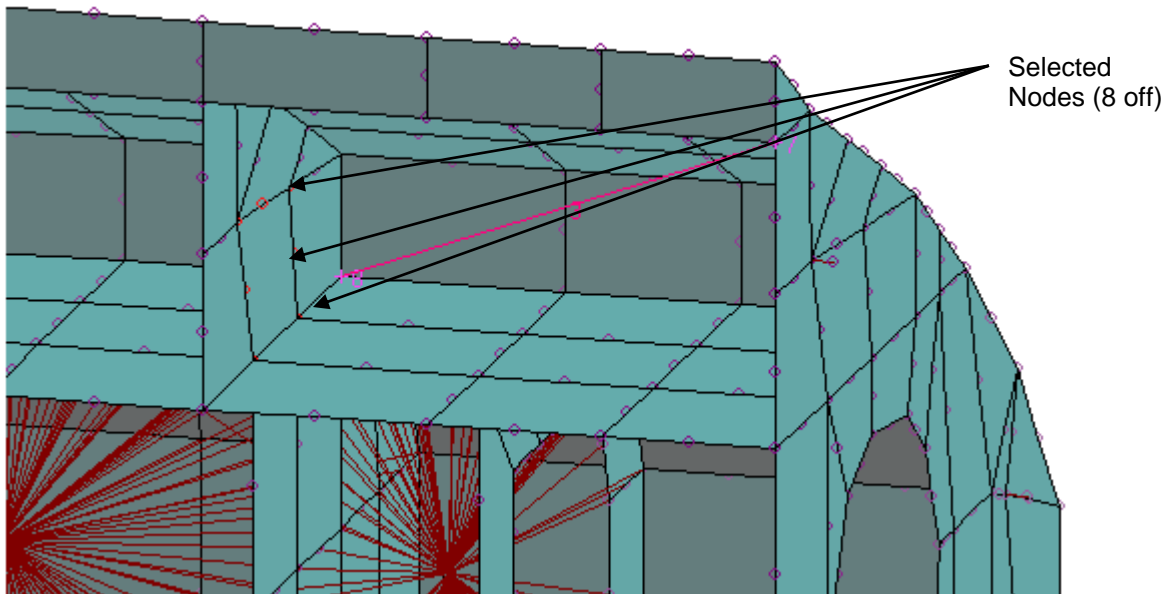
Select Menu Item '**FE Model| Rigid Links| Add| Pick Master Node Location**'. Then interactively pick the location of the master node. The node need not exist, as if there isn't a node at the pick point one will be automatically created. In this example the pick location is at the centre of line 3, where no node existed before the action:-
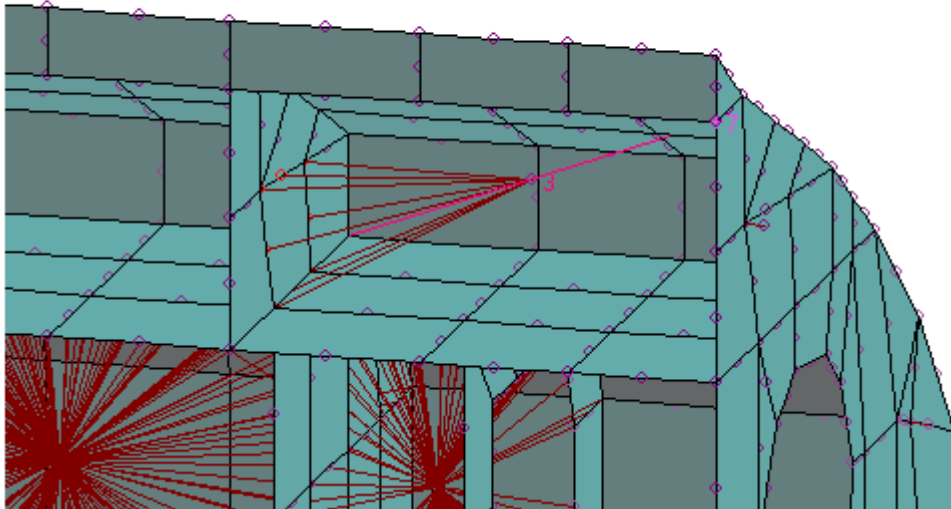
Now a the master node can be loaded to simulate a 'Remote Load', or a Mass element put at the node.

2.4.2   Creating Nodal Pressures

What is a 'Nodal Pressure'?
It's a slightly odd, but very useful, idea that must be credited to Mike Hoey at AIM Aviation (Jecco) Ltd in Bournemouth.
Imagine a compartment with baggage in it. As the compartment is accelerated the baggage will tend to act as a pressure load on the compartment wall. This is fine if the compartment wall is perpendicular to the acceleration, however if the wall is angled compared to the acceleration, then there will be a residual load that is perpendicular to the acceleration. If the baggage was perfectly fluid, then this would create an equal and opposite load on another panel. However real baggage does not behave like this, so you are left with either applying a fake pressure to resolve the resultant load, or ending up with an out of balance residual load. The problem is seen below:-

Upwards component not wanted
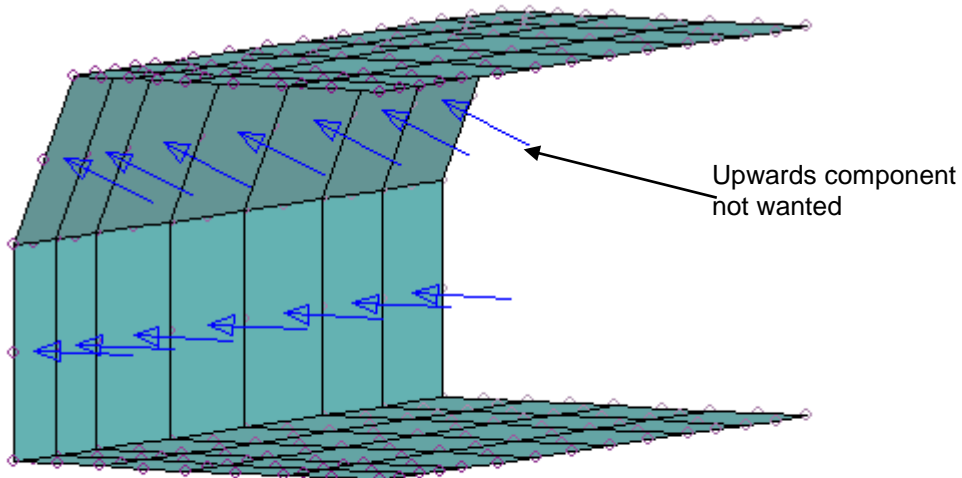
'Nodal Pressures' take the guesswork out of the calculation by applying loads to the nodes around each element in proportion to the projected area of the element in each orthogonal direction. The load is also proportioned in accordance with the element type. Instead of applying a pressure in the above example, a single load is applied. The process is as below:-

Select the elements..
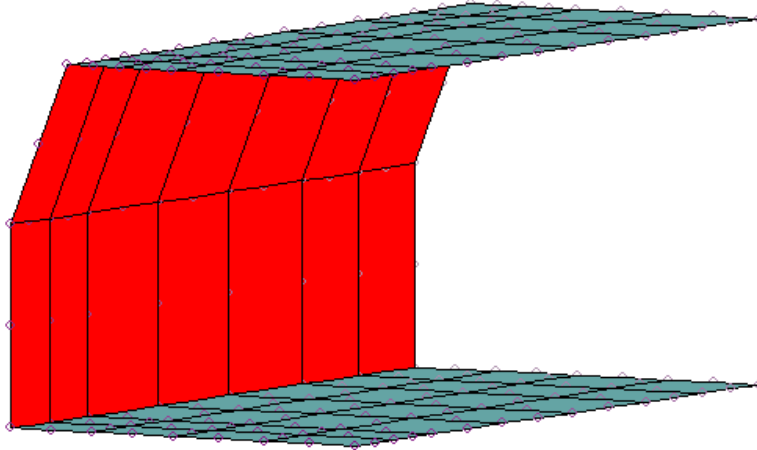
Select Menu Option '**FE Model| Nodal Pressures| Add**' to get the following dialogue window:-
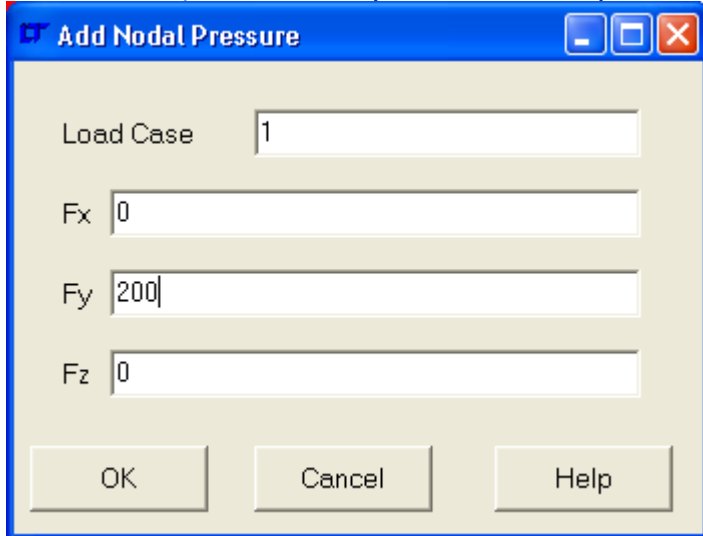
And enter the load desired. Then you will get loads applied to the nodes associated with the elements that were selected:-
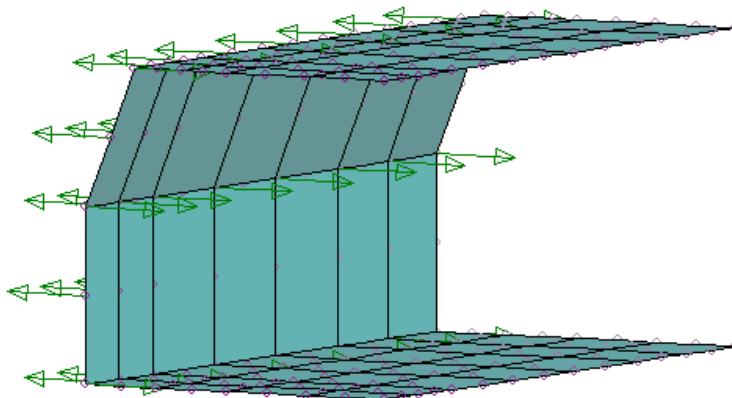
The load on an axis to each element is proportional to its projected area in that axis, so if some of the roof elements above had been selected, no Y direction load would have been applied to them as they have no projected area on the Y axis.

You will notice on parabolic formulation elements that some of the nodal pressure loads are in the reverse direction to the overall load. This is normal and is a consequence of the formulation of the element. The proportional distributions are as below (where 1 = total load to element):-

Formulation	Node Order	Corner Nodes	Midside Nodes
Linear Rectangular	1	1/4	-
Parabolic Rectangular	2	-1/12	1/3
Linear Triangular	10	1/3	-
Parabolic Triangular	11	0	1/3

With these distributions, the element is loaded in the same way as if a pressure were applied to it, only the resultant load need not be perpendicular to the surface. If you don't believe the proportions, apply a pressure to a single element and put displacement constraints on each peripheral node. Analyse the model and look at the reactions.

The use can be extended to any situation where a load needs to be applied to a group of elements. It has the advantage that the total load only need be known. Thus the need to find the area is done away with for Pressures and needing to know the number of nodes in an area is done away with when applying forces.
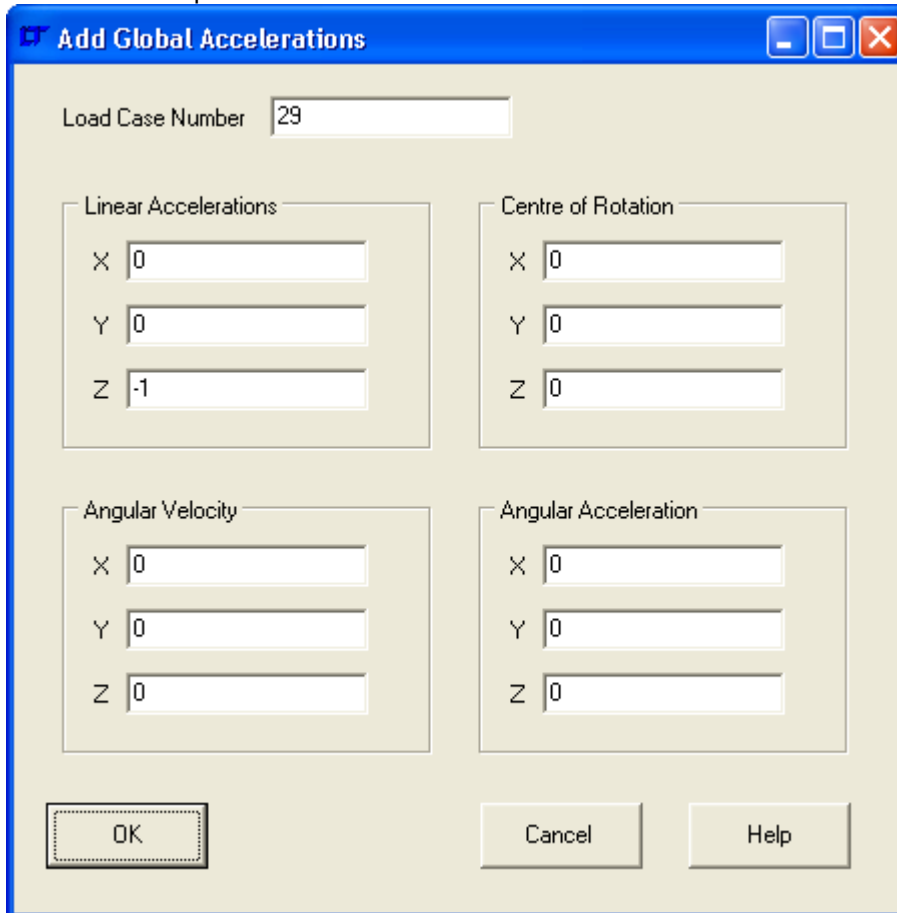
Limitations of Nodal Pressures
1)   Only applies to shell elements.

2)   If all the elements are parallel to a major axis, no nodal pressure load is applied in that axis as the elements have no projected area for the load to be apportioned over.

2.4.3   Creating Accelerations

Accelerations are the finite element analysis method of applying a load to a whole model. If the model has a mass or density, then applying an acceleration to the body will have the same effect as if you were to apply a force at each node equivalent to the mass x acceleration of the elements attached to that node.

To add an acceleration, choose menu option '**FE Model| Accelerations| Add**'. A window similar to the one below will open:-

First off, select the load case number. If '**FE Model| Load Case| Auto Create**' is ticked, then when you accept the acceleration details, a new load case definition set will be created if one does not already exist.

If you are dealing with linear accelerations, ensure that the 'Angular Velocity' and 'Angular Acceleration' values are zero. The acceleration is in length/time^2, and must be consistent with the rest of the model.

Remember that the angular velocity and acceleration values must be in radians/time or radians/time^2 as appropriate.

3.   How To… Guide

3.1   General Information

This 'How To…' Guide is intended to show how a series of tasks would be approached. It is not fully comprehensive, as that's impossible, but it should give pointers that will be applicable over a wide range of requirements. This section only covers items not covered in the 'Quick Start Guide', so basic FE model creation is not covered.

3.2   How To… Index

3.2.1   How to Open a NISA File

This is probably the shortest entry in the 'How to..' guides.

Once the main program is running press menu option '**File| Open**' and select the 'Nisa Input File (*.nis)' filter. Select the file you wish to open.

There are no options, and you cannot import or merge a NISA file. If you wish to merge a NISA model, it will have to be by means of merging Modeller Data Files (.mdf) with menu option '**File| Merge**'.

3.2.2   How to Use Groups and Layers

Groups and layers provide a convenient way of aggregating objects that have something in common so that they can be plotted, selected or erased in one easy action.

Points, Lines, Surfaces, Nodes, Elements and Rigid Links can be placed on a layer. In fact, when these entities are created, they are always put on the current layer – which is 'Layer 0' if no other layer has been defined. For rendering packages the Modelling Studio 'Layer' corresponds to the rendering package 'Material'.

Elements can also be put into a group, which enables a second way of aggregating elements that have something in common. For rendering packages the Modelling Studio 'Group' corresponds to the rendering package 'Group'.

To give an example of layers and groups, see the model of a figure below (used as an example where layers and groups have been used to a great extent). Initially the figure is shown as is, then coloured by Layer, and finally coloured by Group:-

Normal View:-

Coloured by Layer:-
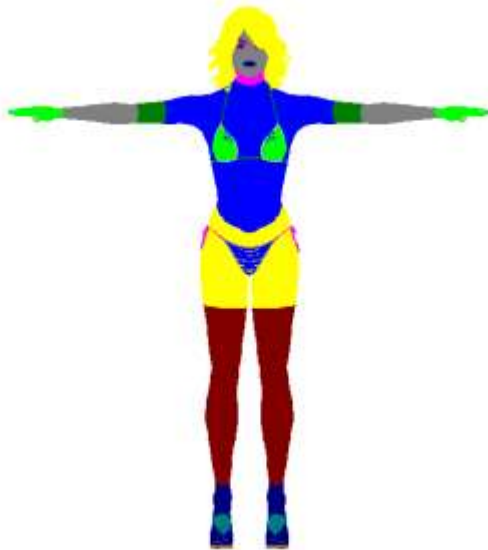
Coloured by Group:-
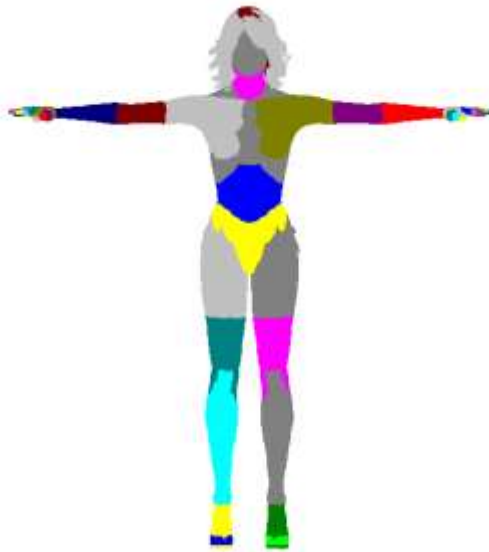
As may be seen, the layer and group boundaries need not occur at the same place.

OK, what can Layers and Groups be used for?

Assume you have a model of an assembly of a number of parts. It's sensible to put the elements in each part into the same group. Take the assembly shown below (used on a number of EPS transport containers, imported from COSMOSWorks as the model was available!). There are six parts to the assembly, and each part is in its own group:-
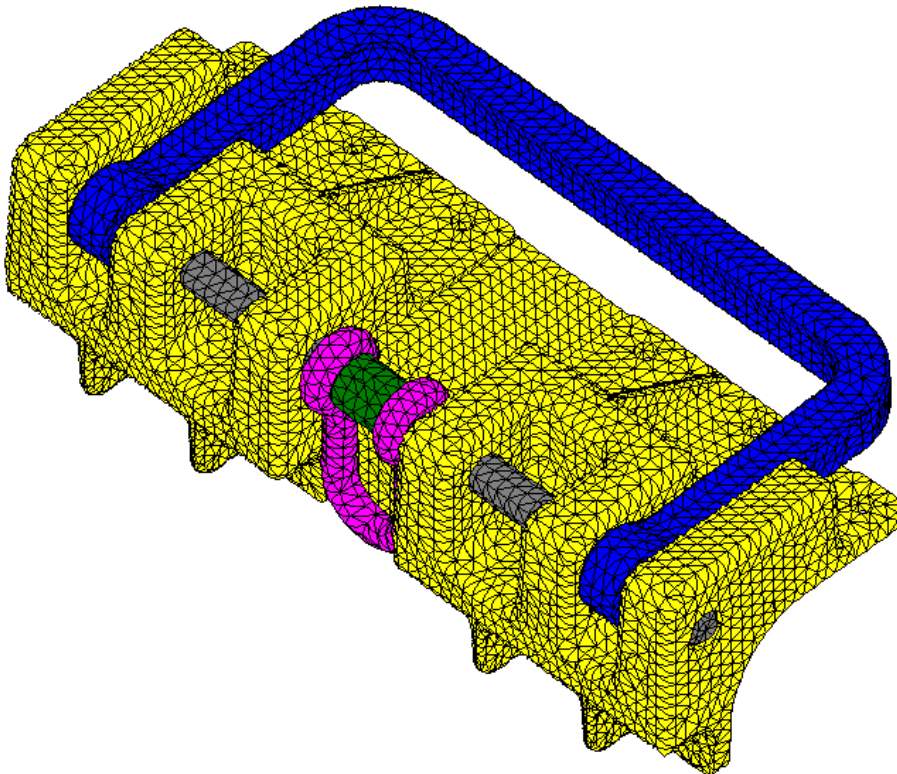
Now, to erase the shackle the only command required is '**View| Erase| Elements| Erase by Group Name**' and select the 'Shackle' group. This saves a great deal of messing about when wanting to view certain parts of the model without the clutter of other parts.

Groups and layers can also be used to select parts of the model to enable transformations (Copy, Move, Mirror, Rotate, Scale), or modifications (like change material) to be carried out with the minimum of fuss.

Groups and Layers are also used for rendering a picture. For this particular application see the 'How to Texture Map a Model' guide.

How to create layers and groups
Before initially creating entities (Points, Lines, Surfaces, Nodes, Elements & Rigid Links), the current layer can be created and set with Menu Option '**View| Layers| Layer Controls**'. Add layers by pressing the 'Add Layer Button'. As each layer is created, click on the default name to rename it, or wait until afterwards, click on the layer to rename, and right-click to get the 'Rename' pop-up menu item. To set the current layer, double click on the layer name. This is shown below:-
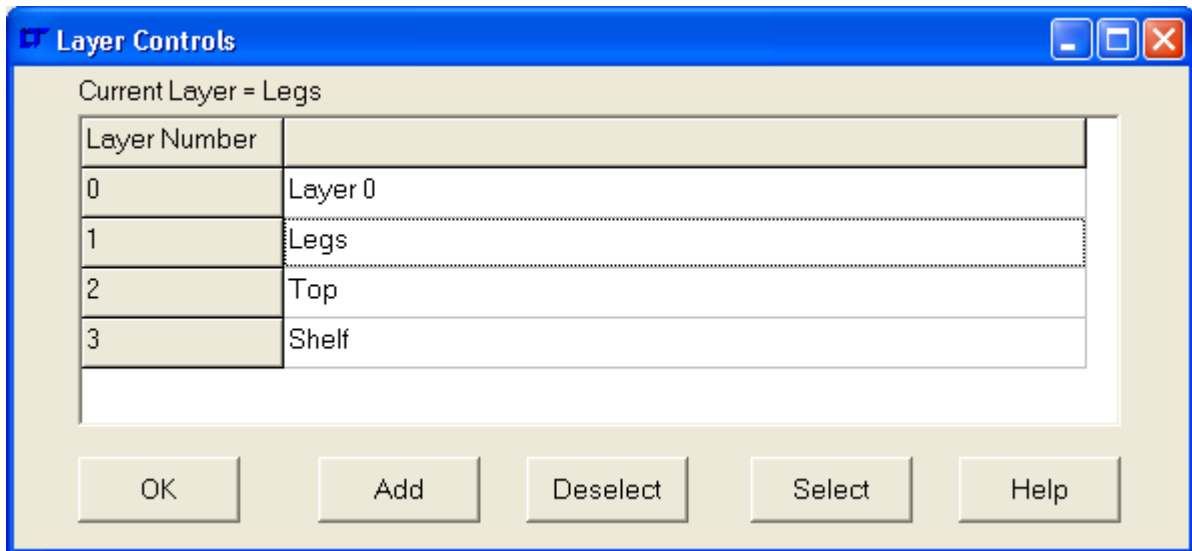
From now on, all newly created entities will be created on the current layer.

Carry out the same procedure with 'Groups' before the first elements are created using '**FE Mode| Elements| Groups| Group Controls**'.

When you wish to allocate existing entities to a new layer, set the current layer to the one the entities are to be allocated to with '**View| Layers| Set Current Layer**'. Then select the entities you wish to change layer, and select '**View| Layers| Add to Current Layer**'.

When you wish to allocate existing elements to a new group, set the current group to the one the elements are to be allocated to with '**FE Mode| Elements| Groups| Set Current Group**'. Then select the elements you wish to change group, and select '**FE Mode| Elements| Groups| Add to Current Group**'.

3.2.3   How to Plan a Model

This little section tries to do the impossible, it is hoped that after reading this section you will think of everything you need to before you create and mesh a model. Having set an impossible task, let's make *some* effort before failure occurs… This section is really advice based on 18 years work creating FE models with tools that don't permit easy alteration of the model once it's created. The Modelling Studio is better at modification of the model than most of the tools used during those 18 years, but it's better and quicker to get it right first time.

When you start to model, the **last** thing you need is a computer – it certainly **isn't** the *first* thing you need.

To begin with, plan the model on a piece of paper, the back of an envelope or whatever. Do not launch into the modelling half cocked.
1)   Get an adequate drawing of the object you wish to model. This may be an engineering drawing or a marked-up sketch.
2)   Simplify the drawing as much as possible.
3)   Mark out the key points of the model: edges, attachment points, openings etc.
4)   Work out the centrelines of any flat panels.
5)   Ignore fillets wherever possible unless they are a significant size relative to the model.
6)   Now try to work out centres of rotations of curves. Divide the circles up into sensible arcs of rotation with straight segments – at the moment the Modelling Studio can't cope with curves.
7)   Mark up areas that will be loaded by the boundary conditions.
8)   Now try to copy key features across the model to form a matrix of lines – do as few as possible to cover the model. These lines will be the edges of the surfaces that will be meshed.
9)   Now work out how many elements will be needed on each surface. Ideally you want to have rectangular elements with an edge length ratio of 1, or triangular elements with all internal angles being 60°. Aim for perfection, but be prepared to compromise.
10)   Remember that, for FE Models, the numbers of elements on each edge of each adjoining surface must be the same. It may be necessary to create surfaces to mesh on the fly after some elements have been created.

Now that you've got it all out in rough on the paper, it's possible to start modelling the job on the computer. With the planning already done, it will be quicker and easier to progress through the model.

The sketch below was one used to plan the model for a simple wardrobe on a Boeing 747.
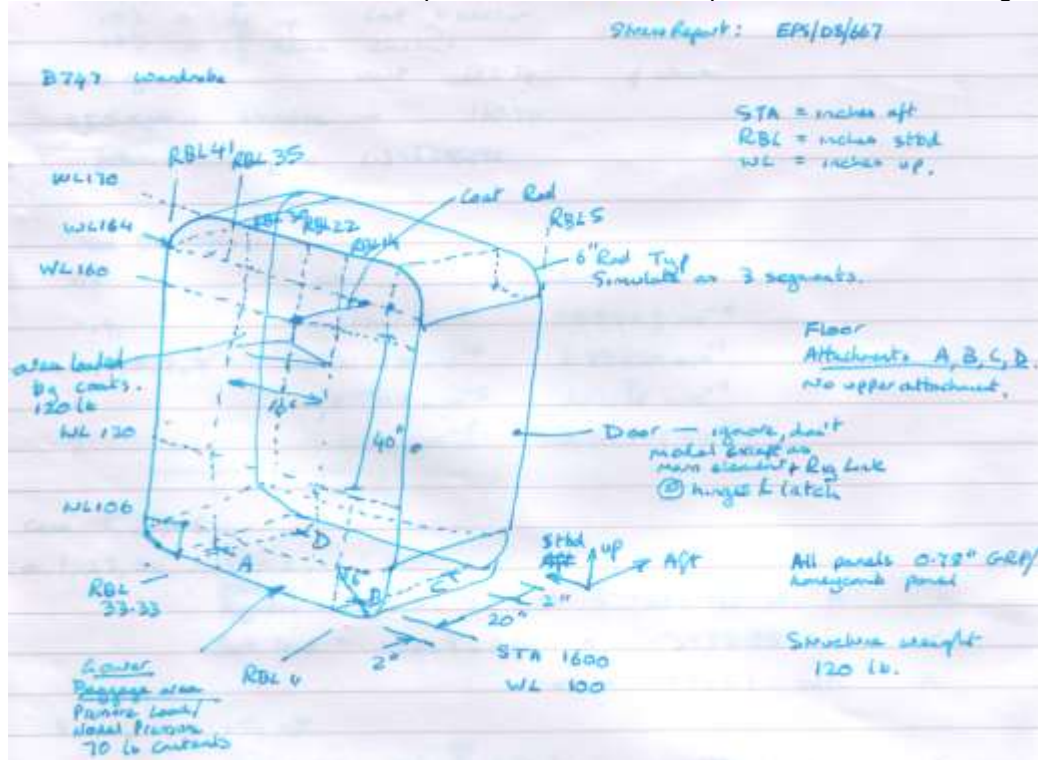
The following geometric model was the result of around 20 minutes modelling:-
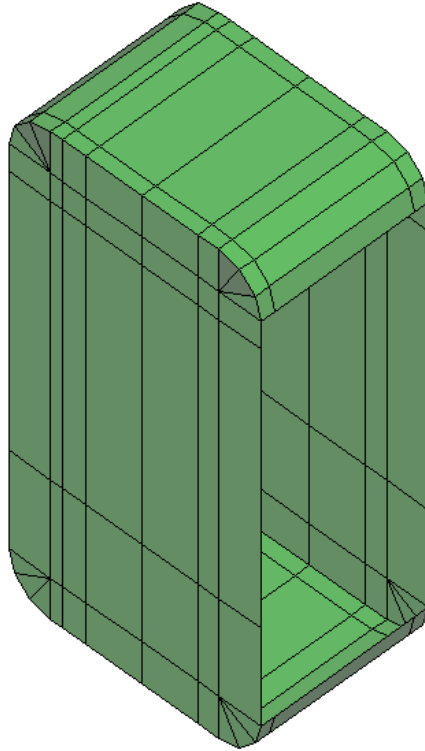
The following FE model was created:-
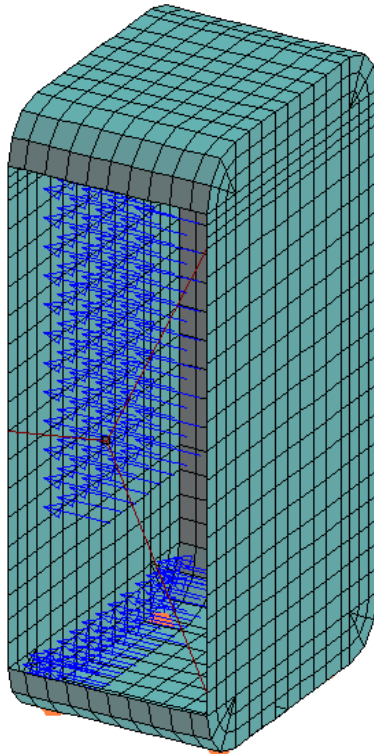
Due to the initial planning on the model, there was no need to go back & delete or modify anything, and it took the minimum of time to create.

3.2.4   How to Verify a Model

Model verification is, to a certain extent, a dark art. It is only possible to give suggestions as to how to go about verifying a model. No two models will have the same potential problems or errors. In some ways a checklist approach is the most widely applicable, as below:-

1)   Merge Nodes. To merge nodes, select all the nodes, then choose menu option '**FE Model| Nodes| Merge**'. You will be asked for a merge tolerance on the nodes. If working in inches, 0.001 may be appropriate, while 0.1 may be suitable for a model dimensioned in millimetres.

2)   If the model is a shell element model, check that all the shell normals are pointing in similar directions on the same surface using '**View| Display Options| Elements| Show Normals**'. Select any elements with inconsistent normals and choose menu option '**FE Model| Elements| Modify| Reverse Normals**'. When complete turn off the display of element normals.

3)   Check the materials are correct by selecting '**View| Colours| Elements| Colour by Material**' and turn element labels on using '**View| Labels| Elements**'. The Modelling Studio now displays the material number on each element. The overall colours will show whether the materials are correct.

4)   Check the properties are correct by selecting '**View| Colours| Elements| Colour by Property**' and turn element labels on using '**View| Labels| Elements**' if it hasn't already been switched on. The Modelling Studio now displays the property number on each element. The overall colours will show whether the properties are correct. Turn 'Colour by Property' back to 'Infinite Light Shaded' when finished.

5)   If boundary conditions are specified these can be checked at a glance by '**View| Erase| Boundary Conditions| All**' and then selecting '**View| Plot| Boundary Conditions| Range by Case Number**', and typing the case you wish to see. This will give an overview of what is applied to the model. If what you see isn't correct, obviously it should be amended.

6)   If the materials, properties and lamination sequences have been defined, then they can be verified using '**FE Model| Materials| Inspect**', '**FE Model| Laminations| Inspect**', '**FE Model| Lamination Angles| Inspect**' and '**FE Model| Properties| Inspect**'.

7)   If orthotropic materials or laminates are used for the model, check the element material directions are correct using '**View| Colours| Elements| Show Material X Direction**'. The resulting plot will show the 0° orientation of the material in each element. If there are some elements that are incorrect due to being shell elements or laminate elements with fibre direction defined by the 'Right Hand Rule about shell normal', select the elements that are incorrect and choose menu option '**FE Model| Elements| Modify| Shift Element Origin**', otherwise correct the element definition in the laminate definition. See the 'How to create a Laminate' User Guide for more information on this.

8)   If accelerations have been defined for various load cases, then they can be inspected using '**FE Model| Accelerations| Inspect**'.

9)   Perform an 'Analysis Readiness Check' using menu option '**FE Model| Analysis Readiness Check**'. This check opens a window that reports on a number of aspects of the model. It checks to see if there are nodes that should be merged, whether all the required material, property and laminate information is defined. Also, whether the load cases are properly constrained or whether any loads are applied in the load cases. There are also some basic checks to see whether the elements have duplicated nodes in their definition. See the Command Reference for full details of the checks performed.

There is a feature which would help verify a model that *may* be added at some point in the future. It is:-
1)   Boundary Line Check, to show which edges are not shared between two elements.

3.2.5   How to Input FE Information

It's possible that it isn't entirely clear how to input FE Model Information, so this 'How To..' guide attempts to remedy the problem.

Nodes

Nodes can be created directly using the **Add Node** tool button.
Nodes can be created indirectly by meshing geometry using the **Mesh Points**, **Mesh Lines** or **Mesh Surfaces** tool buttons.
Nodes can also be created indirectly as a consequence of creating a Rigid Link. If there is no node at the master node position, a node is created there.

Elements

Elements can be created by meshing points, lines or surfaces. This may be done using the **Mesh Points**, **Mesh Lines** or **Mesh Surfaces** tool buttons.
Elements can also be created by nodal picks. The element type to be created is selected using '**FE Model| Elements| Add| Settings**'. Then the nodes needed for the particular element are picked using '**FE Model| Elements| Add| Pick Nodes**'. The nodes must be picked in the order shown in the 'Element Library' section of this User Guide. The 'Pick Nodes' method of adding elements is very tedious, but it is possible to add solid elements this way.

The preferred element types are:-
4	3D Brick Elements (at the moment can only be *created* using 'Pick Nodes' method.)	
12	3D Beam Elements	
14	3D Strut Elements	
17	3D Spring Elements	
20	3D Shell Elements	
26	3D Mass Elements	
32	3D Composite Shell Elements	
33	3D Composite Sandwich Shell Elements	

Rigid Links

Rigid Links are created by selecting the slave nodes, and choosing menu option '**FE Model| Rigid Links| Add| Pick Master Node Location**'. Then the position of the master node is picked. If there is no node at the master node position picked, a master node is created there.

Materials

Materials are added using menu option '**FE Model| Materials| Add**'.

Laminations

Laminations are added using menu option '**FE Model| Laminations| Add**'.

Lamination Angles

Lamination Angles are added using menu option '**FE Model| Lamination Angles| Add**'.

Properties

Properties are added using menu option '**FE Model| Properties| Add**'.

Load Cases

Load Cases are added directly using menu option '**FE Model| Load Case| Add**'.
Load Cases may be added indirectly by setting menu option '**FE Model| Load Case| Auto Create**', and then every new load case referenced by a boundary condition will automatically create a new Load Case.
Load Cases are created indirectly using the 'Load Combiner'.

Displacement Constraints

Displacement Constraints may be created directly by selecting the nodes to have the displacement constraint, and then selecting menu option '**FE Model| Displacement Constraints| Add**'.
Displacement Constraints are created indirectly by the 'Load Combiner'.

<u>Pressures</u>
Pressures may be created directly by selecting the elements to have the pressure applied, and then selecting menu option '**FE Model| Pressures| Add**'.
Pressures are created indirectly by the 'Load Combiner'.

<u>Nodal Pressures</u>
Nodal Pressures may be created directly by selecting the elements to have the Nodal Pressure applied, and then selecting menu option '**FE Model| Nodal Pressures| Add**'. The load to be applied is the total load to be divided up over the all the nodes on the selected elements. The load at each node is proportional to the area of the elements attached to the node.
Nodal Pressures are created indirectly by the 'Load Combiner'.

<u>Forces</u>
Forces may be created directly by selecting the nodes to have the Force, and then selecting menu option '**FE Model| Forces| Add**'. The force to be applied is the force per node, not the total force.
Forces are created indirectly by the 'Load Combiner'.

<u>Accelerations</u>
Accelerations are added directly using menu option '**FE Model| Accelerations| Add**'.
Accelerations are created indirectly by the 'Load Combiner'.

3.2.6   How to Create a Laminate

Laminate Creation has been developed as a NISA function. MYSTRAN V4 and above can use laminates too, though the laminate creation process may seem convoluted for a MYSTRAN user.

Laminate creation feels like an afterthought tacked on to NISA after most other element details had been set and could not be changed, as a result it's a little bit less obvious than one might expect. The Modelling Studio was initially aimed at the NISA package for laminates, so it makes the NISA process as easy as possible.

For a laminate element the 'Element Property' is not applicable – any number can be used. The 'Element Material' number refers to the lamination number used to define the element and not to any material definition. REMEMBER THIS! Didn't mean to shout, but do remember that one piece of information.

The best thing to do is to sketch a cross section of the laminate and a table of thicknesses, fibre angles and materials. We will take an example of a very impractical laminate, a three layer laminate with a thick glass re-inforced plastic skin, a Nomex honeycomb core and an aluminium backing plate. As the middle of the laminate is a honeycomb, this will only apply to elements of type 33 (3D Laminated Sandwich [Honeycomb]), however the principal applies to all laminate elements:-
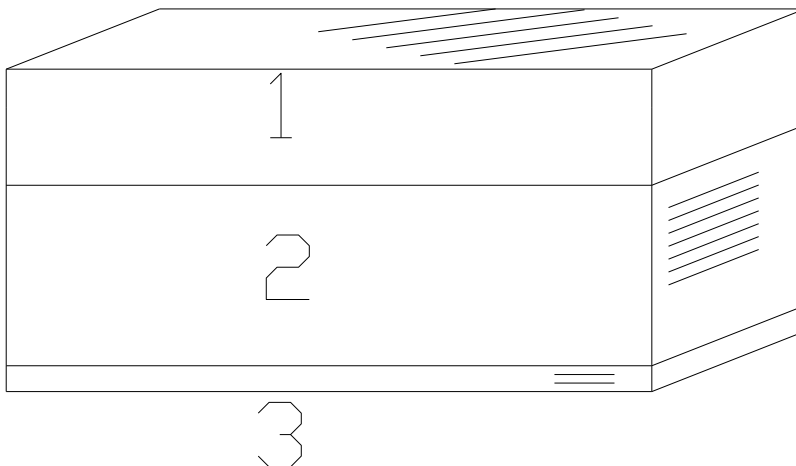
Layer	Thickness	Fibre Angle	Material
1 (top)	2.5mm	-45°	GFRP
2	7.5mm	90°	Nomex Core
3 (bottom)	1mm	0°	Aluminium

Now a property has to be created for each different thickness, a lamination angle for each different fibre angle and a material for each different material. Armed with this table it's possible to assign a property for every thickness, we'll do it starting at 10, with increase in thickness. The fibre angles will start at 20 with increase in angle, and the material numbers will start at 1, so the table assignments become:-

Layer	Thickness	Fibre Angle	Material
1 (top)	2.5mm = Property 11	-45° = Lamination Angle 20	GFRP = Material 1
2	7.5mm = Property 12	90° = Lamination Angle 22	Nomex Core = Material 2
3 (bottom)	1mm = Property 10	0° = Lamination Angle 21	Aluminium = Material 3

Now, these assignments are arbitrary, which doesn't matter one iota. The *only* important thing is to keep track of these assignments, which is why writing down the table is so necessary.

Now, assume this is laminate 14. Select Menu Option '**FE Model| Laminations| Add**', and add the table data as below:-
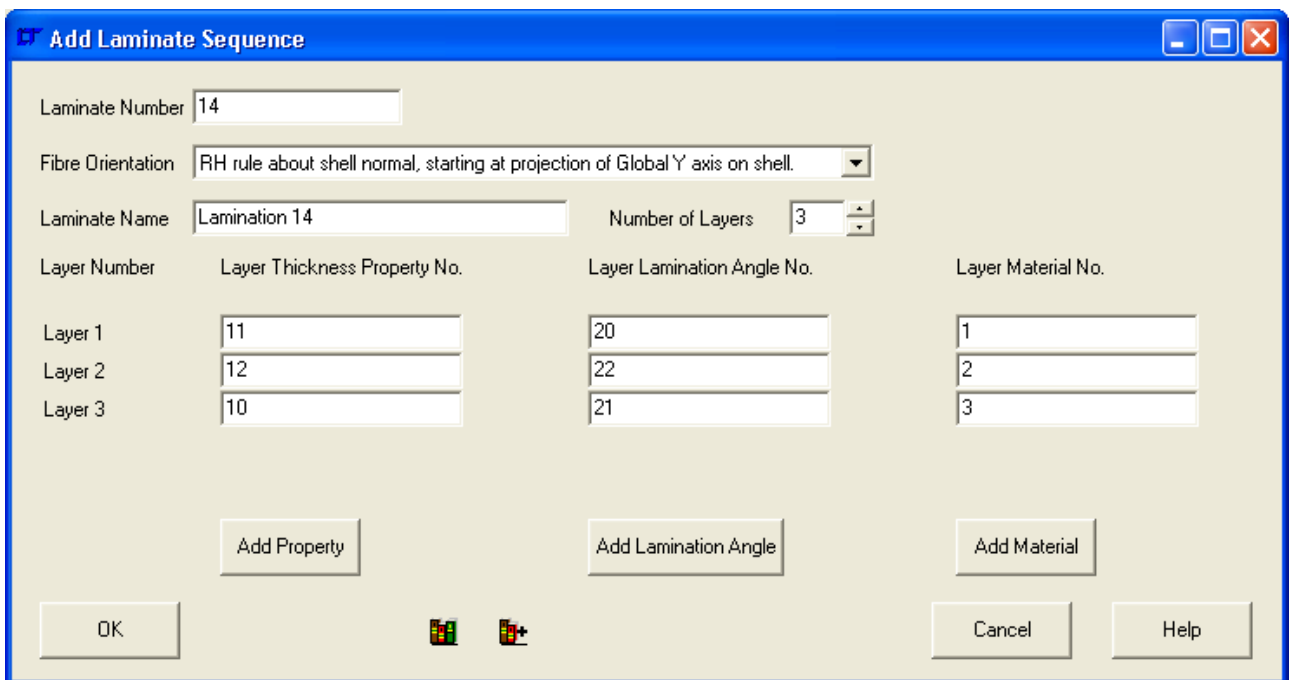
The buttons 'Add ******' buttons enable you to add properties, Lamination Angles or Materials without using the normal Menu commands of :-
> '**FE Model| Property| Add**'
> '**FE Model| Lamination Angles| Add**'
> '**FE Model| Material| Add**'

You can still add the properties, angles and material the old fashioned way if you wish.

The 'Load from Library' button opens a file with all the materials, properties and lamination angles stored within the file to create a consistent laminate with no other files being needed – very useful given how much data needs to be entered and cross referenced! Using the 'Load from Library' button does away with the need for a table etc. You just have to work it all out once. When you load a library laminate in, the Modelling Studio checks the existing model for the correct materials, lamination angles and properties, and only creates new data if it will not duplicate existing data. The laminate material, angle and property numbers in the file are updated to avoid clashing with any existing data – so you need not worry that the laminate you are loading was saved with data reference numbers that could over write your existing model.

Now other things that you have to remember and deal with:-

<u>Fibre Orientation</u>
The 0° axis for the fibre orientation is specified in the fibre orientation combo box. The 0° axis is in element local axes or global axes.

For NISA users:-
     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. (ie not in the XZ plane)
         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!

     What is often done at EPS is to have two identical laminate sequences, one with the fibre orientation defined by the 'RH Rule about shell normal starting at projection of Global X axis on shell.' And the second with the fibre orientation defined by the 'RH Rule about shell normal starting at projection of Global Y axis on shell.'. Then any panels only in the YZ plane are set to the second laminate type. This prevents many laminate definition problems.

     Try to avoid the 'Right Hand Rule about shell normal' option for the fibre orientation, as this will depend on how you meshed the model. The same applies to Display III – meshing this way with 'Automesh' leads to whirls in the fibres because in each 2 x 2 group of 4 elements the element orientation rotates by 90° (been there, done that, been thoroughly confused by the results for ages!). The picture below shows the results of a model where the 'Automesh' caused fibre whirls:-
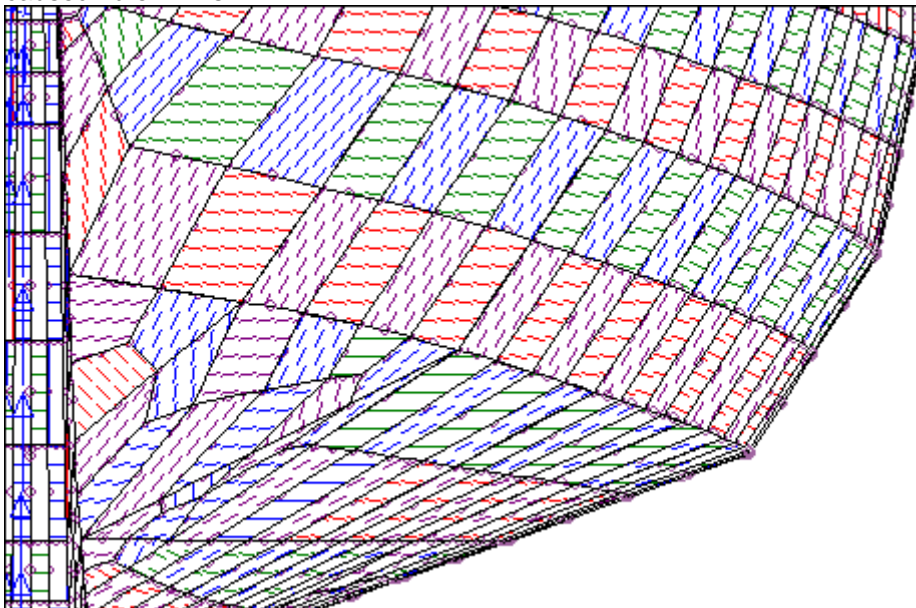
This is *not* good! However in models with isotropic materials only, there should be no effect.

For MYSTRAN users:-
> MYSTRAN users have no option; you must use the 'Right Hand Rule about shell normal' option for the fibre orientation. The fibre orientation depends on how you meshed the model. The fibre directions always follow the 'U' direction of the surface you meshed.

To verify that the fibre orientations are in the correct direction, use the menu option '**View| Colours| Elements| Show Material X Direction**'. The fibre directions are then drawn on the screen, with different colours representing different angle quadrants:-

Blue	0°	to	90°
Green	91°	to	180°
Purple	181°	to	269°
Red	270°	to	359°

When viewing the element on screen, 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. It is, however, good enough to see if there is a problem.

If the fibre angle is wrong, there are several options you can take. If you are a NISA user, select a different fibre orientation definition method for the laminate – or define a new laminate. If, however you are using the 'Right Hand Rule about shell normal' option for the fibre orientation (NISA or MYSTRAN users), you can change the fibre orientation of an element by selecting the element, then choosing menu option '**FE Model| Elements| Modify| Shift Element Origin**'. This option rotates the element origin by one corner anti-clockwise around the element normal. Using this method it is possible to manually re-align the fibre orientations on shell elements (not point, line or solid elements).

Material (NISA Only)
When working with Laminates of type 33, the core material (honeycomb or foam) MUST have the value of EX set to 0. This is tells NISA to treat the material differently from the other ones in the laminate.

Tapered Laminates (NISA only)
Due to the way NISA requires the thickness to be defined at each node for each layer it is possible for NISA to model tapered laminates. This is rarely required (once in 20 years…), and VERY time consuming to do as each tapered element really requires a new laminate definition set. The 'Load from Library' tool allows you to simplify the process somewhat by editing the thickness in the .mlf file – at the expense of creating a multitude of properties. The element orientation will need to be found by inspecting the element, noting the node numbers and then cross referencing against the thickness required at each node. There is no simpler method of doing it - Sorry!

3.2.7   How to Use the Load Combiner

The load combiner is a very powerful tool; however, the best way to use it is not immediately obvious. This section of the 'How to…' guide attempts to explain how to best use the tool.

The load combiner comes into its own when you have multiple load cases that can be defined as factors of other load cases. Take the example of doing stresses for aircraft interior components; it is usually important to know the stresses on a component in up to 27 load cases. All the load cases can be generated from combinations of loads generated for a 1 g_n acceleration in the aft, starboard and up directions.

The best way to show how to use the load combiner is by example, though there is no real need to work through it alongside this 'How to…' guide.

Let's define the example:-

We have a galley that is due to go in an aircraft, and need to know both the interface loads from the galley to the airframe and the structural stresses when this occurs. The structural components of the galley are shown below:-
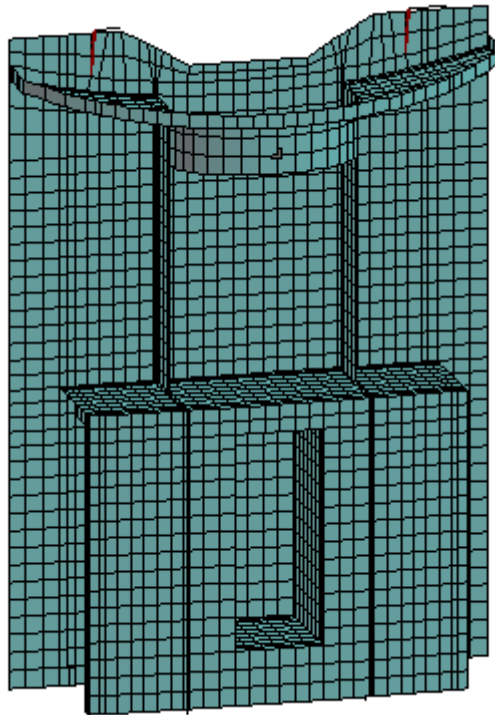
Now to define the load cases that have to be considered (all loads and combinations up to a max resultant of 9 g_n):-

Case	Forward	Aft	Port	Starboard	Up	Down
1	9 g_n	-	-	-	-	-
2	-	1.5 g_n	-	-	-	-
3	-	-	3 g_n	-	-	-
4	-	-	-	3 g_n	-	-
5	-	-	-	-	3 g_n	-
6	-	-	-	-	-	6 g_n
7	8.485 g_n	-	3 g_n	-	-	-
8	8.485 g_n	-	-	3 g_n	-	-
9	8.485 g_n	-	-	-	3 g_n	-
10	6.708 g_n	-	-	-	-	6 g_n
11	7.937 g_n	-	3 g_n	-	3 g_n	-
12	7.937 g_n	-	-	3 g_n	3 g_n	-
13	6 g_n	-	3 g_n	-	-	6 g_n

Case	Forward	Aft	Port	Starboard	Up	Down
14	$6\ g_n$	-	-	$3\ g_n$	-	$6\ g_n$
15	-	$1.5\ g_n$	$3\ g_n$	-	-	-
16	-	$1.5\ g_n$	-	$3\ g_n$	-	-
17	-	$1.5\ g_n$	-	-	$3\ g_n$	-
18	-	$1.5\ g_n$	-	-	-	$6\ g_n$
19	-	$1.5\ g_n$	$3\ g_n$	-	$3\ g_n$	-
20	-	$1.5\ g_n$	$3\ g_n$	-	-	$6\ g_n$
21	-	$1.5\ g_n$	-	$3\ g_n$	$3\ g_n$	-
22	-	$1.5\ g_n$	-	$3\ g_n$	-	$6\ g_n$
23	-	-	$3\ g_n$	-	$3\ g_n$	-
24	-	-	$3\ g_n$	-	-	$6\ g_n$
25	-	-	-	$3\ g_n$	$3\ g_n$	-
26	-	-	-	$3\ g_n$	-	$6\ g_n$

Just a few… We won't work through them all as that would be *extremely* tedious.

This is one case where automatic load case creation would be a disadvantage, so make sure that menu option '**FE Model| Load Case| Auto Create**' is not ticked.

Now, for the sake of this example, assume that the aft load is a pressure load and is applied to the same elements as the forward load. The starboard load is by means of forces and is applied to the same nodes as the port load. The up load is by nodal pressures applied to the same elements as the down load. Also, in all cases the displacement constraints are the same.

Right… what to do now?

We have 26 load cases we need to create, but this can be done by combinations of just three load cases, $1\ g_n$ Aft, $1\ g_n$ Starboard and $1\ g_n$ Up.

First add the displacement constraints on load case 101.
Add the $1\ g_n$ aft pressure loads and $1\ g_n$ aft acceleration on load case 101 too.
Add the $1\ g_n$ starboard forces and $1\ g_n$ starboard acceleration on load case 102.
Add the $1\ g_n$ up nodal pressures and $1\ g_n$ up acceleration on load case 103.
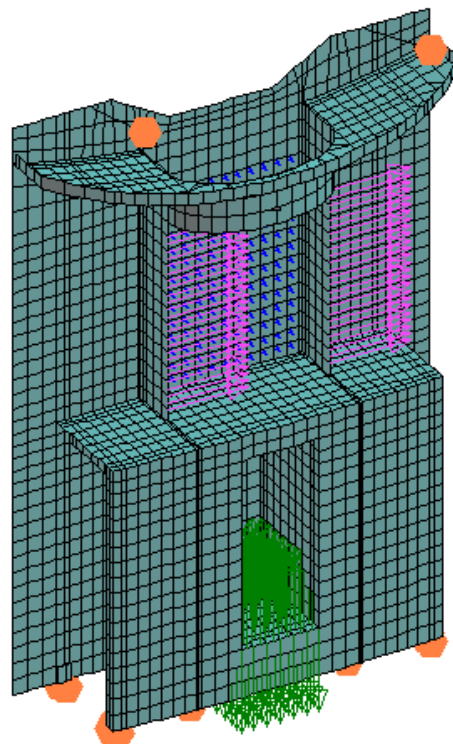
Now use the load combiner to create load case 1. Select menu option '**FE Model| Load Combiner**', and make the window look like the one below:-
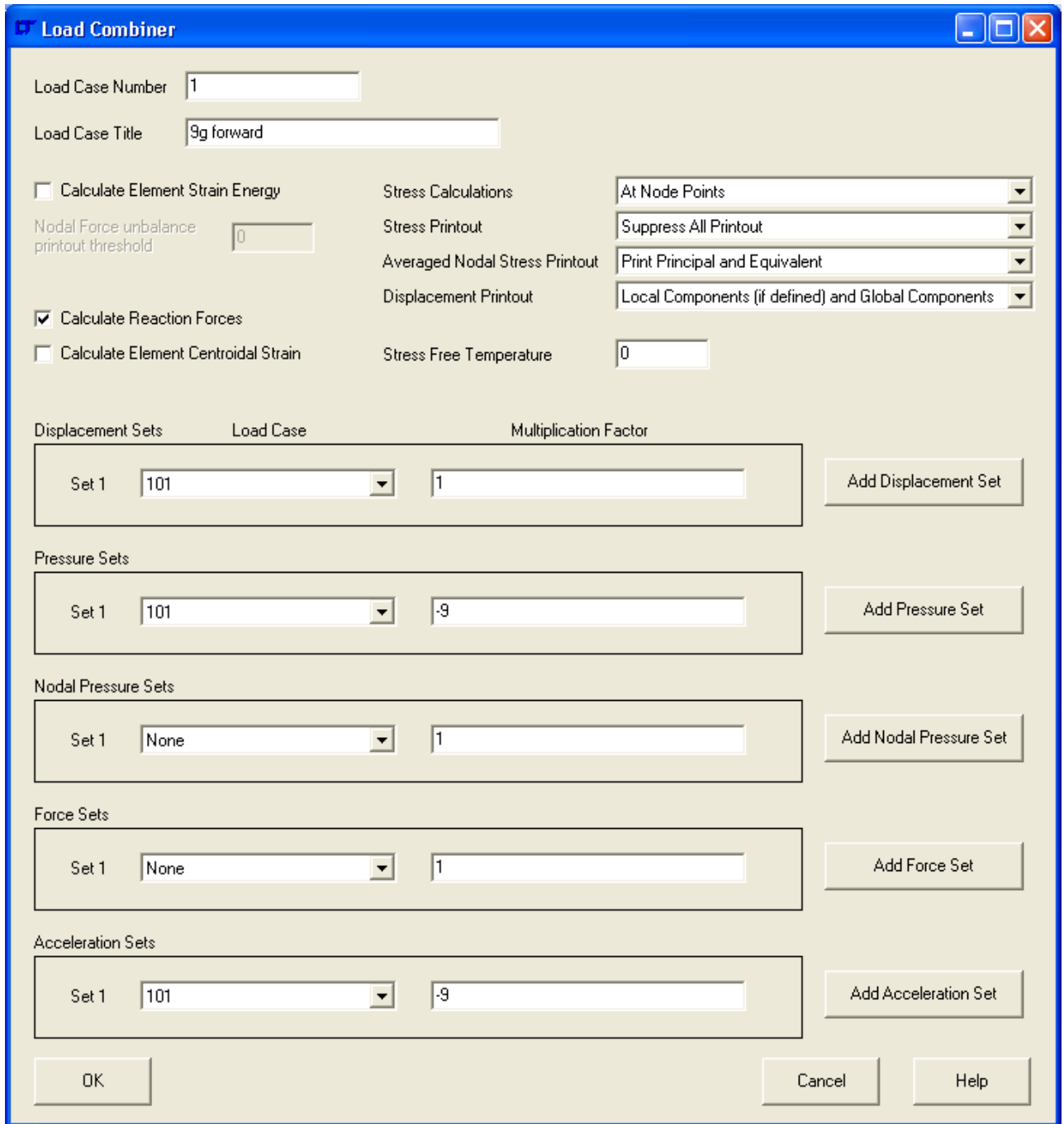
Press OK when it looks correct. The tool creates a new load case with all the individual displacement constraints, pressures, nodal pressures, forces and accelerations in the same way as if they had been created manually. The multiplication factors etc cannot be amended after 'OK' is pressed.

Now on to load case 2. Select menu option '**FE Model| Load Combiner**', and make the window look like the one below. This time you will see that load case 1 is an option to be combined. Don't bother with it.
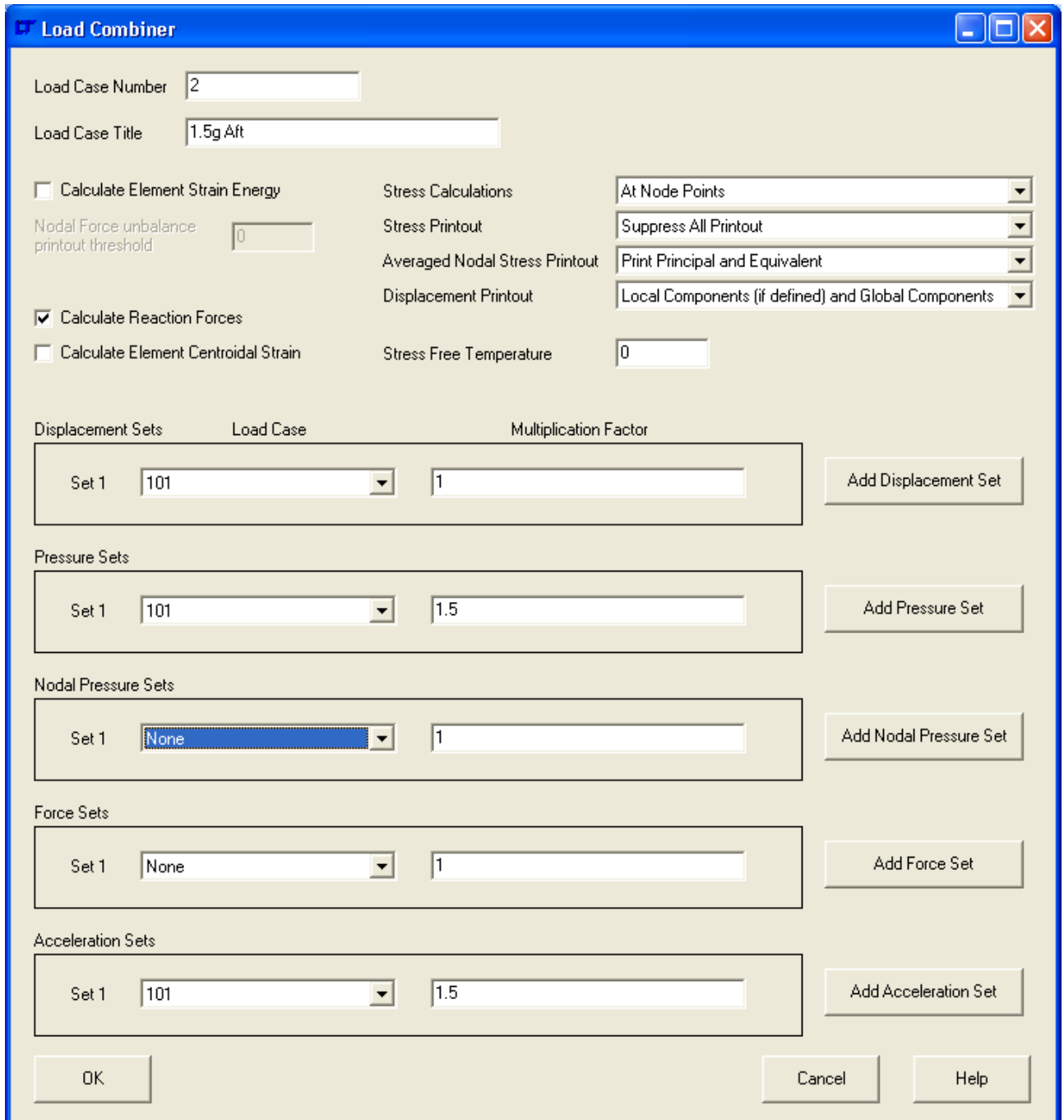
Once again press 'OK' when you're happy. Get the idea?

Now to consider a combination load case. Let's take case 14, 6 g_n forward, 3 g_n Starboard and 6 g_n down. This is all pretty straight forward until we get to the acceleration. Here we need a combination of accelerations from sets 101, 102 and 103.

But there's only one place to put the acceleration!
Never fear.

When you need to add an acceleration set, press the 'Add Acceleration Set' button, and you will get a new load case and multiplication factor appear in the acceleration sets. [This applies to all the 'Add ***** Set' Buttons.] Don't worry if you press the button too often – choose 'None' as the Load Case

when this happens. Enter the appropriate acceleration sets and multiplication factors, and you will get the window below:-
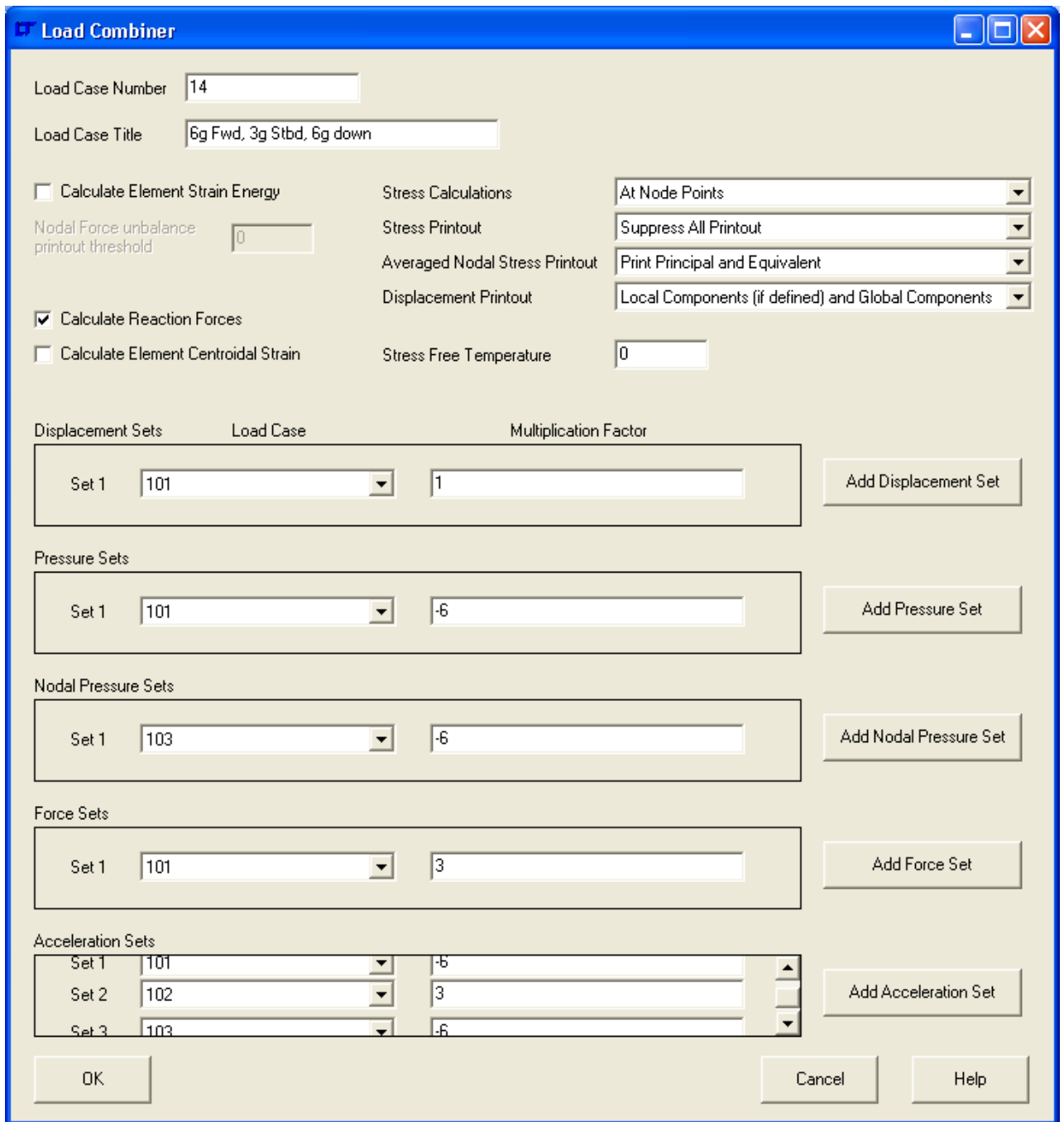
When the analysis file [whether it's NISA or MYSTRAN] is exported, because there is no 'Load Case' definition card for load cases 101, 102 and 103, they are not exported, and none of the boundary conditions associated with them are passed to the analysis data file.

There. That's how the tool works. Not bad really. Creating 26 load cases is still a slog, but not as bad as doing everything for each load case (been there, done that, and hated every second!).

3.2.8   How to Model for Rendering

If you are someone who just wants to create a model to render, you are could well be slightly baffled by now. 'What use is all this to me?' is a question that's probably near the forefront of your mind. This little section tells you what you do and don't need to care about.

1)   The first thing you need to know is that the geometry of the model is important. Create surfaces in the same way as is required for a FE model.

2)   The model needs to be meshed – none of the geometry is exported to a Wavefront Obj file. The quality of the mesh need not be too high (say 3 x 3 meshing) unless you really need to deform it to make morph targets. Just keep the elements as 'Linear' formulation (the default) as the triangular elements exported from 'Parabolic' formulation elements sometimes show as stressed areas in rendering packages. The FE material and properties specified during meshing are irrelevant.

3)   Use Layers and Groups as necessary.

4)   Where you need surfaces to blend with each other, merge the nodes at the joint between the surfaces you meshed, otherwise do not merge the nodes.

5)   Don't bother to specify any FE features (Rigid Links, Materials, Laminates, Lamination Angles, Properties, Load Cases, Displacement Constraints, Forces, Pressures or Nodal Pressures).

6)   Texture map and export the model as described in the 'How to Texture Map a Model' and 'How to Export a Model for Rendering' guides.

That's it. Not a lot [extra] to remember.

3.2.9   How to Texture Map a Model.

Creation of texture mapped models is a function used for rendering packages, and is not a core FE analysis package function.

Firstly, what *is* a texture mapped model?
A texture mapped model is one where a picture can be wrapped over the surface to make the model more realistic and add detail that cannot be added in the mesh without huge processor & modelling time implications.

The mechanics of texture mapping is comparatively simple. At each node on an element face an invisible 'Texture Node' is created. The co-ordinates of the texture node represent a 2D point on the texture that is to be wrapped over the model. The section of the texture enclosed by all the texture nodes on the element face represents the section of the texture stretched over that element face. Because the texture node is referenced in the element face, there is no necessity for adjacent element faces to have merged texture nodes – consequently adjacent elements may be mapped over different parts of the texture.

Texture mapping may also be used for transparency maps and 'bump' maps where surface displacements of the mesh are specified by a picture. Reflections, refractions, glossiness and specularity can also be subject to texture mapping. The effect of mapping can be seen in the model below. Both pictures have the same element colour, ambient light, specularity and glossiness. The only difference between them is that the left model has texture map related features turned off, while the right hand model has them on.

(both pictures rendered in DAZ| Studio)
As you can see, the level of realism that can be obtained with texture mapping is quite surprising.

There are lots of ways that models may be texture mapped. The figure shown above has five very complex texture maps, one for the body, one for the head, a third for all body hair (eyebrows, eyelashes etc), the fourth is for the teeth and the final map is for the eyes – then all the clothes have their own maps too. It is not possible to go to that level of sophistication in the Modelling Studio, but it is possible to get useful texture mapping for objects with a little thought, preparation and effort.

The best way to go through the process of texture mapping is by a simple example, that way all the features can be discussed as they are encountered.

In this example a model of a very basic garden table will be mapped. The model is as below. To create the model yourself, follow the instructions in Appendix 1.
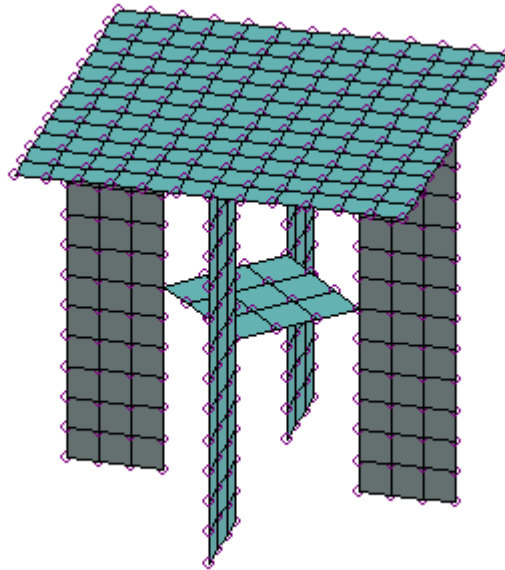
There are 4 legs, a top and a lower shelf.

To identify the different sections and allow the different parts of the table to have different basic colours, the entire table is to be added to the 'Table' element group, and the different parts onto different layers.

Creating the Group
There will only be one group, so it's possible to rename the element group that is created by default. All elements are added to the 'Current Group' as they are created, so the default group will be sufficient. Select menu option '**FE Model| Elements| Groups| Group Controls**'. Click on the default group name and right click over it, to get the 'Rename' pop-up menu item. Rename the group as 'Table'.
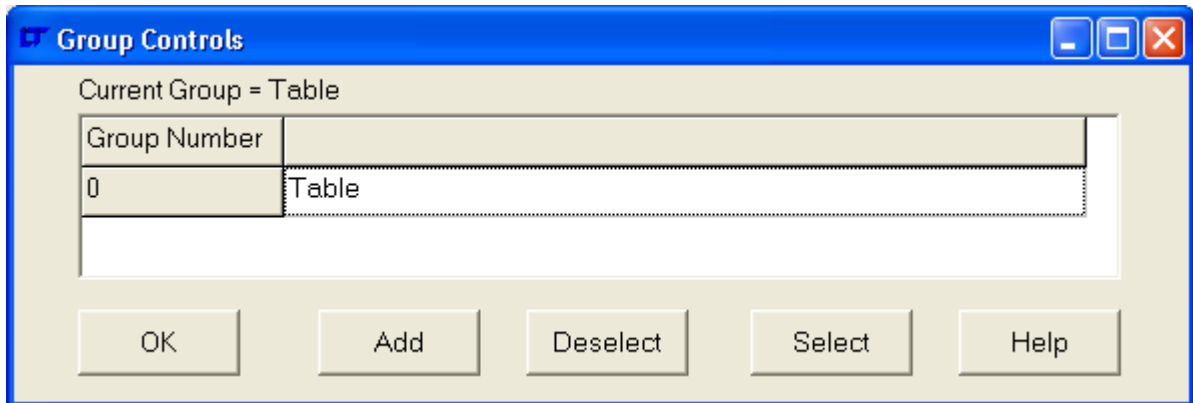
Creating the Layers
We want to create layers because each layer can have its own texture map, element colour, specularity and glossiness in the rendering package. If we did not create the layers, the rendering package would be forced to apply the same settings to all parts of the model – which may be a problem if you wished to have [for example] yellow legs for the table while having a green top.

Select Menu Option '**View| Layers| Layer Controls**' and add layers by pressing the 'Add Layer Button'. As each layer is created, click on the default name to rename it, or wait until afterwards, click on the layer to rename, and right-click to get the 'Rename' pop-up menu item. Once finished renaming the layers, double click on the 'Legs' layer to make it the current layer. The final window should look similar to the one below:-
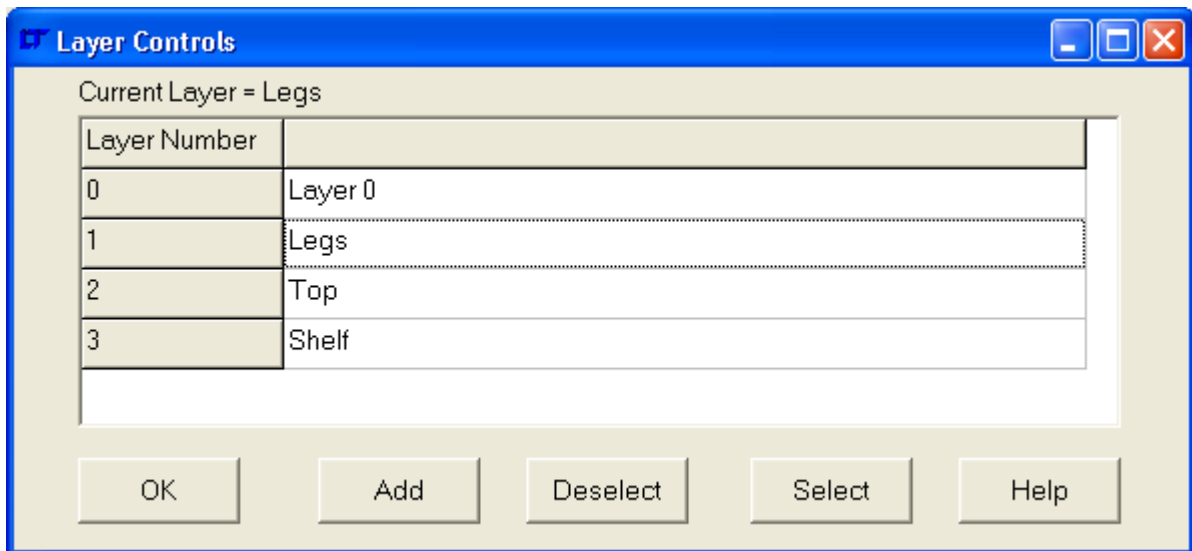
Now, select all the elements making the legs, as below:-
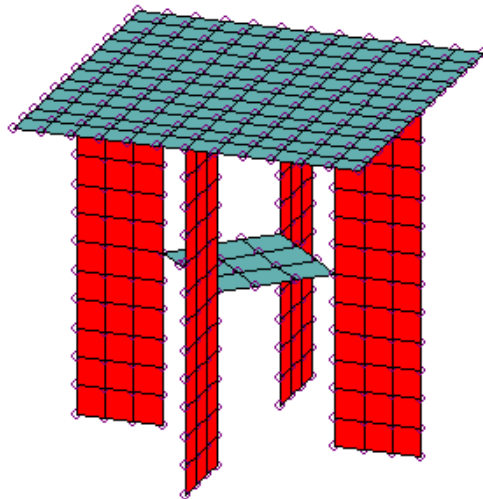
Select menu option '**View| Layers| Add to Current Layer**'. Now select '**View| Layers| Set Current Layer**' and set the shelf layer as the current layer. Add the shelf elements to the layer. Finally set the current layer to 'Top' and add the top elements. When you set the element colour to 'Colour by Layer', the following is visible:-
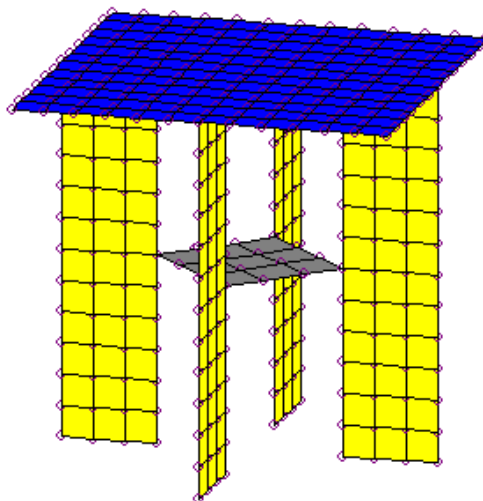
Now is the time to make a decision about how many texture maps there are going to be, and where each layer is going to be mapped. To see where the individual elements are mapped, use the '**Mapping| Templates| All Layers**'. If there is to be more than one texture map, it's possible to overlap where the layers are mapped, but it can get confusing when you plot a texture template for all layers – see the picture below for the texture template of the figure used at the start of this guide:-
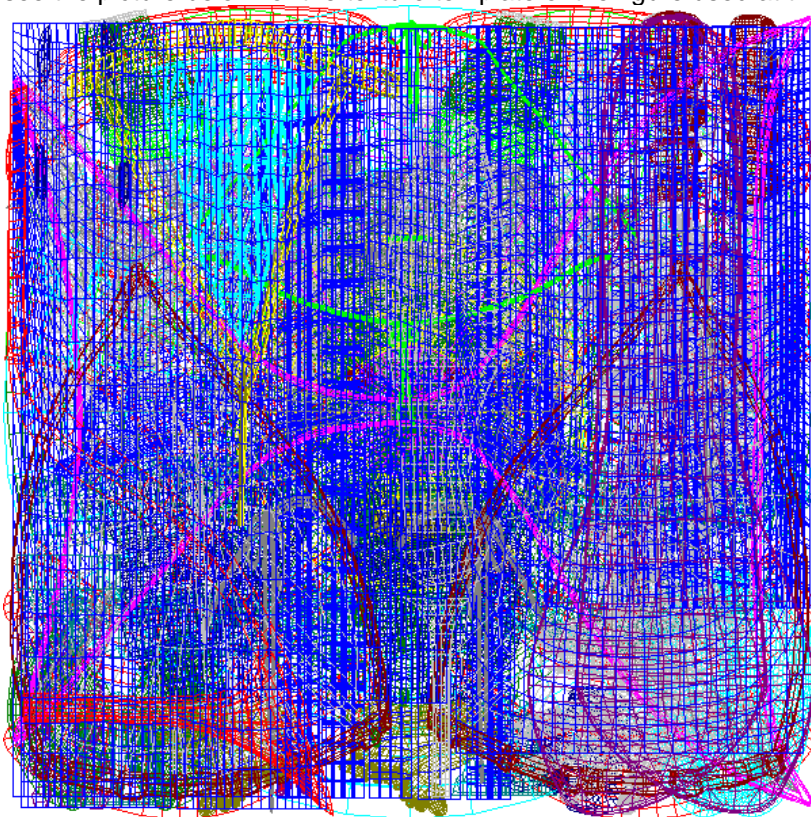
Under all that mess of hair, clothing etc. is the texture template for the 'SkinFace' (honest!):-
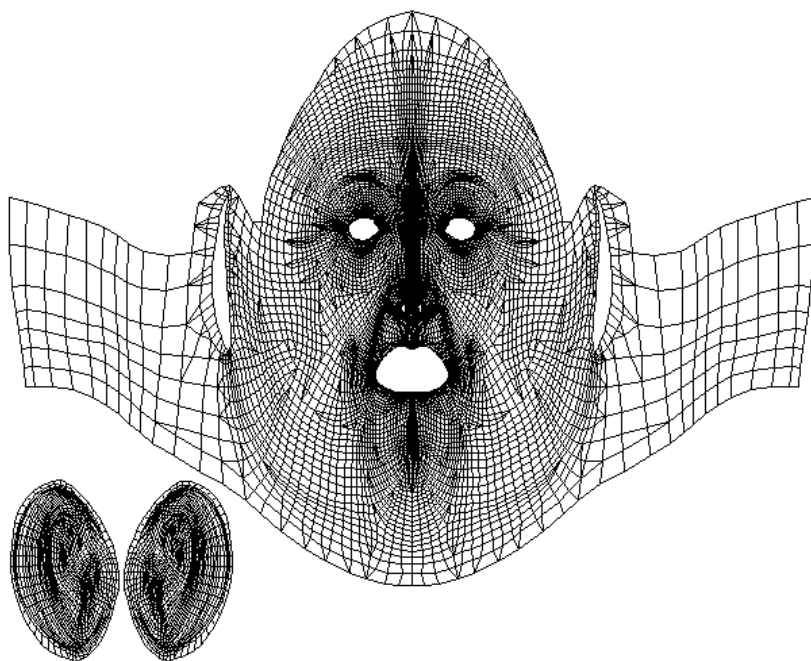
This table model could be mapped with one layer per texture map, however that is quite inefficient in terms of numbers of files for the model. It's easy to get multiple files lost or mislaid, so let's opt for 1 texture map for this model.

Note:-   One thing to remember is that all elements on a layer have to be mapped on to the same texture map, because the rendering package will only open one texture per layer. If this is a problem, just create a new layer for the elements that will not fit on the first texture map and move them to that layer.

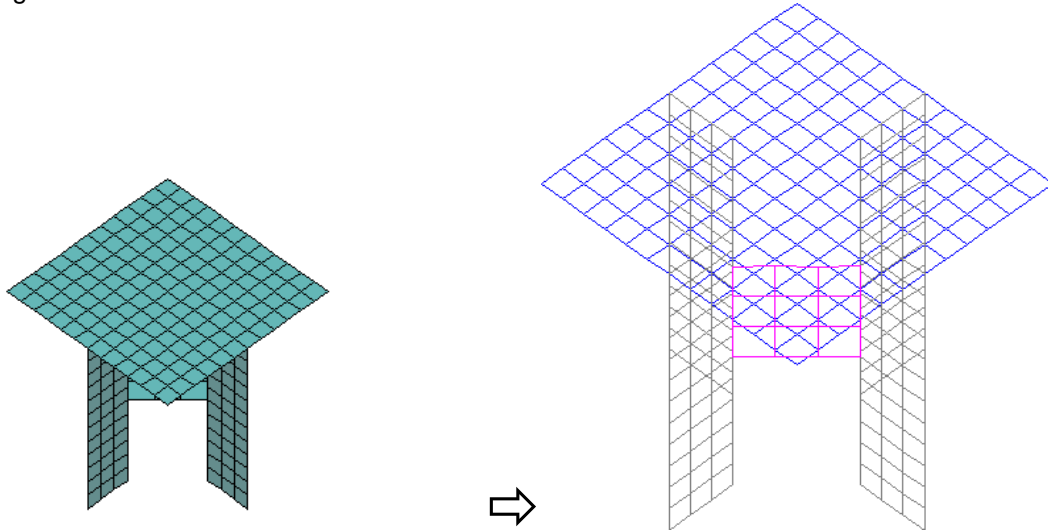The Modelling Studio creates texture mapping by scaling the elements to be mapped in the current view to fit the allocated area. If the elements are perpendicular to the view, they will be mapped onto a line – which is not very realistic. To give an example of mapping with a poor angle of view, look at this mapping below:-

In this poor mapping example, parts of the tabletop overlap the legs and shelf – this will require separate textures for each layer to keep the texturing distinct, and then a large area of each texture is blank, requiring a large texture just to get moderate detail. Additionally, the legs overlap each other – meaning that the texture applied to the top of one leg is also applied to the bottom of the other leg. This is very poor – what could you do if you wanted to make the two legs different colours in the texture? It's important to get the correct view before mapping and to plan where the map for each area is going to be.

Now to plan where each set of elements are to be mapped. The texture co-ordinates are *always* 0,0 at the bottom left of the picture and 1,1 at the top right of the picture, with X across the picture and Y up the picture. Let's adopt the template layout below:-

Table Top, X0.05,Y0.55 to X0.45,Y0.95	Shelf, X0.55,Y0.55 to X0.95,Y0.95
Left and Right Legs, from X0.05,Y0.05 to X0.45,Y0.45.	Fwd and Aft Legs, from X0.55,Y0.05 to X0.95,Y0.45
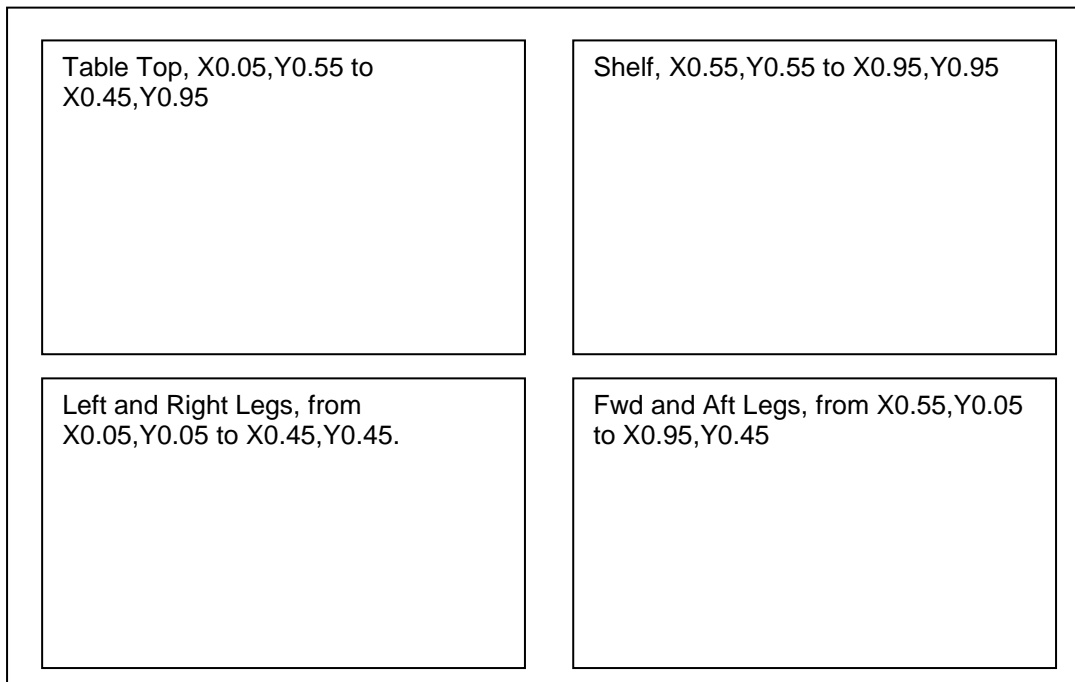
It will be noted that there are margins around each area, and that the scale will not be the same when mapped onto the texture. This really does not matter unless it's imperative to get the best detail possible on a map. If the margins cause problems, change the size of the notional area on the template that is going to be mapped onto. If more detail is required, increase the pixel dimensions of the texture.

Now to start mapping the top of the table.
Getting back to doing it right… We cannot map the whole table at once, because the elements are orthogonal to each other and whatever view we take, it's impossible to get a perpendicular view of all faces. If we have an angled view we loose detail in the texture map by smearing the map over a larger area. So, to map the top of the table, select menu item '**View| Angle| Top**', and select the table top elements by their layer name – menu item '**Selection| Layers| Select by Layer Name**', and pick the 'Top' layer from the selection dialogue box that opens. Once finished, the model will look as below:-
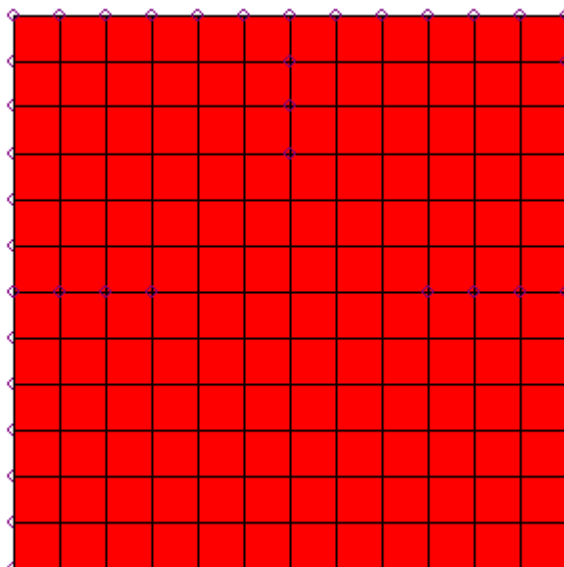
Do the mapping of the top by selecting menu item '**Mapping| Add| Selected**'. In the mapping dialogue window that opens, type the bottom left as 0.05, 0.55 and the top right at 0.45,0.95, as below:-

Texture Map Area

Bottom Left of Texture

X   0.05

Y   0.55

Top Right of Texture

X   0.45

Y   0.95

OK   Cancel   Help
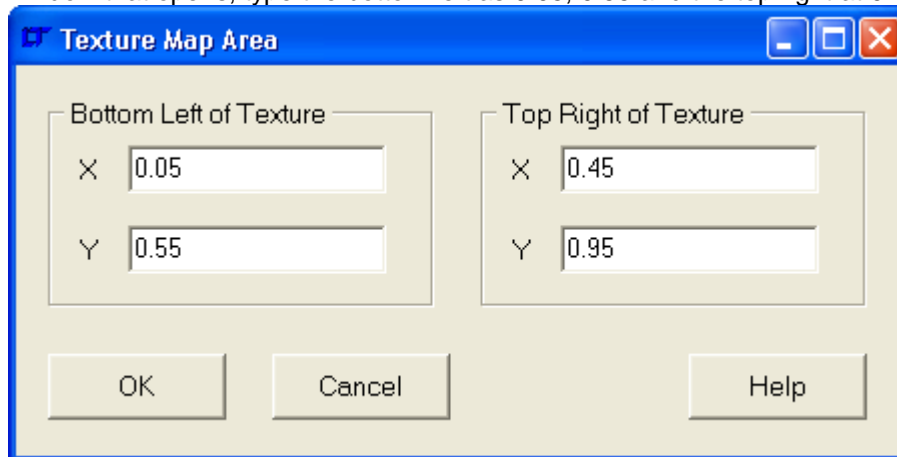
This gives the following texture template to show what we've done [menu item '**Mapping| Templates| All Layers**']:-
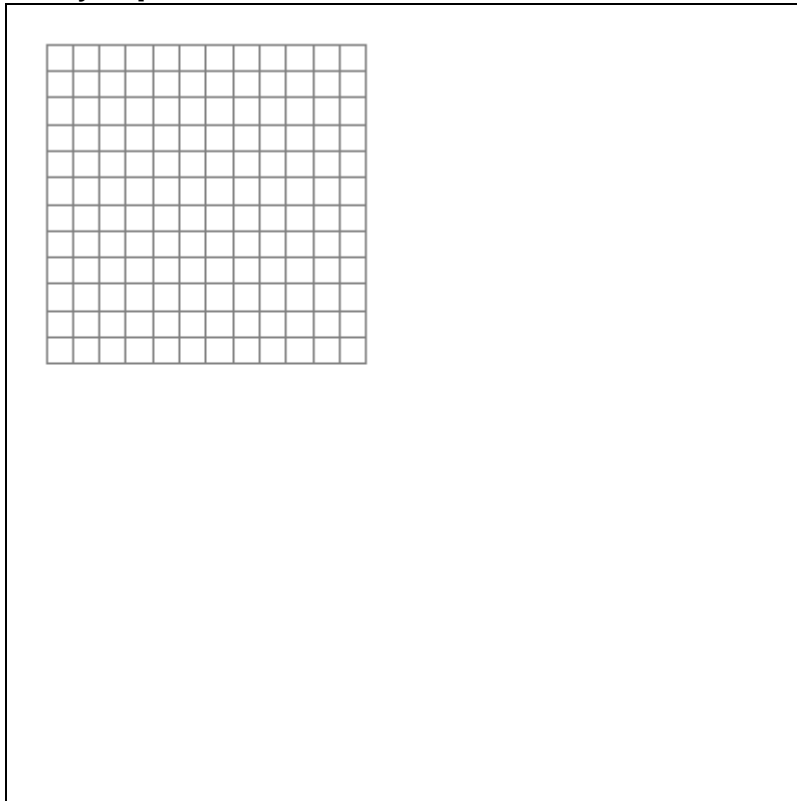
Repeat with the shelf, only giving the new co-ordinates for it, getting the following template:-
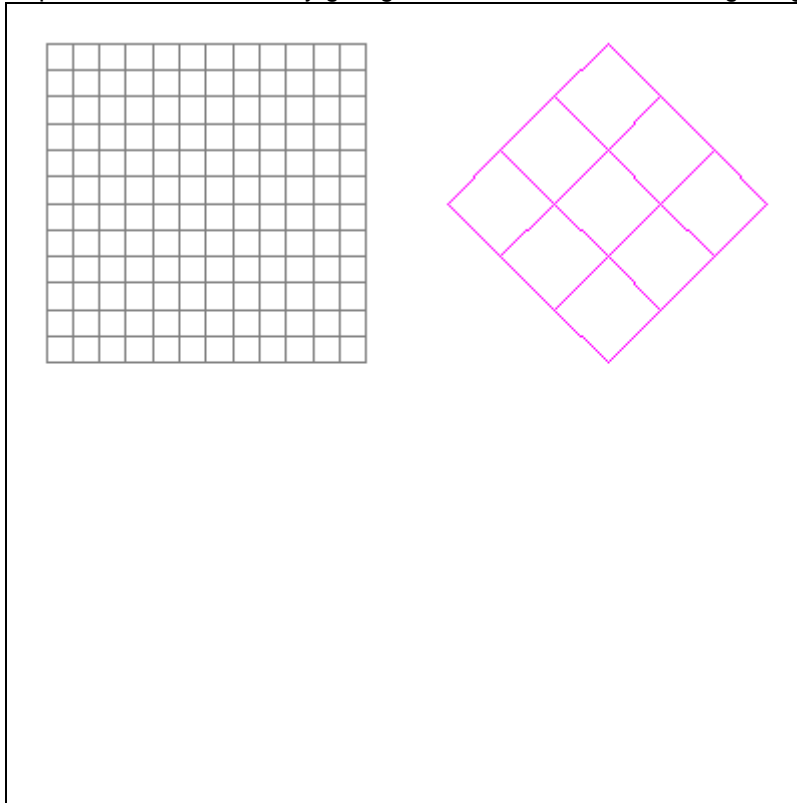
The legs cannot be mapped in one go. Begin with the fore/aft legs. Select the front view. Select the fore/aft leg elements, select menu item '**Mapping| Add| Selected**'. In the mapping dialogue window that opens, type the bottom left as 0.55, 0.05 and the top right at 0.95,0.45.
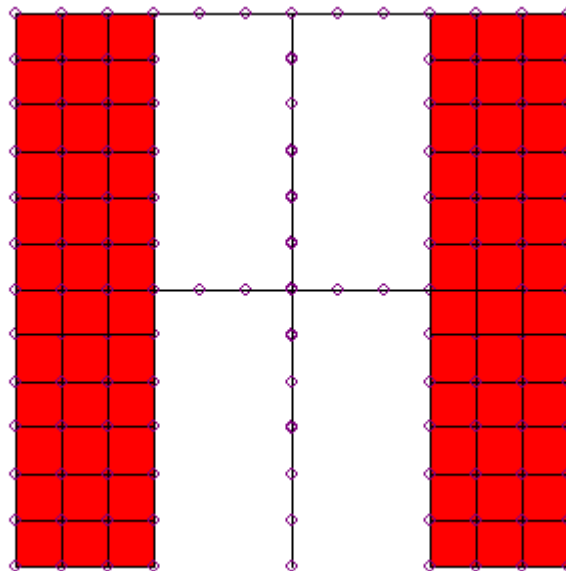
Do the same with the left/right legs after choosing the right view. The final texture template is:-
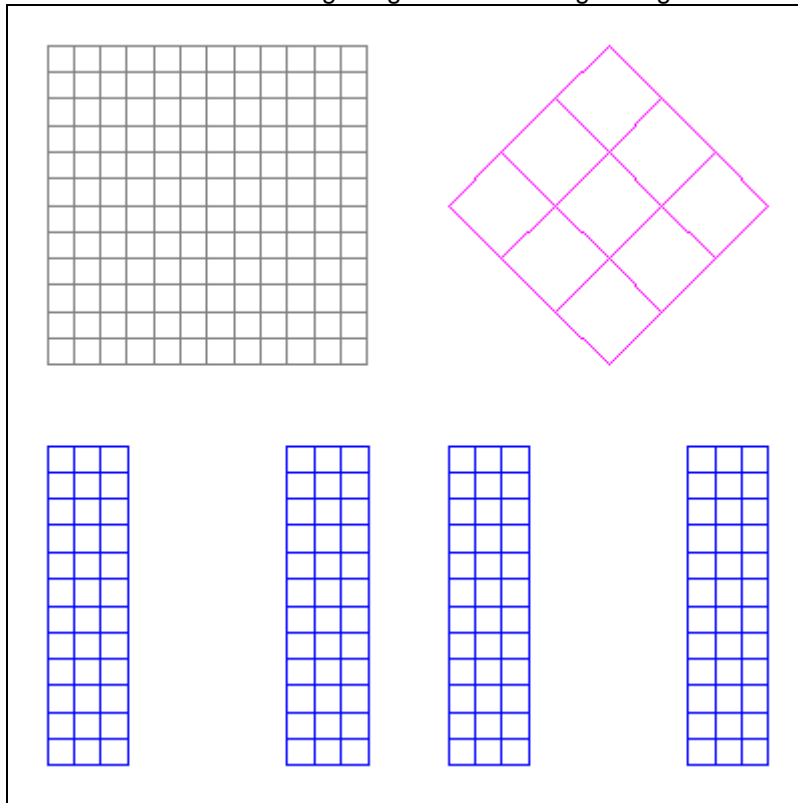
Now to export the model file, save the template and use it to create a transparency map in an image editing program (like Paint Shop Pro or Photoshop). Finally, take the model into a rendering package and do something with it:-

Transparency map (black = transparent, white = opaque, colours = semi-transparent):-
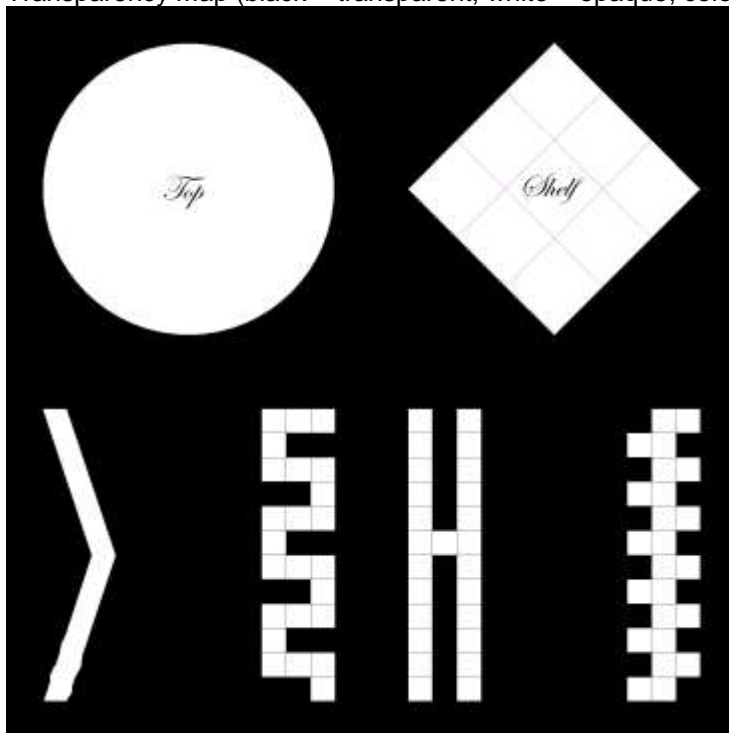
Render:-

And that's virtually all there is to it. Just remember, it really can work out in the end!

3.2.10 How to Export a Model for Rendering

On the face of it, the way to export a file for rendering is to choose menu option '**File| Export**', and choose the Wavefront Object file export type. This will do the job, but the results may be unsatisfactory to one degree or another.

Causes of Unsatisfactory Results
There are a number of causes of unsatisfactory results, each of which has a separate cure.

1)   Symptom: Transparent near faces of Model in rendering package. This can be really peculiar, when parts of the model disappear while other parts appear as the view is rotated. However, when the model is rendered, everything is OK.
Cause: Element normals are inverted compared to the angle of view and the rendering package is using 'Back Face Culling' to speed the display.
Solution: Make sure that all the element normals point out from the centre of the model when in the Modelling Studio. It may be necessary to select the offending elements and choose menu option '**FE Model| Elements| Modify| Reverse Normal**'. If the model is not a closed model, it is inevitable that there will be some of this effect visible somewhere.

2)   Symptom: Edges not crisp when rendering, with adjacent faces being blended.
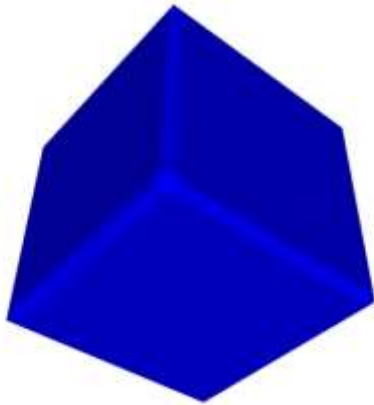
Cause: Most rendering packages smooth the model as it is rendered. Sharp corners are blended in an attempt to create a smooth looking surface where nodes are shared between faces. Sometimes this isn't wanted.
Solution: The faces that have edge nodes merged must be separated. Select the elements on the entire face that is to be separated, and choose menu option '**FE Model| Elements| Modify| Separate**'. It may be necessary to do this more than once. This will then look as below:-
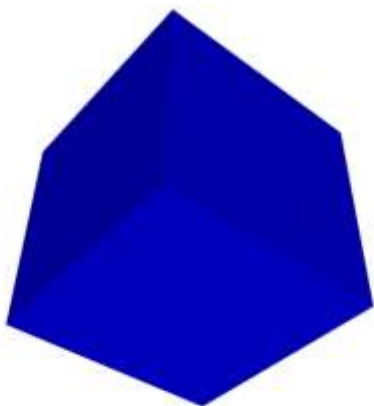
3)   <u>Symptom</u>: The model appears to have rotated 90° forward on export.
        <u>Cause</u>: Most rendering packages use the Y axis as the vertical axis, rather then the Z axis.
        <u>Solution</u>: To resolve the problem, rotate the model by -90° about the X axis prior to exporting it.

4)   <u>Symptom</u>: The model size/ scale is unexpected.
        <u>Cause</u>: The rendering package being used has a default scaling factor on import.
        <u>Solution</u>: Find the scale factor assumed by the package and scale the model in the Modelling Studio to match the assumed dimensions.
        For example:-
        Take an 1850 (mm) high figure in the Modelling Studio, which is to be exported to Poser 4, where 1 Poser Unit = 96”, then the figure would have to be scaled by a factor of:-
                Factor = $1/(96 \times 25.4) = 4.101 \times 10^{-3}$,
        This gives a figure height of 0.75869 Poser Units, which is correct.

5)   <u>Symptom</u>: The model in the rendering package is composed of triangles. The Modelling Studio modelled rectangles.
        <u>Cause</u>: The Modelling Studio exports parabolic formulation faces as groups of triangles.
        <u>Solution</u>: Convert all the elements from parabolic formulation to linear formulation by selecting them and choosing menu option '**FE Model| Elements| Modify| Parabolic to Linear**'. It will then be useful to select all the nodes and choose menu option '**FE Model| Nodes| Delete Inactive**' to get rid of the nodes that are no-longer needed.

3.2.11 How to Create a Morph Target

Creation of morph targets is a function used for rendering packages, and is not a core FE analysis package function.

<u>Firstly, what *is* a morph target?</u>
A morph target is a model with a distorted shape relative to an original. This way a rendering package can load the original shape, the morph target, and then dial in a percentage morph to distort the original to the new shape, or any combination of the two. This is shown with the model below of a simple dinner plate:-

Original Model
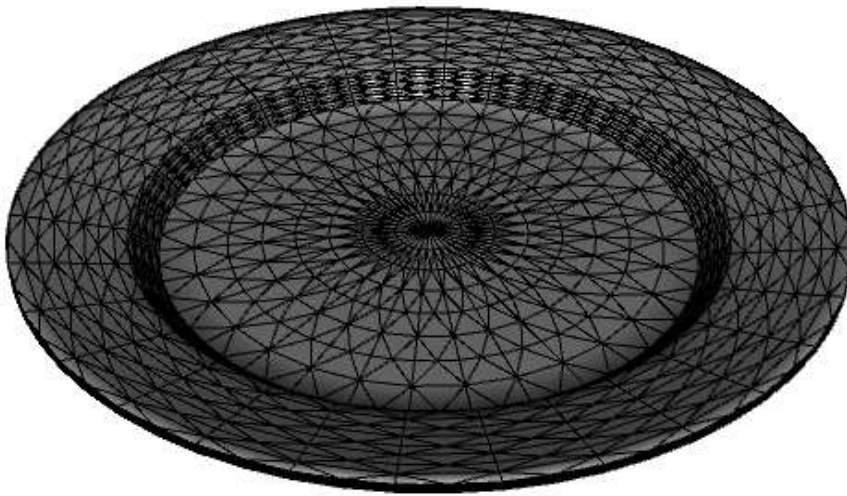
Morph Target Model (100% morph strength):-
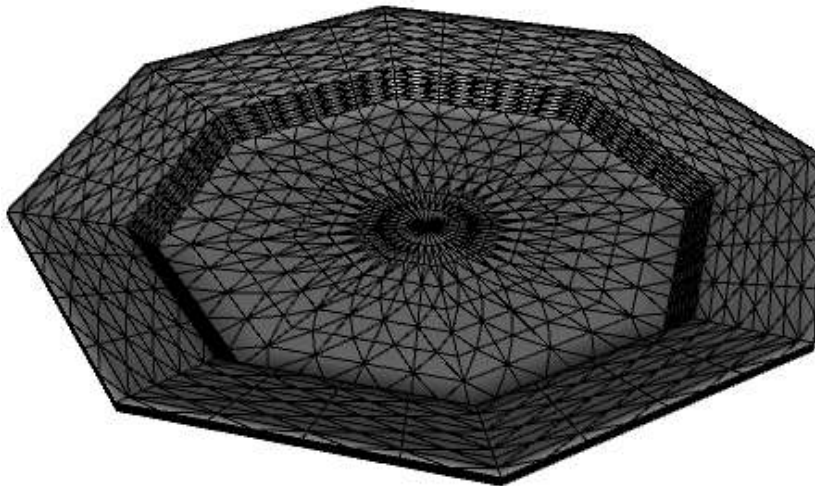
Unexpected, and frequently quite attractive, effects can occur when different morph strengths are used, as below:-

200% strength:-
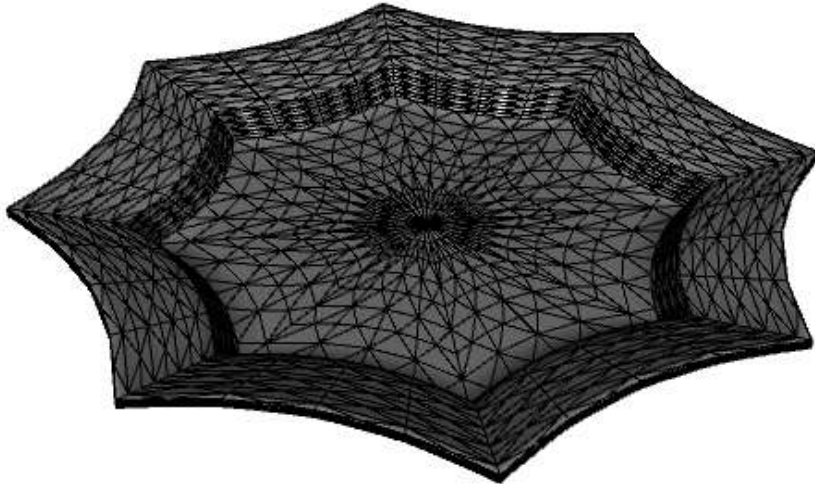
-200% morph strength
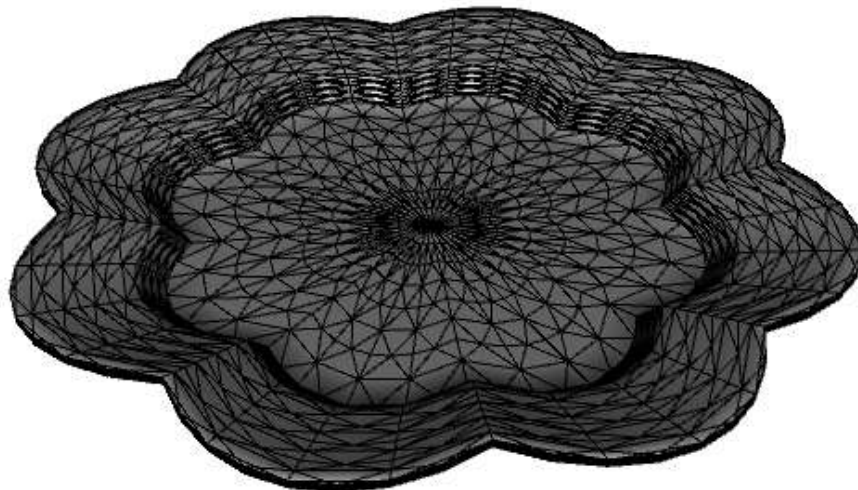
(All pictures taken as screen shots from DAZ| Studio using a plate modelled in the Modelling Studio, with each surface meshed with a 4 x 4, 8 noded, shell element mesh – triangular elements created on the fly by the export routine, see notes to section 4.6).

How to create a morph target.

Most rendering packages will create morph targets from Wavefront Object files. So, create the original shape model, and texture map it as necessary. When you are happy that there are no more changes to be made to it, export it as a Wavefront Object file.

Now distort the mesh to the new shape. Whatever you do, DO NOT renumber, add or delete any of the nodes or elements. Only move the nodes or elements using menu commands '**FE Model| Nodes| Move**' or '**FE Model| Elements| Move**' as required. Geometry (such as Points, Lines and Surfaces) can be added and used to help the distortion process, as they are not exported in the Wavefront Object export routine.

When satisfied with the distorted shape, export it as a Wavefront Object file with a new file name.

Go to the rendering package, import the original file, load the distorted shape file as a morph target, and dial in the morph value you want.

3.2.12 Understanding Undo's

The Modelling Studio keeps a list of undo events in memory, using the commands found in the 'Command Line Reference' section of the 'Command Reference'. Each different undo event is prefaced by an undo title, which appears in the 'Undo' tool button hint. The undo list for adding a line with the 'Add Line' Modeller Tool Button would be as below:-

```
Title: Add Line 1
Point,Delete,1
Point,Delete,2
Line,Delete,1
```

Then if Line 2 were to be added, connecting Point 2 to a new Point 3, the undo list would become:-

```
Title: Add Line 1
Point,Delete,1
Point,Delete,2
Line,Delete,1
Title: Add Line 2
Point,Delete,3
Line,Delete,2
```

The undo list now has two undo events (consisting of five command line commands).

When the 'Undo' button is pressed, the command line at the end of the undo list is acted upon, the line is deleted and so on until the undo title line is reached. The undo title is deleted and the undo event stops.

When any entities are selected, the selection has to be undone, and the entity erased again if it was initially invisible, so the Undo list entry for a 5000 element model could be:-

```
Title: Select All Elements
Element,Erase,1
Element,Unselect,1
Element,Erase,2
        .
        .
Element,Unselect,4999
Element,Erase,5000
Element,Unselect,5000
```

As can be seen, it's possible to build up a very large undo list very rapidly from seemingly simple commands. A large undo list can take up a large amount of memory; to prevent this, the Modelling Studio has a number of strategies:-
1) The undo list is cleared every time there is a successful saving of the .mdf file.
2) The oldest undo commands are deleted once a set number of undo commands has been reached. This 'Undo History Length' can be altered by the user. By default the length of the undo history is 20 undo commands, but can be increased or decreased using '**Edit| Undo History Length**'. The 'Undo History Length' is persistent, and will carry through to the next time the Modelling Studio is used.
3) When the mouse hovers over menu option '**Edit| Undo History Length**', the Status Bar hint shows how long the Undo History limit is, how many Undo events there are, and how many command line commands there are – which can help to determine whether it's necessary to alter the 'Undo History Length'.

With this overview of how the 'Undo' system works, you can manage the system's memory usage more effectively.

3.2.13 How to Create Elements with 'Pick Nodes'

Creating elements with the 'Pick Nodes' option to is not an efficient way to create elements. It will take an excessive length of time; however, it allows maximum flexibility in the element creation process. You are urged **not** to create elements this way, however as it's possible, it had better be described fully.

Firstly, select menu option '**FE Model| Elements| Add| Settings**' to specify the elements that will be created when picking nodes. The following window will open:-
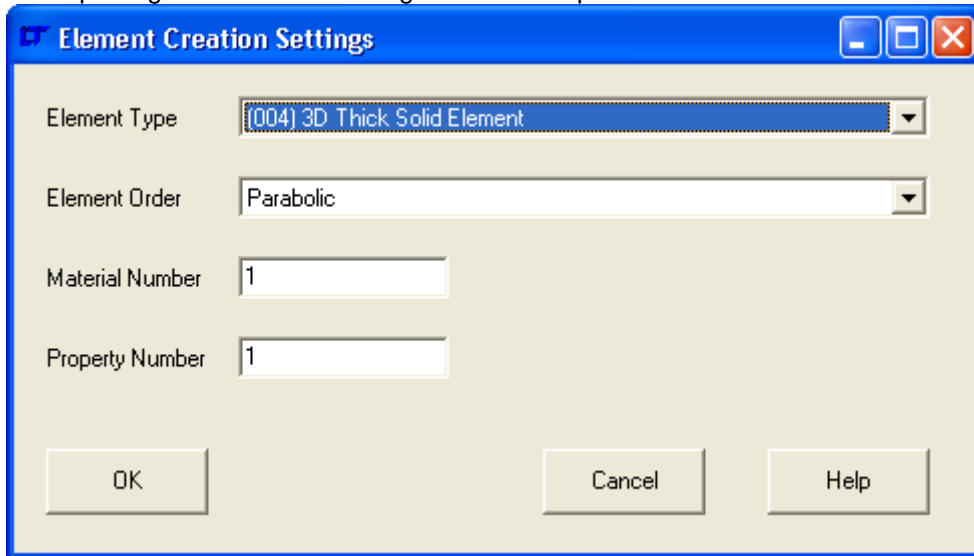
Choose the correct element type.

Now select menu option '**FE Model| Elements| Add| Pick Nodes**'. Now pick the nodes in the order shown in the 'Element Library' section on page 73. 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 all the nodes, 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 – the sequences are given below:-

Examples:-
To create a point element, pick one node:-

■1

To create a line element, pick two nodes:-
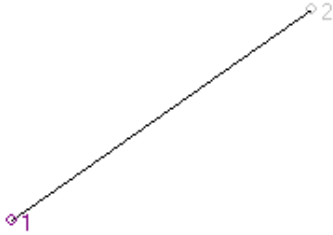
To create a linear rectangular shell element, pick the four corner nodes, as below:-
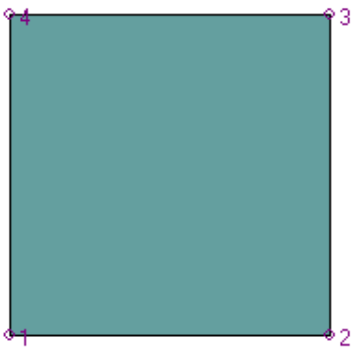
The node pick sequence would be 1,2,3,4. The face visible is the TOP face.

To create a parabolic rectangular shell element, pick the 8 nodes below:-
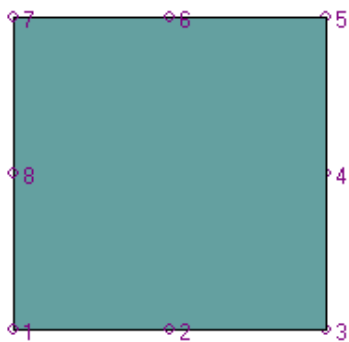
The node pick sequence would be 1,2,3,4,5,6,7,8. The face visible is the TOP face.

To create a linear triangular element, pick on the first node after picking the final node.
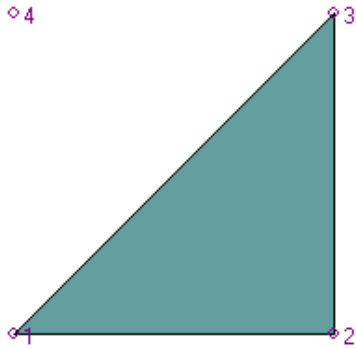
The node pick sequence would be 1,2,3,1. The face visible is the TOP face.

To create a parabolic triangular element, pick on the first node after picking the final node.
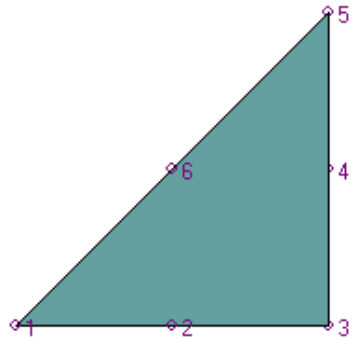
The node pick sequence would be 1,2,3,4,5,6,1. The face visible is the TOP face.

To create a linear rectangular Brick Element, pick the nodes in the sequence shown below:-
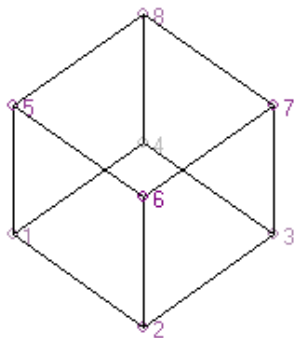
The node pick sequence is given by the node numbers.

To create a parabolic rectangular Brick Element, pick the nodes in the sequence shown below:-
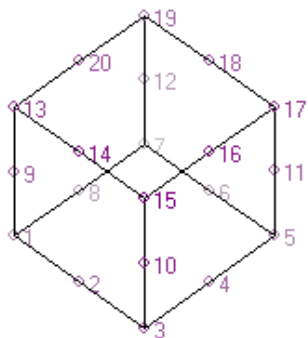
The node pick sequence is given by the node numbers.

To create a linear 4 noded pyramid element, pick the nodes in the sequence shown.
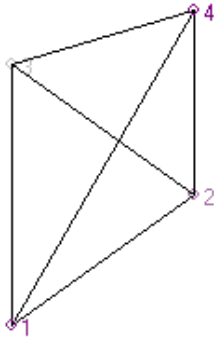
The node pick sequence is 1,2,3,4,1.
If you do not repeat node 1 at the fifth pick, the Modelling Studio assumes you want to pick an 8 noded brick element.

To create a parabolic 10 noded pyramid element, pick the nodes in the sequence shown.
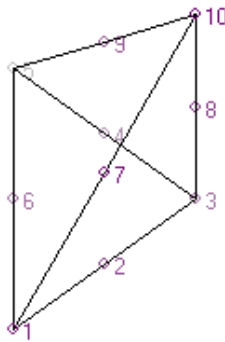
The node pick sequence is 1,2,3,4,5,6,7,8,9,10,1.
If you do not repeat node 1 at the eleventh pick, the Modelling Studio assumes you want to pick a 20 noded brick element.

3.2.14 How to Apply a Moment

It is not possible to apply a moment load directly to a node from within the Modelling Studio, though the Modelling Studio enables you to import FE Analyses with moment loads without losing the moments on export.

The way to know how to apply a moment load is answered by the definition of a moment load.

A moment load is the twisting effect due to applying a load at an offset distance.

So, to apply a moment load at a node, create a Rigid Link with one slave node. The slave node is the node with the moment load applied to it. Apply an offset load to the master node of the rigid link such that:-

Load x Rigid Link Length = Moment

This is shown below:-
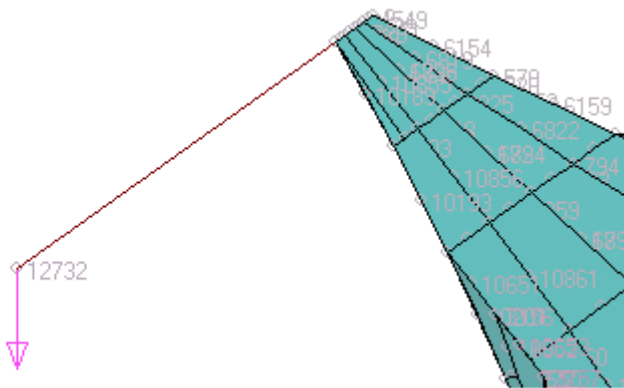
If the moment load that must be applied is intended to be a pure moment (and those don't happen very often), make the Rigid Link longer and the applied load less. Alternatively, make a small element at the load application point, and put in a second rigid link with an equal and opposite force, as below:-
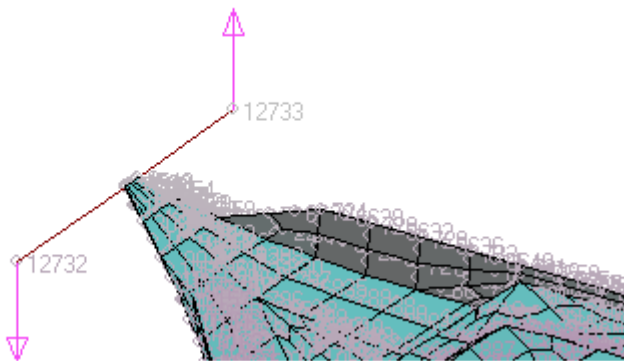
Remember, a Node can't be the slave of more than one Rigid Link.

4.   File Reference

4.1   General Information

4.1.1   Section 4 is the reference section for the import/ export of various file types and gives file information relating to file formats designed for the Modelling Studio only. The remainder of section 4.1 specifies the files that are installed when the program is first added to the PC.

4.1.2   When the Modelling Studio is installed the following files are created:-

File	Location	Description
NISATools.exe	<Program Directory>	The Modelling Studio executable file.
NISATools.ini	<Program Directory>	Initialisation file holding persistent data
Command Reference.pdf	<Program Directory>	Modelling Studio Command Reference
Quick Start Guide.pdf	<Program Directory>	Modelling Studio Quick Start Guide
User Guide.pdf	<Program Directory>	Modelling Studio User Guide
NISATools.hlp	<Program Directory>	Help file: NOT for Modelling Studio
NISATools.gid	<Program Directory>	Help file index.
NISATools.fts	<Program Directory>	Help file full text search.
History Release 1_4.rtf	<Program Directory>	Details of final release.
History.rtf	<Program Directory>	History of program
NISA Tools Message1.rtf	<Program Directory>	Ensures that all e-mail message files
NISA Tools Message10.rtf	<Program Directory>	sent from within the Modelling Studio
NISA Tools Message2.rtf	<Program Directory>	are deleted on un-installation.
NISA Tools Message3.rtf	<Program Directory>	
NISA Tools Message4.rtf	<Program Directory>	
NISA Tools Message5.rtf	<Program Directory>	
NISA Tools Message6.rtf	<Program Directory>	
NISA Tools Message7.rtf	<Program Directory>	
NISA Tools Message8.rtf	<Program Directory>	
NISA Tools Message9.rtf	<Program Directory>	
Error.rtf	<Program Directory>	Default file containing text specifying error.
DXFError.rtf	<Program Directory>	File containing text specifying DXF file error
OBJError.rtf	<Program Directory>	File containing text specifying Obj file error
Sample.dbs	<Program Directory>	Sample DISPLAY III V9 dbs file
Sample.nis	<Program Directory>	Sample NISA file.
Sample.out	<Program Directory>	Sample NISA output file
Case01.dat	<Program Directory>	Bolt Group Analysis Verification File
Case02.dat	<Program Directory>	Bolt Group Analysis Verification File
Case03.dat	<Program Directory>	Bolt Group Analysis Verification File
Case04.dat	<Program Directory>	Bolt Group Analysis Verification File
Case05.dat	<Program Directory>	Bolt Group Analysis Verification File
Case06.dat	<Program Directory>	Bolt Group Analysis Verification File
Case07.dat	<Program Directory>	Bolt Group Analysis Verification File
Case08.dat	<Program Directory>	Bolt Group Analysis Verification File
Case09.dat	<Program Directory>	Bolt Group Analysis Verification File
Case10.dat	<Program Directory>	Bolt Group Analysis Verification File
Case11.dat	<Program Directory>	Bolt Group Analysis Verification File
Case12.dat	<Program Directory>	Bolt Group Analysis Verification File
Case13.dat	<Program Directory>	Bolt Group Analysis Verification File
Steel BS EN 10025 S275 N & mm Density per m cubed.mmf		
	<Program Directory>	Sample Modeller Material File
Aluminium BS EN 485 AW 6082-T6 N & mm Density per m cubed.mmf		
	<Program Directory>	Sample Modeller Material File
50 x 25 x 3 RHS.mpf	<Program Directory>	Sample Modeller Property File
Panel 0.4inch thick.mlf	<Program Directory>	Sample Modeller Laminate File

File	Location	Description
About Blurb.txt	<Common Files>	Text scrolled in 'About Box'.
EPS Logo.avi	<Common Files>	Animation in 'About Box'
Dictionary.dic	<Common Files>	Dictionary for spell checking
Custom.dic	<Common Files>	Custom Dictionary
Speller.hlp	<Common Files>	Spell check help file
Speller.fts	<Common Files>	Spell check help full text search
Speller.gid	<Common Files>	Spell check help index

The <Program Directory> is the installation directory for the Modelling Studio.
The <Common Files> directory is the 'EPS Logistics Technology Ltd' subdirectory of the system common files directory, usually 'C:\Program Files\Common Files\EPS Logistics Technology Ltd\'.

The Modelling Studio is being offered as Open Source software, so that the program can be checked by the users. Please note that all the code is copyright and cannot be used in other applications without permission. The source code for the Modelling Studio main window (but not its child windows) is given in the following files:-

File	Location	Description
Structures.h	<Program Directory>	Structures used by the Modelling Studio.
base.h	<Program Directory>	Word processor ancestor
base.dfm	<Program Directory>	of the Modelling Studio.
base.cpp	<Program Directory>	
StringHelp.h	<Program Directory>	Text functions.
StringHelp.cpp	<Program Directory>	
modeller.h	<Program Directory>	The Modelling Studio Class.
modeller.dfm	<Program Directory>	
modeller.cpp	<Program Directory>	
modeller2.cpp	<Program Directory>	Overflow code from Modeller.cpp after it got too big to link.

4.1.3   When the Modelling Studio tool is used a session file is automatically created with a new name every time the Modelling Studio is started. The name includes a three digit number to avoid over-writing a previous one. It will be worth clearing out unwanted session files regularly. The session files will all need to be deleted manually before the Modelling Studio can be fully un-installed, which can be done from within the Modelling Studio using menu option '**Edit| Delete Session Files**'.

File	Location	Description
Modeller_Session_File_***.msf	<Program Directory>	Modeller Session file.

4.1.4   No registry settings are made apart from those required by the InstallShield wizard to un-install the program.

4.2   Modelling Studio Formats

4.2.1   Modeller Data File (.mdf)

The Modeller Data File is a binary file format which is both extremely long and complex. It has the facility to add extra data as time goes on without disturbing existing data. If the file information is needed or wanted, contact richard.williams@epslt.co.uk for the latest version information. You will probably only get source code as your answer; but as it'll be the code used to create the .mdf file, you can't get anything more authoritative!

4.2.2   Modeller Session File (.msf)

The modeller session file is a plain text file which consists of all the commands to follow in the order in which they must be carried out. There must be only one command per line and one line per command. Full details of every possible command are given in section 2 of the 'Command Reference'. The file name of the session file is specified on the third line of the '**Progress Information**' box. The session file is located in the same directory as the NISA Tools program. It will be worth clearing out unwanted session files regularly.

4.2.3   Modeller Laminate File (.mlf)

The Modeller Laminate File is a plain text file based on a Windows .ini file format. The required entries are as below:-

```
[LaminateData]
LaminateName=Laminate Name Here
Laminate Orientation=m
NumberofLayers=n
Property0=Property Layer 0
LaminateAngle0=LaminateAngle Layer 0
Material0=Material Layer 0
Property1=Property Layer 1
LaminateAngle1=LaminateAngle Layer 1
Material1=Material Layer 1
..
..
Property[n-1]=Property Layer n-1
LaminateAngle[n-1]=LaminateAngle Layer n-1
Material[n-1]=Material Layer n-1
```

The [LaminateData] section is imperative.

The Laminate Orientation, m, defines the 0° fibre orientation for the laminate, and must be one of the following:-

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.

The materials, properties and laminate angles are all zero referenced, so the layer counting starts at zero and increments to [n-1] where n is the number of layers.

The order of the layers, property, material and laminate angle data is unimportant provided it is all in a contiguous section after the [LaminateData] heading.

Each material, fibre angle and property referenced in this laminate is also saved in the laminate file with the same data as required in sections 4.2.4 (Modeller Material File), 4.2.5 (Modeller Property File) and 4.2.6 (Modeller Fibreangle File). The only difference is that the data is headed by a [MaterialData*n*], [PropertyData*n*] or [LamAngleData*n*] section title.

The Laminate Angle data is in the same format as the property data, headed by a [LaminateAngle*n*] section title.

One entry per line and one line per entry.

There is a system limit of 64k bytes to the size of the laminate file. If this is exceeded, then you will not be able to recover all the data saved. The likelihood of exceeding this capacity is very low unless you have an extremely complex laminate with multitudes of materials and numerous layers. A typical 3 layer/ 2 material/ 2 property/ 1 angle laminate takes just over 2k bytes.

4.2.4   Modeller Material File (.mmf)

The Modeller Material File is a plain text file based on a Windows .ini file format. The required entries are as below:-

```
[MaterialData]
MaterialName=Material Name
EX=Value
EY=Value
EZ=Value
NUXY=Value
NUXZ=Value
NUYZ=Value
GXY=Value
GXZ=Value
GYZ=Value
DENS=Value
FXT=Value
FYT=Value
FZT=Value
FXC=Value
FYC=Value
FZC=Value
FSXY=Value
FSXZ=Value
FSYZ=Value
F12=Value
F13=Value
F23=Value
FS=Value
FTS=Value
ALPX=Value
ALPY=Value
ALPZ=Value
KXX=Value
KYY=Value
KZZ=Value
C=Value
Isotropic= 1 or 0
```

The order of the material data is unimportant provided it is all in a contiguous section after the [MaterialData] heading.

Isotropic=1 if the material is Isotropic, else Isotropic=0

All entries must be entered. One entry per line and one line per entry.

4.2.5   Modeller Property File (.mpf)

The Modeller Property File is a plain text file based on a Windows .ini file format. The required entries are as below:-

```
[PropertyData]
PropertyName=Property Name
Property Type=m
Data1=Value
Data2=Value
..
..
Data[n]=Value
Number of Entries=n
```

For the 'Property Type', the property type value m 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 **NISA Only**
9	=	3D Mass/Inertia **NISA Only**
10	=	2D General Mass **NISA Only**
11	=	3D General Mass **NISA Only**
12	=	2D Gap
13	=	3D Gap
14	=	2D Spar
15	=	3D Spar/Strut
16	=	**For NISA Compatibility only ** 3D Cable
17	=	**For NISA Compatibility only ** 3D Damper
18	=	**For NISA Compatibility only ** 3D Elbow
19	=	**For NISA Compatibility only ** 3D Pipe
20	=	**For NISA Compatibility only ** 2D Gap/Friction
21	=	**For NISA Compatibility only ** 3D Gap/Friction
22	=	**For NISA Compatibility only ** 3D Thermal Bar
23	=	**For NISA Compatibility only ** 3D Convection Link
24	=	**For NISA Compatibility only ** 3D Radiation Link

The Data entry is 1 referenced, so when there are 24 data entries the data goes from 'Data1' to 'Data24'.

The order of the data is unimportant provided it is all in a contiguous section after the [PropertyData] heading.

All entries must be entered. One entry per line and one line per entry.

4.2.6   Modeller Fibre-Angle File (.mff)

The Modeller Fibre-Angle File is a plain text file based on a Windows .ini file format. The required entries are as below:-

```
[LamAngleData]
LamAngleName=Angle Set Name
NumberofNodes=n
data0=Value
data1=Value
..
..
Data[n-1]=Value
```

The Data entry is zero referenced, so when there are 8 data entries the data goes from 'Data0' to 'Data7'.

The order of the data is unimportant provided it is all in a contiguous section after the [LamAngleData] heading.

All entries must be entered. One entry per line and one line per entry.

4.3   EMRC Formats

4.3.1   NISA File (.nis)

NISA files may be opened and exported by the Modelling Studio. The following data cards are supported:-

Executive Commands	ANALYSIS=STATIC or EIGEN
	AUTO CONSTRAINT
	BLANK COMMON
	EIGEN EXTRACTION
	ELEMENT ECHO
	EXECUTION
	FILE
	GEOM PROPERTIES
	MASS FORMULATION
	NODE ECHO
	PATH
	RESEQUENCE
	SAVEFILE
	SOLV
	SORT STRESS
	WARNING

```
*TITLE
*ELTYPE
*RCTABLE
*LAMANGLE
*LAMSEQ
*NODES
*ELEMENTS
*RIGLINK
*MATERIAL
*LDCASE
*LCTITLE
*SPDISP
*CFORCE
*PRESSURE
*BODYFORCE
*PRINTCNTL                        Export only
        ELFORCES    ,0            Export only. Fixed output.
        ELSENERGY   ,0            Export only. Fixed output.
        ELSTRESSES  ,0            Export only. Fixed output.
        LDVECTOR    ,0            Export only. Fixed output.
        REACTIONS   ,0            Export only. Fixed output.
        RLFORCES    ,0            Export only. Fixed output.
        SLFORCES    ,0            Export only. Fixed output.
*ENDDATA
```

Notes

1)   With Nodes it is imperative that the Global Cartesian co-ordinate system is used for the node position definition prior to import.

2)   The Local Displacement Co-ordinate System on each node is retained on reading a .nis file. The LCS remains hidden in the modeller and only re-appears on exporting the amended .nis file. It must be noted though, that user defined *LCSYSTEM definitions are lost on importing the NISA file, and must be copied into the exported .nis file if the full *LCSYSTEM definitions are to be retained.

3)   A new Modelling Studio window is opened for every NISA file opened. It is not possible to 'Import' a NISA file into an existing model; this has to be done by merging the Modeller Data Files with '**File| Merge**'.

4)   NISA does not support Nodal Pressures as separate entities, so on export the NISA file has the Nodal Pressures and Forces added together to form a single nodal load.

5)   The NISA file format does not allow for lamination sequence numbers to be defined, the lamination number is inferred from the number of definitions from the start of the *LAMSEQ data card. The Modelling Studio permits non-consecutive lamination numbers which could cause problems in the NISA file when exporting data. Consequently, dummy lamination sequences are created and put into the NISA file to pad out the *LAMSEQ data sets to arrange the correct numbers.

6)   The *PRINTCNTL data sets exported may cause extra analysis time and increase the NISA output file size beyond the minimum necessary, however it will ensure that no output is accidentally missed and thereby requiring a re-run of the model.

4.3.2   Display II Neutral File (.ne2)

Display II neutral files may be exported by the Modelling Studio. The following data cards are supported:-

1	Grids
2	Lines
3	Patches (from Modeller Surfaces)
21	Nodes
22	Elements

4.3.3   Display III Neutral File (.neu)

Display III neutral files may be exported by the Modelling Studio. The following data cards are supported:-

1	Grids
2	Lines
3	Patches
16	Material Data
17	Property Data
21	Nodes
22	Elements
104	Rigid Links
401	Displacement Constraints
402	Forces

Note:-   Display III does not support Nodal Pressures as separate entities, so on export the neutral file has the Nodal Pressures and Forces added together to form a single nodal load.

4.3.4   Display III Session File (.ses)

Display III session files may be exported by the Modelling Studio. The following data card is supported:-

FOR,ADD,

This is only used to export nodal pressures or combined nodal pressures and forces to Display III. It was particularly useful when the Modelling Studio had enough functionality to be a useable tool, but not enough to create a full model.

4.4   MYSTRAN Formats

4.3.1   MYSTRAN Data File (.dat)

MYSTRAN data files may be exported by the Modelling Studio. The following data cards are supported:-

        SOL1 and/or SOL3
        CEND
        TITLE
        SUBTITLE
        ECHO
        ELFORCE(BOTH)   Fixed output.
        ELSTRESS
        GPFORCES
        OLOAD
        SUBCASE
        LABEL
        LOAD
        SPCFORCES
        MPCFORCES
        STRESS
        BEGIN BULK
        CBAR
        CROD
        CELAS
        CHEXA
        CONM2
        CQUAD4
        CPENTA
        CTETRA
        CTRIA3
        FORCE
        GRAV
        GRID
        MAT1
        MAT8
        PARAM   ELFORCEN LOCAL.   Fixed output.
        PARAM   GRIDSEQ BANDIT   Fixed output.
        PBAR
        PCOMP
        PELAS
        PLOAD2
        PROD
        PSHELL
        PSOLID
        RBE2
        RFORCE
        SPC
        ENDDATA

Notes
1)   MYSTRAN treats rigid links as Elements with the RBE2 data card. When the rigid links are exported by the Modelling Studio, the RBE2 element number is formed by adding the Rigid Link number to the maximum element number.

2)   When writing an RFORCE data card the modeller adds a new GRID at the centre of rotation for each load case that has a rotational acceleration or velocity. The new grid is constrained using a SPC data card to restrain the grid in all directions for each load case in the analysis. The GRID number is automatically allocated as being the next available grid number after the existing GRID definitions. If the rotational acceleration contains both angular acceleration and angular velocity there will be two

RFORCE data cards for the load case – one for the acceleration and one for the velocity. At present this has not been tested in MYSTRAN.

3)   At the present it is not possible to apply pressure loads to solid elements in MYSTRAN. A future enhancement to the Modelling Studio may be to give the option of applying the pressure load as nodal pressure loads.

4)   MYSTRAN does not support Nodal Pressures as separate entities, so on export the MYSTRAN file has the Nodal Pressures and Forces added together to form a single nodal load.

5)   MYSTRAN versions V1 to V3 do not support laminates, so any laminated elements (Types 32 or 33) exported with the PCOMP property card may cause problems.

6)   At present MYSTRAN does not accept parabolic formulation shell elements. Any parabolic formulation shell elements are exported as linear formulation elements. The unused mid-side nodes and any boundary conditions on the mid-side nodes are not exported, because MYSTRAN would probably throw an error.

4.5   COSMOSWorks Formats

4.5.1   COSMOSWorks FE Geometry File (.geo)

COSMOSWorks FE Geometry data files may be imported by the Modelling Studio. The following data cards are supported:-

ACEL	Specifies Linear accelerations
EGROUP	Specifies element group
RCONST	Specifies element properties.
MPROP	Specifies material properties.
ACTSET EG	Sets current element group
ACTSET MP	Sets current element material
ACTSET RC	Sets current element properties
ND	Node
DND	Displacement Constraint
FND	Force on node.
EL PT	Point Element
EL CR	Line Element
EL RG	Shell Element
EL VL	Solid Element

Notes
1)   The Local Co-ordinate System used by COSMOSWorks is not compatible with NISA or MYSTRAN, so the LCS is ignored in the input. Consequently, loads and displacement constraints may be imported incorrectly – particularly if defined on cylindrical surfaces in COSMOSWorks. Fortunately the nodal co-ordinates are always exported in the global Cartesian co-ordinate system.

2)   COSMOSWorks only exports a single load case. Creation of multi-load case analyses will have to be carried out in the Modelling Studio.

3)   COSMOSWorks does not export remote loads or rigid links. These will have to be reproduced in the Modelling Studio. Also, in COSMOSWorks, it's possible to export assemblies where the items do not meet. This will be difficult to deal with. Gap elements etc will have to be added in the Modelling Studio.

4)   All solid elements import as Element Type 4, while all Shell elements import as General Shell Elements (Type 20). All Line elements import as 3D General Beam Elements (Type 12) and all Point elements import as 3D Mass elements (Type 26). The point elements may be the source of a problem. COSMOSWorks has 1 noded gap elements – a type not catered for by the Modelling Studio – as well as 1 noded mass elements. Please check that all the imported Type 26 mass elements should be mass elements and not gap elements. Delete all incorrect mass elements and create 2 noded gap elements in their stead. This cannot be done automatically – the Modelling Studio is *good*, but not that good!

5)   The COSMOSWorks file format has been inferred from examining output from known models. The author of the Modelling Studio had no access to the file format documentation – as a result there may be some unknown problems. Having said that – all problems encountered have been documented above.

4.6   Wavefront Object Format (.obj)

Wavefront Object Format files may be imported and exported by the Modelling Studio. The following data cards are supported:-

v	Vertex. Imported as a Node.
vt	Texture Vertex. Imported as a Texture Node. 2 dimensional texture maps only are catered for.
g	Group. Set current element Group.
usemtl	Use Material. Sets current Modelling Studio layer and, if the associated .mtl file exists, the element colour.
f	Facet. Imported as a General Shell Element (Type 20).

In the optional .mtl file the following is also read (but not exported):-

newmtl	Specifies layer name.
kd	Specifies element colour on the layer.

Notes

1)   The Wavefront Object format does not explicitly allocate a vertex or facet number; they are referenced by implication from the start of the object starting at 1, and it is possible to have more than one object per file. The Modelling Studio must have a unique node and element number, so the vertex and facet numbers are allocated on import. This can cause problems when creating morph targets if any of the nodes or element numbers are changed in the Modelling Studio. In addition, node renumbering can have a catastrophic effect on the morph target.

2)   Many functions within the modeller use a proximity test to eradicate problems due to decimal place inaccuracies. The accuracy test is set at 0.001 (by default), which may be a problem with Poser models (where 1 Poser Unit = 100 inches [Poser 6] or 96 inches [Poser 4 & 5]), where 0.001 is equivalent to 2.54mm or 2.43mm respectively. The work around is to scale the figure by 1000 after importing and by 0.001 just prior to exporting.

3)   There are a large number of programs that export Wavefront Object files. Not all of them are very precise about the type of carriage return or tab characters used. These can confuse the modeller import routine. If the model or texture template map is rubbish, abandon the model before working on it. The most commonly seen error is for the model to be distorted onto a diagonal line with equation $y=x$.
Load the Wavefront Object into another package (like DAZ| Studio – which is free) and export it. Open this file. The output from DAZ| Studio object export is perfectly understandable to the Modelling Studio and should be read easily.

4)   Most rendering packages can only cope with facets that have 3 or 4 vertices. Thus element faces with more than 4 vertices are converted to groups of triangular elements on export. An 8 noded face is exported as 6 triangular facets, while the 6 noded triangular face is exported as 4 triangular facets.
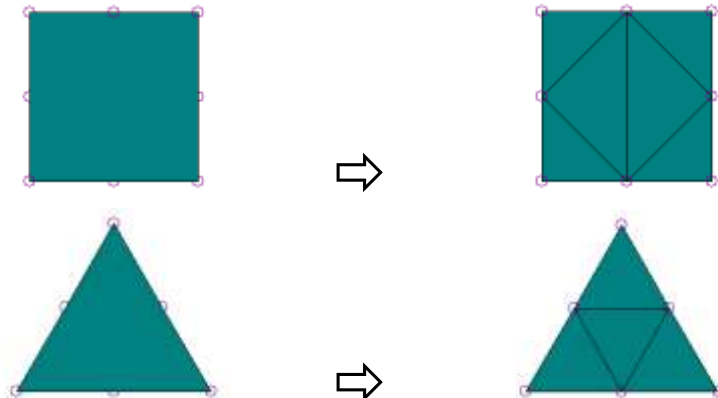
5)   The Wavefront Object format is a surface definition format only. Where solid elements are present in the model, element faces marked as internal faces are not exported.

6)   The Wavefront Object 'Material' imports as the Modelling Studio 'Layer'. It was felt that retaining the Wavefront Object 'Material' as 'material' properties would introduce irrelevant items into the FE material and confuses the issue; No properties of the Object 'Material' have any bearing on the FE Model 'Material', hence the decision to keep them separate. On export, the 'Layer' exports as the Wavefront Object 'Material'.

7)   Most Rendering Packages use the Y axis as the vertical axis. This can cause the model to have a difficult orientation for viewing. The answer is to rotate the whole model by 90° about the X Axis after import and -90° immediately before export.

8)   On import all facets are imported as general shell elements (Type 20) with Material 1, Property 1. All elements are visible. All vertices import as nodes, and are not visible.

9)   On export all Layer and Group names are checked for [space] characters, which are replaced with '_' characters as many rendering packages cannot cope with layer or group names with spaces in them. This way 'Layer 0' becomes 'Layer_0' in the exported file, rather than 'Layer' as would have occurred if the replacement had not been carried out.

4.7   Stereo Lithography Format (.stl)

Stereo Lithography Format files may be imported or exported by the Modelling Studio. Both ASCII and Binary format files are supported for import and only ASCII format for export.

Notes
1)   The STL file represents a solid model. The Modelling studio imports the surface of the solid as a shell model, *not* as a solid.

2)   The facets in the STL file may be imported as Geometry (Points, Lines and Surfaces) or as Finite Element Entities (Nodes and Shell Elements). The decision between the two is made from within the Modelling Studio by selecting '**File| File Preferences| Import: STL as FE Entities**'. The selection is persistent.

3)   STL files do not often display well on the screen as there is usually a large difference between the maximum and minimum facet size, leading to facets being drawn on screen in a confusing order.

4)   The vertices at the corners of the facets are duplicated on import, and then they are merged. When a STL model is imported as elements and nodes, the new nodes are not merged with the existing model. When a STL model is imported as geometry, duplicate imported points, lines and surfaces are merged with the existing model, as duplicates of these entity types are not permitted within the Modelling Studio.

5)   At the moment only surfaces export, and if you have '**File| File Preferences| Import: STL as FE Entities**' selected, no facets will be written.

4.8   DXF Format (.dxf)

DXF Format files may be imported and exported by the Modelling Studio. The following data cards are supported:-

POINT   Modelling Studio Point entity.
LINE   Modelling Studio Straight Line entity.
3DFACE   Modelling Studio Surface entity with two straight lines.
62   Entity colour. Ignored on import.
8   Entity Layer, corresponds to Modelling Studio Layer.

Notes
1)   The entities converted are a *very* small subset of the entities available in the DXF format, but should be useful for some modelling purposes.

2)   FE Model entities cannot be exported to the DXF format.

3)   The DXF format is a surface definition format only, so Modelling Studio Solids will never be exported.

4)   Some older DXF compatible programs cannot cope with spaces within layer names, so all spaces are converted to the underscore "_" character on export, for example 'Layer 0' becomes 'Layer_0' on export.

5)   Blocks should be exploded before saving the DXF file in the CAD program. The entities grouped together in blocks are imported, but the block references are ignored in the main body of the DXF file.

6)   Duplicate imported points, lines and surfaces are merged with the existing model, as duplicates of these entity types are not permitted within the Modelling Studio.

4.9   Radioss Format (.rad)

Radioss .rad Format files may be imported by the Modelling Studio. The following data cards are supported:-

/NODE	Modelling Studio Node.
/SHELL	4 Noded Shell Element (Type 20)
/SH3N	3 Noded Shell Element. (Type 20)
/BRICK	8 Noded Brick Element. (Type 4, Node Order 1)
/BRIC20	20 Noded Brick Element. (Type 4, Node Order 2)
/TETRA 4	4 Noded Tetrahedral Brick Element (Type 4, Node Order 10).
/TETRA 10	10 Noded Tetrahedral Brick Element (Type 4, Node Order 11)
/ACEL	Acceleration data set
/EGROUP	Element Group data set
/PROP/SHELL	Shell properties data set
/MAT/PLAS_JOH	Material data set
/MAT/JOHNS	Material data set
/RBODY	Rigid Link, but only if the next line is a /GRNOD/NODE/ data set

Notes

1)   The entities converted are a *very* small subset of the entities available in the Radioss format, but should be useful for some modelling purposes.

2)   At the moment it is an import only format.

3)   Radioss files can come in two data card types, with long or short number types. This should be set in the NISATools.ini file before importing the .rad file, under
[Modeller]
RadiossFieldSize=8, 10 or 16

4)   Radioss can wrap the title line, so in the NISATools.ini file, before importing the radios file, set the following:
[Modeller]
RadiossWrapTitleLine=0 or 1

5)   Node numbers, Element numbers etc are offset when imported into an existing model.

5.   Element Library

5.1   General Information

5.1.1   Section 5 is the reference section for the elements that may be used in the FEA model. This section details the elements that may be used by the Modelling Studio. The Element Type numbers used in the Modelling Studio are identical to those used by NISA. If you have a NISA manual available, you will find more information in that. If you do not have a NISA manual, it is hoped that the Library description below is sufficiently comprehensive for your needs - if not, contact the author at either of the e-mail addresses given in the suggestions section.

5.1.2   The element library list includes only those elements that you are encouraged to use. The Modelling Studio can cope with all NISA elements types, however the Modelling Studio is aimed at general FE stress analysis, so elements with a too specific application (eg thermal & pipes) are not documented here.

5.2   Element Library Details

The element details are as below:-
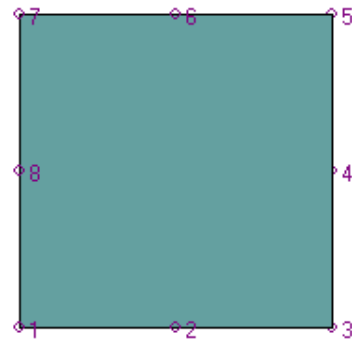
5.2.1   Element Type 1: 2D Plane Stress Element

Node Order 1, 4 nodes.
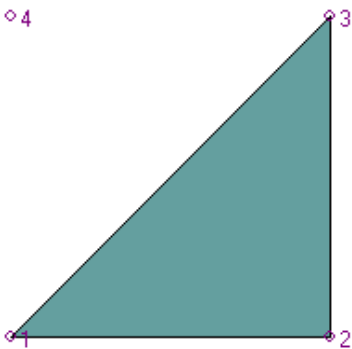

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.
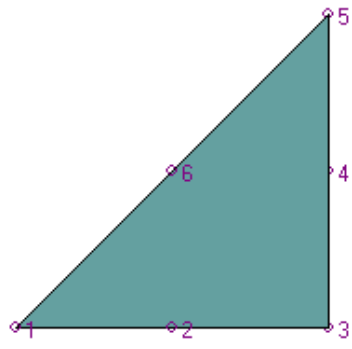

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9,12 for elements loaded into Modelling Studio.

Properties:-
Nodal Thicknesses. Same number as number of nodes. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 8 properties are always specified.
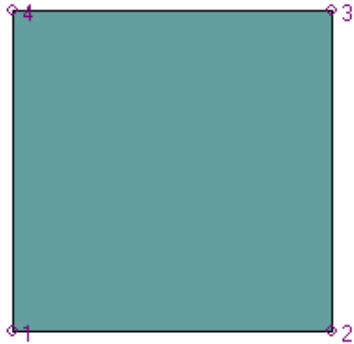
Material Properties.
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ (Orthotropic Material).

Degrees of Freedom at Nodes.
UX, UY.

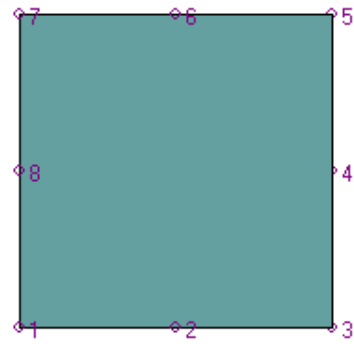Notes
1)   Must be on Global XY Plane, in +X half.

5.2.2   Element Type 2: 2D Plane Strain Element

Node Order 1, 4 nodes.
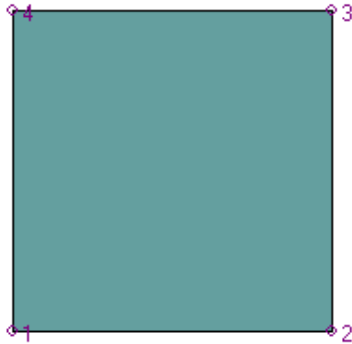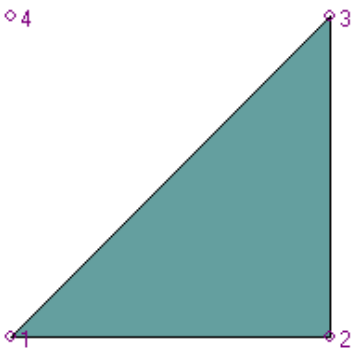
Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.
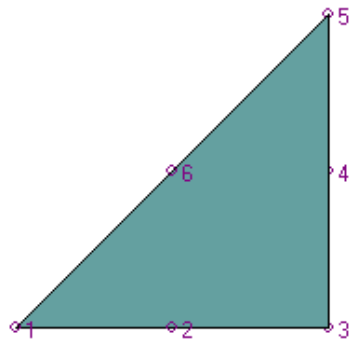
Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9,12 for elements loaded into Modelling Studio.

Properties:-
Not Required.

Material Properties.
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ (Orthotropic Material).

Degrees of Freedom at Nodes.
UX, UY.

Notes
1)   Must be on Global XY Plane, in +X half.

5.2.3   Element Type 3: Axisymmetric Solid Element

Node Order 1, 4 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9,12 for elements loaded into Modelling Studio.

Properties
Not Required.

Material Properties
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ (Orthotropic Material).

Degrees of Freedom at Nodes
UX, UY.

Notes
1)   Global Y axis is the rotational axis. All elements to be in the +X half of the model, on the XY
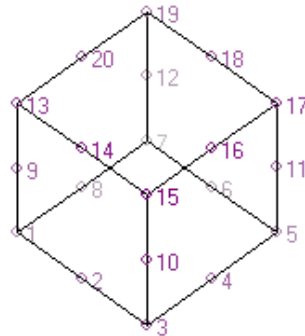   Plane.

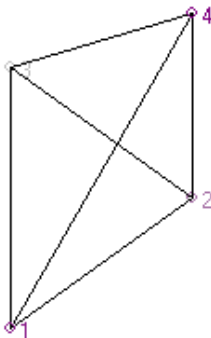### 5.2.4   Element Type 4: 3D Thick Solid Element

Node Order 1, 8 nodes.
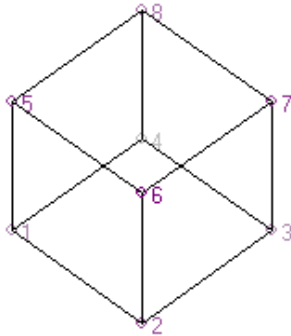
Node Order 2, 20 nodes.

Face   1: Nodes 1,2,3,4
        2: Nodes 5,6,7,8
        3: Nodes 1,2,6,5
        4: Nodes 2,3,7,6
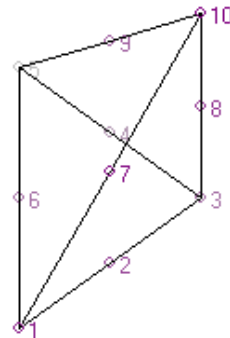   5: Nodes 3,4,8,7
        6: Nodes 4,1,5,8

Face   1: Nodes 1,2,3,4,5,6,7,8
        2: Nodes 13,14,15,16,17,18,19,20
        3: Nodes 1,2,3,10,15,14,13,9
        4: Nodes 3,4,5,11,17,16,15,10
        5: Nodes 5,6,7,12,19,18,17,11
        6: Nodes 7,8,1,9,13,20,19,12

Node Order 20, 4 nodes.

Node Order 21, 10 nodes.

Face   1: Nodes 1,2,3
        2: Nodes 1,2,4
        3: Nodes 2,3,4
        4: Nodes 3,1,4

Face   1: Nodes 1,2,3,4,5,6
        2: Nodes 1,2,3,8,10,7
        3: Nodes 3,4,5,9,10,8
        4: Nodes 5,6,1,7,10,9

Also NISA compatible Node Orders 10,11,12,30 & 31 for elements imported into Modelling Studio.
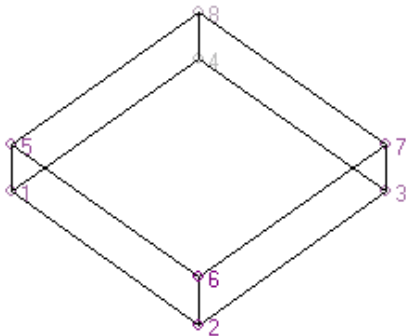
Properties
Not Required.

Material Properties
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GXZ, GYZ, DENS, ALPX, ALPY, ALPZ (Orthotropic Material).

Degrees of Freedom at Nodes
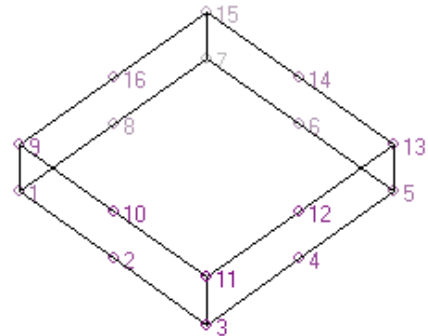UX, UY, UZ.

Notes
1)   None.

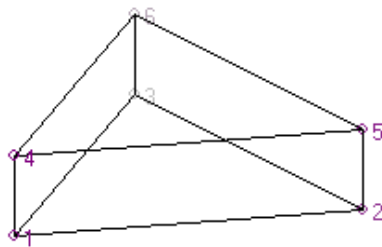### 5.2.5   Element Type 5: 3D Thick Shell Element

Node Order 1, 8 nodes.   Node Order 2, 16 nodes.

Face   1: Nodes 1,2,3,4   Face   1: Nodes 1,2,3,4,5,6,7,8
       2: Nodes 5,6,7,8   2: Nodes 9,10,11,12,13,14,15,16
       3: Nodes 1,2,6,5   3: Nodes 1,2,3,11,10,9
       4: Nodes 2,3,7,6   4: Nodes 3,4,5,13,12,11
       5: Nodes 3,4,8,7   5: Nodes 5,6,7,12,19,18,17,11
       6: Nodes 4,1,5,8   6: Nodes 7,8,1,9,13,20,19,12

Node Order 10, 6 nodes.   Node Order 11, 12 nodes.
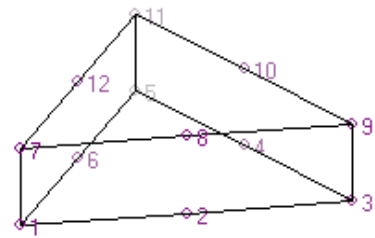
Face   1: Nodes 1,2,3   Face   1: Nodes 1,2,3,4,5,6
       2: Nodes 4,5,6   2: Nodes 7,8,9,10,11,12
       3: Nodes 1,2,5,4   3: Nodes 1,2,3,9,8,7
       4: Nodes 2,3,6,5   4: Nodes 3,4,5,11,10,9
       5: Nodes 3,1,4,6   5: Nodes 5,6,1,7,12,11

Also NISA compatible Node Orders 7,8,9 for elements imported into Modelling Studio.

Properties
Not Required.

Material Properties
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GXZ, GYZ, DENS, ALPX, ALPY, ALPZ (Orthotropic
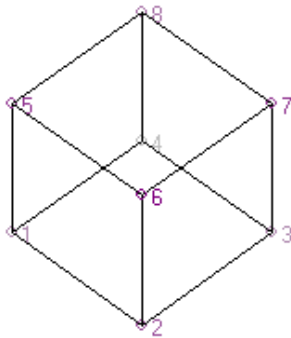Material).

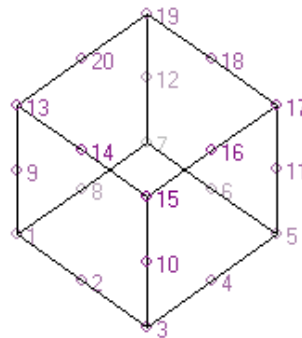Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
1)   None.

5.2.7   Element Type 7: 3D Laminated Solid Element

Node Order 1, 8 nodes.   Node Order 2, 20 nodes.

Face	1: Nodes 1,2,3,4	Face	1: Nodes 1,2,3,4,5,6,7,8
	2: Nodes 5,6,7,8		2: Nodes 13,14,15,16,17,18,19,20
	3: Nodes 1,2,6,5		3: Nodes 1,2,3,10,15,14,13,9
	4: Nodes 2,3,7,6		4: Nodes 3,4,5,11,17,16,15,10
	5: Nodes 3,4,8,7		5: Nodes 5,6,7,12,19,18,17,11
	6: Nodes 4,1,5,8		6: Nodes 7,8,1,9,13,20,19,12

Lamination Sequence
Specified in as element material number.

Laminate Fibre Angles
Specified at each node on top face. First specified angle is taken as representative for the layer.

Properties
4 Layer Thicknesses for each layer.

Material Properties
Orthotropic Elastic Properties:- EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GXZ, GYZ, DENS, ALPX, ALPY, ALPZ .
Strength Properties:- FXT, FXC, FYT, FYC, FZT, FZC, FSXY, FSXZ, FSYZ, F12, F13, F23.

Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
1)   The laminate sequence starts with layer 1 on the top face (face 2).
2)   To find how to create a laminate see the 'How to Create Laminates' user guide.
3)   The element formulation is such that if the layer thicknesses do not add up to the total element thickness, the layer thicknesses are scaled to fit. It is suggested that the layer thicknesses are specified as percentages of the laminate thickness.

5.2.9   Element Type 9: 3D Hybrid Solid Element

Node Order 1, 8 nodes.
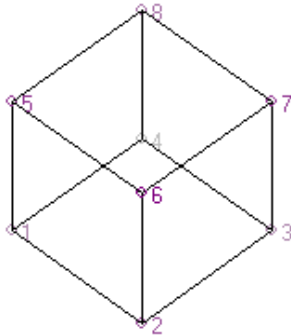
Face   1: Nodes 1,2,3,4
         2: Nodes 5,6,7,8
         3: Nodes 1,2,6,5
         4: Nodes 2,3,7,6
         5: Nodes 3,4,8,7
         6: Nodes 4,1,5,8

Properties
Not Required.

Material Properties
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GXZ, GYZ, DENS, ALPX, ALPY, ALPZ (Orthotropic Material).
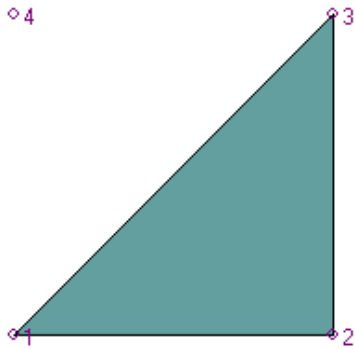
Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
1)   May be used for solids or shells, where the smallest edge length is greater than 1/1000th the maximum edge length.

5.2.10 Element Type 10: 3D Membrane Element

<u>Node Order 10, 3 nodes.</u>

Edge   1: Nodes 1,2
          2: Nodes 2,3
          3: Nodes 3,1
Face   1: Bottom face
          2: Top Face (Shown)

<u>Properties</u>
Nodal thicknesses. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that at least 3 properties are always specified.

<u>Material Properties</u>
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, NUXY, GXY, DENS, ALPX, ALPY (Orthotropic Material).
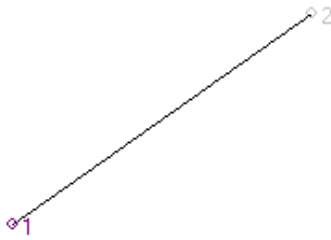
<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ.

<u>Notes</u>
1)   The shell element plane is the midplane of the element.

5.2.12 Element Type 12: 3D Beam Element

Node Order 1, 2 nodes.

Element axes defined in note 1.

Properties
Up to 43 element property entries in one property set:-
1   Cross Sectional Area
2   IYY
3   IZZ
4   J
5   Eccentricity in Y direction
6   Eccentricity in Z direction
7   Depth in Y direction (for temperature gradients)
8   Depth in Z direction (for temperature gradients)
9   ALPHAY. Shear co-efficient in Y direction. ALPHAY = 1.111 for solid circle. ALPHAY = 1.2 for solid rectangle. ALPHAY = 1.887 for circular tube. ALPHAY = 2.000 for square tube. ALPHAY $\approx$ Area/Web Area for I beam.
10   ALPHAZ. Shear co-efficient in Z direction. ALPHAZ = 1.111 for solid circle. ALPHAZ = 1.2 for solid rectangle. ALPHAZ = 1.887 for circular tube. ALPHAZ = 2.000 for square tube. ALPHAZ $\approx$ Area/Web Area for I beam.
11   Node 1 X direction beam centroid offset.
12   Node 1 Y direction beam centroid offset.
13   Node 1 Z direction beam centroid offset.
14   Node 2 X direction beam centroid offset.
15   Node 2 Y direction beam centroid offset.
16   Node 2 Z direction beam centroid offset.
17   OKEY. See note 1.
18   THETA or X3. See note 1.
19   Y3. See note 1.
20   Z3. See note 1.
21   SKEY. Number of points on cross section (0 to 4) for stress calculation
22   TKEY. Section type key. TKEY = 0, solid or thick walled sections. TKEY = 1, open, thin walled sections. TKEY = 2, closed, thin walled sections.
23   1st stress point y co-ordinate.
24   1st stress point z co-ordinate.
25   Torsional shear factor
26   Y shear due to bending factor.
27   Z shear due to bending factor.
28-32   Repeat 23-27 for 2nd stress point.
33-37   Repeat 23-27 for 3rd stress point.
38-42   Repeat 23-27 for 4th stress point.
43   IYZ. Product of moment of inertia in yz plane.

Material Properties
EX, NUXY, DENS, ALPX (Isotropic Material)

Degrees of Freedom at Nodes
UX, UY, UZ, RotX, RotY, RotZ.

Notes
1)   Element X axis is from node 1 to node 2.
        If OKEY (property 17) = 0
        Then:- Element XZ plane is defined by the vectors (node2-node 1) and (0,0,1). The element Z
        axis is on the element XZ plane, and is perpendicular to the element X axis. The Element Y
        axis is perpendicular to the element X & Z axes, using the right hand rule. If the element is
        parallel to the global Z axis, the procedure is carried out using with the global vector (0,1,0)
        instead of (0,0,1). The element rotation (THETA, property 18) is a rotation of the previously
        calculated element Y & Z axes in degrees about the element X axis, using the right hand rule.
        If OKEY (property 17) = 1
        The element XY plane is defined by node 1, node 2 and the third point (co-ordinates X3, Y3,
        Z3) defined in property entries 18, 19 & 20.

5.2.13 Element Type 13: 2D Beam Element

Node Order 1, 2 nodes.
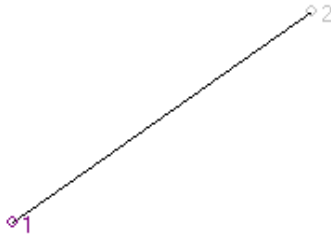
Properties
4 element property entries in one property set:-
1   A. Cross Sectional Area.
2   IZZ.
3   ALPHAY. Shear co-efficient. ALPHAY = 1.111 for solid circle. ALPHAY = 1.2 for solid
          rectangle. ALPHAY = 1.887 for circular tube. ALPHAY = 2.000 for square tube. ALPHAY ≈
          Area/Web Area for I beam.
4   BY. Local y co-ordinate for stress recovery.

Material Properties
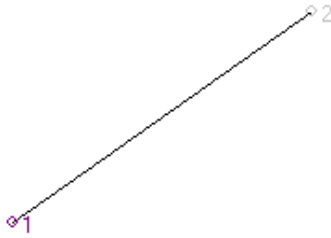EX, NUXY, DENS, ALPX (Isotropic Material)

Degrees of Freedom at Nodes
UX, UY, RotZ.

Notes
          1)   Nodes 1 & 2 must be in the XY plane.

5.2.14 Element Type 14: 3D Spar Element

Node Order 1, 2 nodes.

Properties
Up to 5 element property entries in one property set:-
1   A1. Cross Sectional Area at node 1.
2   A2. Cross Sectional Area at node 2.
3-5 are for non-linear analysis only, and the Modelling Studio cannot specify non-linear analysis, so leave as 0.
3   TENSIN. Initial Tension.
4   TCKEY. Member type. TCKEY = -1; Compression only strut. TCKEY = 0; Tension/ Compression strut. TCKEY = 1; Tension only strut.
5   C. Artificial stiffness for tension/ compression only members when loaded in reverse direction.

Material Properties
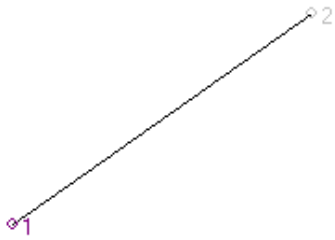EX, DENS, ALPX (Isotropic Material)

Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
        1)   None.

5.2.15 Element Type 15: 2D Spar Element

<u>Node Order 1, 2 nodes.</u>

<u>Properties</u>
2 element property entries in one property set:-
1   A1. Cross Sectional Area at node 1.
2   A2. Cross Sectional Area at node 2.

<u>Material Properties</u>
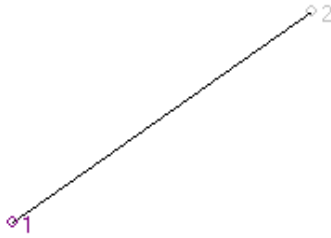EX, DENS, ALPX (Isotropic Material)

<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ.

<u>Notes</u>
         1)   Nodes 1 & 2 must be in the XY Plane.

5.2.17 Element Type 17: 3D Spring Element

<u>Node Order 1, 2 nodes.</u>

<u>Properties</u>
Up to 7 element property entries in one property set:-
1   K. Spring Stiffness.
2-4   Vector direction of spring axis if node 1 & node 2 are co-incident. Do not use the element this way! You'll get in a muddle.
5   Co-ordinate system for 2-4. Set to 0.
6   Non-linear spring data number. The Modelling Studio cannot specify non-linear analysis, so set as 0.
7   Non-linear analysis initial deflection. The Modelling Studio cannot specify non-linear analysis, so set as 0.

<u>Material Properties</u>
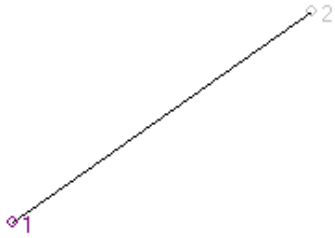None

<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ.

<u>Notes</u>
        1)   None.

5.2.18 Element Type 18: 2D Spring Element

<u>Node Order 1, 2 nodes.</u>

<u>Properties</u>
Up to 5 element property entries in one property set:-
1	K. Spring Stiffness.
2-4	Vector direction of spring axis if node 1 & node 2 are co-incident. Do not use the element this way! You'll get in a muddle.
5	Co-ordinate system for 2-4. Set to 0.

<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
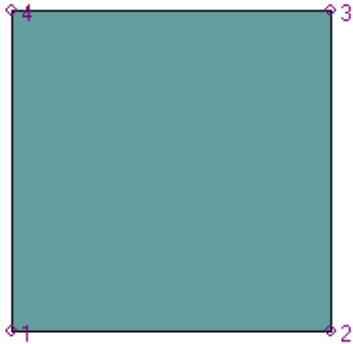UX, UY.

<u>Notes</u>
   1)   Nodes 1 & 2 must be on the XY plane.
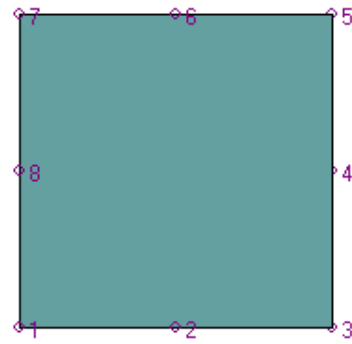
5.2.20 Element Type 20: 3D General Shell Element

Node Order 1, 4 nodes.
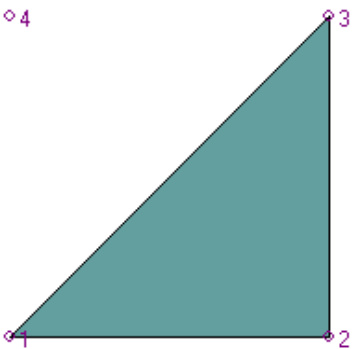

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.
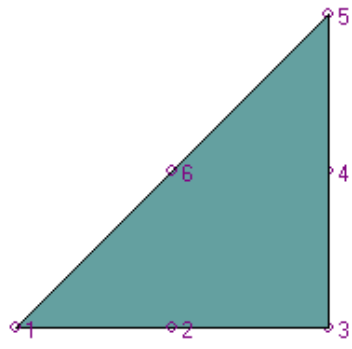

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9 for elements loaded into Modelling Studio.

Face   1: Bottom face
        2: Top Face (Shown above)

Properties:-
Nodal Thicknesses. Same number as number of nodes. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 8 properties are always specified.

Material Properties.
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY (Orthotropic Material).
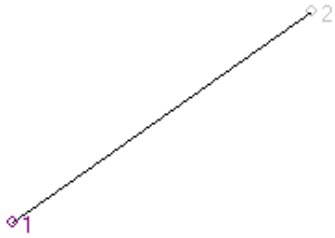
Degrees of Freedom at Nodes.
UX, UY, UZ, RotX, RotY, RotZ.

Notes
1)   The shell element plane is the midplane of the element.

5.2.21 Element Type 21: 3D Torsional Spring Element

Node Order 1, 2 nodes.

Properties
5 element property entries in one property set:-
1   K. Spring Stiffness.
2-4   Vector direction of spring axis if node 1 & node 2 are co-incident. Do not use the element this
         way! You'll get in a muddle.
5   Co-ordinate system for 2-4. Set to 0.

Material Properties
None

Degrees of Freedom at Nodes
RotX, RotY, RotZ.

Notes
        1)   None.

5.2.22 Element Type 22: 2D Torsional Spring Element

<u>Node Order 1, 2 nodes.</u>

<u>Properties</u>
5 element property entries in one property set:-
1   K. Spring Stiffness.
2-4   Vector direction of spring axis if node 1 & node 2 are co-incident. Do not use the element this
         way! You'll get in a muddle.
5   Co-ordinate system for 2-4. Set to 0.

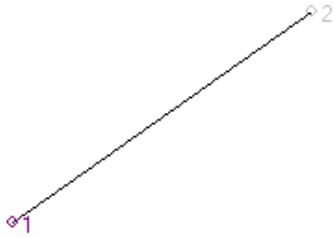<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
RotX, RotY.

<u>Notes</u>
        1)   Nodes 1 & 2 must be in the XY Plane.

5.2.25 Element Type 25: 2D Mass Element

Node Order 1, 1 node.

■1

Properties
1 element property entry in one property set:-
1   Mass.

Material Properties
None

Degrees of Freedom at Nodes
UX, UY.

Notes
        1)   Node 1 must be in the XY Plane.

5.2.26 Element Type 26: 3D Mass Element

<u>Node Order 1, 1 node.</u>

■1

<u>Properties</u>
1 element property entry in one property set:-
1   Mass.

<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ.

<u>Notes</u>
        1)   None.

5.2.27 Element Type 27: 2D General Mass Element

Node Order 1, 1 node.

■1

Properties
2 element property entries in one property set:-
1   Mass in X Direction.
2   Mass in Y Direction.

Material Properties
None

Degrees of Freedom at Nodes
UX, UY.

Notes
        1)   Node 1 must be in the XY Plane.

5.2.28 Element Type 28: 3D General Mass Element

<u>Node Order 1, 1 node.</u>

■1

<u>Properties</u>
3 element property entries in one property set:-
1   Mass in X Direction.
2   Mass in Y Direction.
3   Mass in Z Direction.

<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ.

<u>Notes</u>
        1)   None.

5.2.29 Element Type 29: 2D General Mass Element with Rotary Inertia

<u>Node Order 1, 1 node.</u>

■1

<u>Properties</u>
3 element property entries in one property set:-
1   Mass in X Direction.
2   Mass in Y Direction.
3   Rotary Inertia.

<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
UX, UY, RotZ.

<u>Notes</u>
          1)   Node 1 must be in the XY Plane.

5.2.30 Element Type 30: 3D General Mass Element with Rotary Inertia

<u>Node Order 1, 1 node.</u>

■1

<u>Properties</u>
6 element property entries in one property set:-
1   Mass in X Direction.
2   Mass in Y Direction.
3   Mass in Z Direction
4   Rotary Inertia about X axis.
5   Rotary Inertia about Y axis.
6   Rotary Inertia about Z axis.

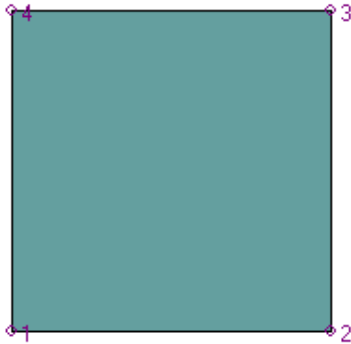<u>Material Properties</u>
None

<u>Degrees of Freedom at Nodes</u>
UX, UY, UZ, RotX, RotY, RotZ.

<u>Notes</u>
        1)   None.

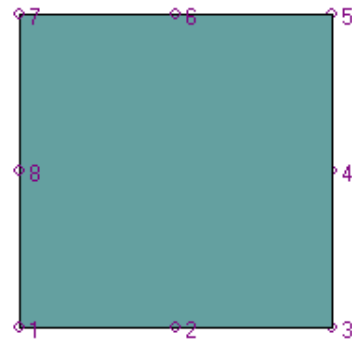### 5.2.32 Element Type 32: 3D Laminated Composite General Shell Element

Node Order 1, 4 nodes.
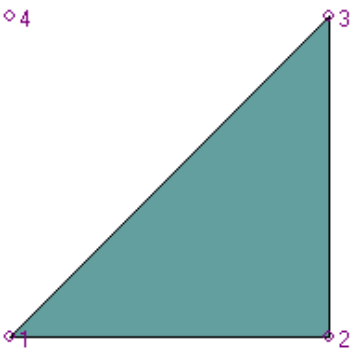
Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.
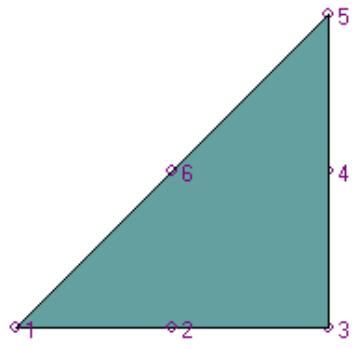
Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9 for elements loaded into Modelling Studio.

Face   1: Bottom face
        2: Top Face (Shown above)

Lamination Sequence
Specified in as element material number.

Laminate Fibre Angles
Specified at each node.

Properties
Layer Thicknesses at each node. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 8 properties are always specified.

Material Properties for each Layer
Orthotropic Elastic Properties:- EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY.
Strength Properties:- FXT, FXC, FYT, FYC, FS, F12.

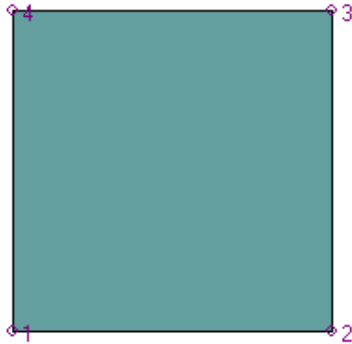Degrees of Freedom at Nodes
UX, UY, UZ, RotX, RotY, RotZ.

Notes
1)   The laminate sequence starts with layer 1 on the top face (face 2).
2)   To find how to create a laminate see the 'How to Create Laminates' user guide.

3)   The shell element plane is the midplane of the element.
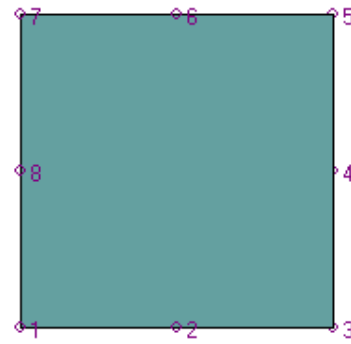
5.2.33   Element Type 33: 3D Laminated Sandwich General Shell Element

Node Order 1, 4 nodes.
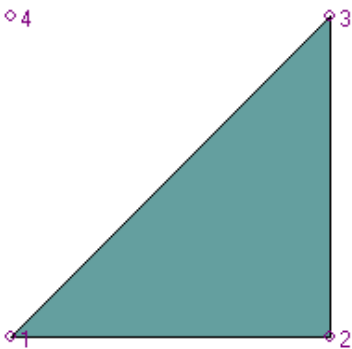
Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 2, 8 nodes.
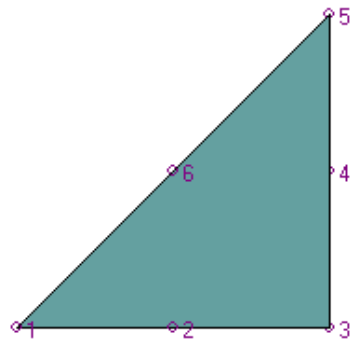
Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,7
        4: Nodes 7,8,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Node Order 11, 6 nodes.

Edge   1: Nodes 1,2,3
        2: Nodes 3,4,5
        3: Nodes 5,6,1

Also NISA compatible Node Orders 3,4,5,6,7,8,9 for elements loaded into Modelling Studio.

Face   1: Bottom face
        2: Top Face (Shown above)

Lamination Sequence
Specified in as element material number.

Laminate Fibre Angles
Specified at each node.

Properties
Layer Thicknesses at each node. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 8 properties are always specified.

Material Properties for each Layer
Orthotropic Elastic Properties:- EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY.
Strength Properties:- FXT, FXC, FYT, FYC, FS, F12.

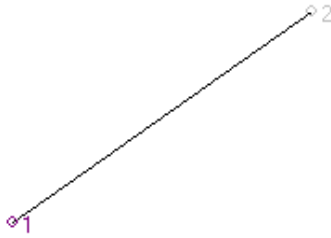Degrees of Freedom at Nodes
UX, UY, UZ, RotX, RotY, RotZ.

Notes
1)   The laminate sequence starts with layer 1 on the top face (face 2).

2)   To find how to create a laminate see the 'How to Create Laminates' user guide.
3)   The core material must have EX = 0.
4)   The shell element plane is the midplane of the element.

5.2.36 Element Type 36: Axisymmetric Shell Element

Node Order 1, 2 nodes.

Also NISA compatible Node Orders 2,3 for elements loaded into Modelling Studio.

Properties
Thickness at each node. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 8 properties are always specified.

Material Properties
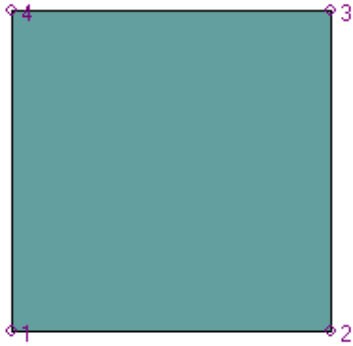EX, NUXY, DENS, ALPX (Isotropic Material)

Degrees of Freedom at Nodes
UX, UY, RotZ.

Notes
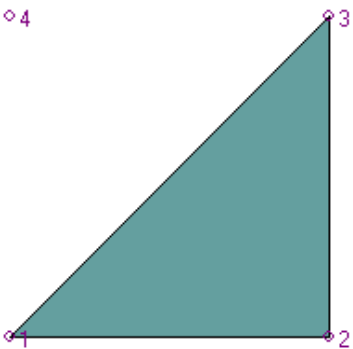1)   Global Y axis is the rotational axis. All elements to be in the +X half of the model, on the XY Plane.

5.2.40 Element Type 40: 3D Thin Shell Element

Node Order 1, 4 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,4
        4: Nodes 4,1

Node Order 10, 3 nodes.

Edge   1: Nodes 1,2
        2: Nodes 2,3
        3: Nodes 3,1

Face   1: Bottom face
        2: Top Face (Shown above)

Properties:-
Nodal Thicknesses. Same number as number of nodes. Properties can specify more nodal thicknesses than there are nodes, but never less. It is suggested that 4 properties are always specified.

Material Properties.
EX, NUXY, DENS, ALPX (Isotropic Material)
EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY (Orthotropic Material).
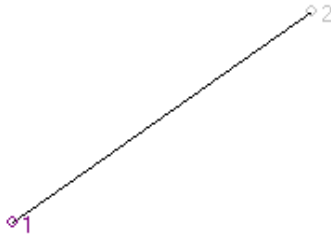
Degrees of Freedom at Nodes.
UX, UY, UZ, RotX, RotY, RotZ.

Notes
1)   The shell element plane is the midplane of the element.

5.2.42 Element Type 42: 2D or Axisymmetric Gap Element

Node Order 1, 2 nodes.

Properties
5 element property entries in one property set:-
1   K. Axial Stiffness for unrestrained Rigid Body Motion
2   F. Flexibility co-efficient. F = 0 for Gap elements. F > 0 for tension only members. F < 0 for compression only members.
3   Initial Gap Width. This MUST be specified. It is not derived from the node positions.
4   Gap Convergence tolerance. Default = 1×10^{-8}.
5   Gap orientation angle from X = 0 axis. This MUST be specified. It is not derived from the node positions.
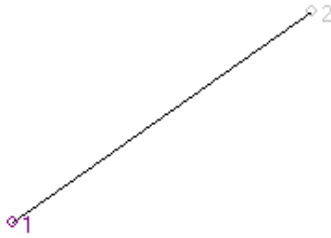
Material Properties
None

Degrees of Freedom at Nodes
UX, UY.

Notes
1)   Global Y axis is the rotational axis. All elements to be in the +X half of the model, on the XY Plane.

5.2.43 Element Type 43: 3D Gap Element

Node Order 1, 2 nodes.

Properties
4 element property entries in one property set:-
1   K. Axial Stiffness for unrestrained Rigid Body Motion
2   F. Flexibility co-efficient. F = 0 for Gap elements. F > 0 for tension only members. F < 0 for compression only members.
3   Initial Gap Width. This MUST be specified. It is not derived from the node positions. +ve = initially open. –ve = initial interference.
4   Gap Convergence tolerance. Default = 1×10^{-8}.

Material Properties
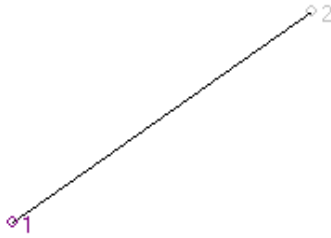None

Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
1)   None.

5.2.49 Element Type 49: 2D Gap/ Friction Element

Node Order 1, 2 nodes.

Properties
9 element property entries in one property set:-
1   K_a. Axial Stiffness
2   K_t. Tangential Stiffness
3   S_a. Small axial stiffness for unrestrained Rigid Body Motion. $S_a \ll K_a$.
4   S_t. Small tangential stiffness for unrestrained Rigid Body Motion. $S_t \ll K_t$.
5   RGKEY. Gap definition method. RGKEY = 0, gap width defined by property entry 6. RGKEY $\geq$ 1.0, gap defined by node positions.
6   Initial Gap Width if RGKEY = 0.
7   RTKEY. Angle definition method. RTKEY = 0, angle defined by property entry 8. RTKEY $\geq$ 1.0, angle is defined by node positions.
8   THETA. Angle of gap from X axis if RTKEY = 0.
9   Gap Convergence tolerance. Default = 1×10^{-8}.

Material Properties
Friction co-efficient as NUXY.
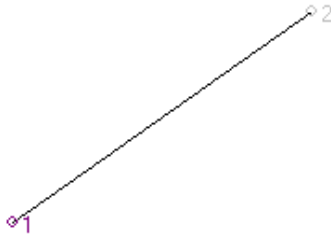
Degrees of Freedom at Nodes
UX, UY.

Notes
1)   Global Y axis is the rotational axis. All elements to be in the +X half of the model, on the XY Plane.

5.2.50 Element Type 50: 3D Gap/ Friction Element

Node Order 1, 2 nodes.

Properties
14 element property entries in one property set:-
1   K_a. Axial Stiffness
2   K_t. Tangential Stiffness
3   S_a. Small axial stiffness for unrestrained Rigid Body Motion. $S_a \ll K_a$.
4   S_t. Small tangential stiffness for unrestrained Rigid Body Motion. $S_t \ll K_t$.
5   RGKEY. Gap definition method. RGKEY = 0, gap width defined by property entry 6. RGKEY $\geq$ 1.0, gap defined by node positions.
6   Initial Gap Width if RGKEY = 0.
7   RTKEY. Angle definition method. RTKEY = 0, angle defined by property node positions. RTKEY $\geq$ 1.0, angle is defined by vector given by 8-10.
8-10   X,Y,Z vector components of gap.
11-13 Optional vector in the local XY plane.
14   Gap Shear Force Convergence tolerance. Default = 1 x 10^{-3}.

Material Properties
Friction co-efficient as NUXY.

Degrees of Freedom at Nodes
UX, UY, UZ.

Notes
1)   None.

6.   Suggestions

If you have any comments/ suggestions/ queries, then please feel free to contact the programmer for this program at:-

1.   richard.williams@epslt.co.uk   or
2.   richardandtracy@yahoo.com

Please make the subject something sensible, including the words 'Modelling Studio' – if you don't, the query will be automatically filtered into 'Junk Mail' and will not be read. Depending on holidays etc, an acknowledgement of/ response to an e-mail will be sent as soon as is possible.

There are no promises to include everything (or anything); however, all suggestions will be considered.

A1   Appendix 1

The following is the session file to create the garden table referred to in the 'How to Texture Map a Model' guide. To use this session file, copy the text, open a text editor tool in the Modelling Studio and paste the text. Save the text as 'Table.msf' somewhere, and then close the text editor. Press menu button '**File**' and select the 'Table.msf' file from the quick list and open it with the Modelling Studio. The garden table model will be re-created.

```
Point,Add,1,0,0.5,0,0
Point,Add,2,0,0.5,0.25,0
Line,Add,1,0,1,2
Point,Add,3,0,0,0.5,0
Point,Add,4,0,0.25,0.5,0
Line,Add,2,0,3,4
Line,Select,1
Plot,Refresh
Point,Add,5,0,0.5,0.75,0
Point,Add,6,0,0.5,1,0
Line,Add,3,0,5,6
Select,Flag,On
Select,None
Line,Select,2
Plot,Refresh
Point,Add,7,0,0.75,0.5,0
Point,Add,8,0,1,0.5,0
Line,Add,5,0,7,8
Select,None
Line,Select,1
Line,Select,2
Line,Select,3
Line,Select,5
Plot,Refresh
Point,Add,9,0,0.5,0,1
Point,Add,10,0,0.5,0.25,1
Line,Add,6,0,9,10
Point,Add,11,0,0,0.5,1
Point,Add,12,0,0.25,0.5,1
Line,Add,7,0,11,12
Point,Add,13,0,0.5,0.75,1
Point,Add,14,0,0.5,1,1
Line,Add,8,0,13,14
Point,Add,15,0,0.75,0.5,1
Point,Add,16,0,1,0.5,1
Line,Add,9,0,15,16
Select,None
Line,Select,None
Surface,Add,2Line,1,0,Line,2,0,Line,7,1,
Surface,Add,2Line,2,0,Line,1,0,Line,6,1,
Surface,Add,2Line,3,0,Line,3,0,Line,8,1,
Surface,Add,2Line,4,0,Line,5,0,Line,9,1,
Point,Add,17,0,0,0,1
Point,Add,18,0,1,0,1
Line,Add,10,0,17,18
Line,Select,10
Plot,Refresh
Point,Add,19,0,0,1,1
Point,Add,20,0,1,1,1
Line,Add,11,0,19,20
Select,None
Line,Select,None
Surface,Add,2Line,5,0,Line,10,0,Line,11,1,
View,UserDefined,-60,0,20
Line,Add,12,0,4,12
Line,Add,13,0,5,13
Line,Add,14,0,7,15
Line,Add,15,0,2,10
Snap,Point,Off
```

```
Snap,Line,End,Off
Snap,Line,Middle,On
Snap,Line,Perpendicular,Off
Snap,Line,Nearest,Off
Snap,Surface,Centroid,Off
Snap,Node,On
Snap,ElementCentroid,Off
Snap,ElementFaceCentroid,Off
Point,Add,21,0,0.5,0.25,0.5
Point,Add,22,0,0.75,0.5,0.5
Line,Add,16,0,21,22
Point,Add,23,0,0.25,0.5,0.5
Point,Add,24,0,0.5,0.75,0.5
Line,Add,17,0,23,24
Line,Select,None
Surface,Add,2Line,6,0,Line,16,0,Line,17,1,
View,UserDefined,-60,0,30
View,UserDefined,-55,0,15
Surface,Mesh,Set,20,1,1,1,12,3,1
Surface,Mesh,Set,20,1,1,1,12,12,1
Surface,Mesh,5
Surface,Mesh,Set,20,1,1,1,3,12,1
Surface,Mesh,4
Surface,Mesh,1
Surface,Mesh,2
Surface,Mesh,3
Surface,Mesh,Set,20,1,1,1,3,3,1
Surface,Mesh,6
Plot,Refresh
Erase,Geometry,All
View,UserDefined,-65,0,15
```