



User Guide

Copyright Notice

The EPS 'LT' logo, this document and the EPS Modelling Studio software are all copyright EPS Logistics Technology Ltd, 152 Staplehurst Road, Sittingbourne, Kent, ME10 1XS, UK. www.epslt.co.uk

'NISA' is a trademark of EMRC, Big Beaver, Troy, Michigan, USA. www.emrc.com

'MYSTRAN' is copyright Dr W Case, www.mystran.com

'Poser' is a trademark of e-frontier America Inc. 5615 Scotts Valley Drive, Suite 210, Scotts Valley, California, USA. www.e-frontier.com

'DAZ| Studio' and 'Bryce' are trademarks of DAZ Productions, Inc. 12637 South 265 West #300, Draper, Utah USA. www.daz3d.com

Contents

- 1. Introduction 3
- 2. Modelling 4
 - 2.1 General Modelling Philosophy..... 4
 - 2.2 General Overview of how the Modelling Studio Works 4
 - 2.3 Modelling Geometry 6
 - 2.4 Modelling Finite Element Entities 9
- 3. How to... Guides 14
 - 3.2.1 How to Open a NISA File 15
 - 3.2.2 How to Use Groups and Layers 16
 - 3.2.3 How to Plan a Model 19
 - 3.2.4 How to Verify a Model 21
 - 3.2.5 How to Input FE Information..... 22
 - 3.2.6 How to Create Laminates 24
 - 3.2.7 How to Use the Load Combiner 27
 - 3.2.8 How to Model for Rendering..... 32
 - 3.2.9 How to Texture Map a Model 33
 - 3.2.10 How to Export a Model for Rendering 43
 - 3.2.11 How to Create a Morph Target..... 45
- 4. File Reference 48
 - 4.1 Files Installed and Used 48
 - 4.2 Modelling Studio Formats..... 50
 - 4.3 EMRC Formats..... 54
 - 4.4 MYSTRAN Formats..... 56
 - 4.5 COSMOSWorks Formats 58
 - 4.6 Wavefront Object Format 59
 - 4.7 Stereo Lithography Format..... 61
- 5. Suggestions..... 63
- A1 Appendix 1 64

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.

2.2 General Overview of how the Modelling Studio works

- 2.2.1 The NISA Tools program 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 NISA Tools child window is a word processor. In the case of the Modelling Studio 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

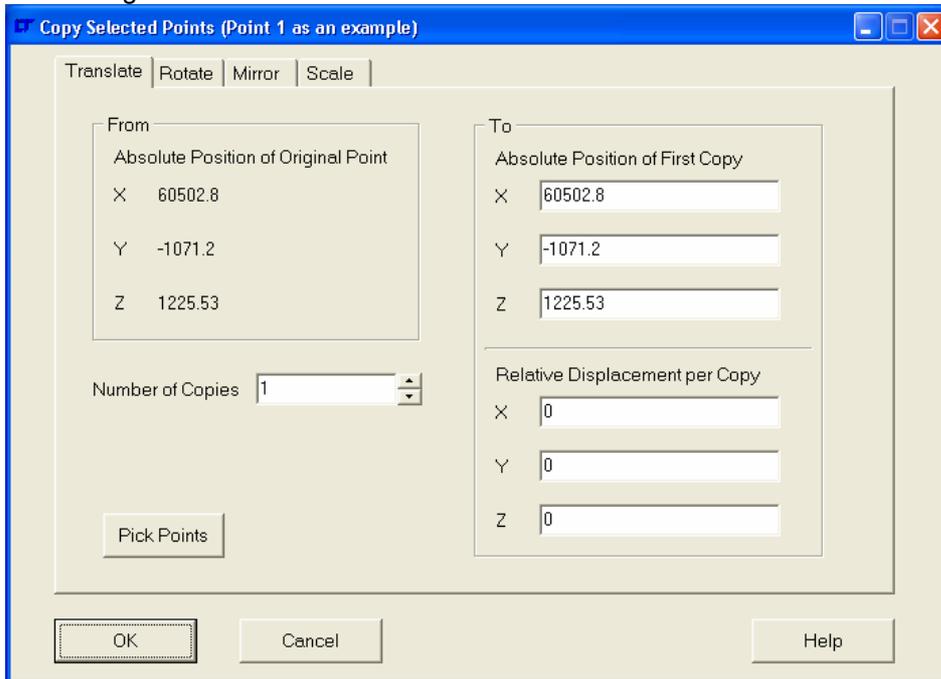
A majority of the information needed to model geometry is covered in the 'Quick Start Guide'. The exceptions are covered below.

2.3.1 Copying Entities

The biggest exception is that very little information is given on how to copy/move entities. The description below concentrates on copying.

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:-



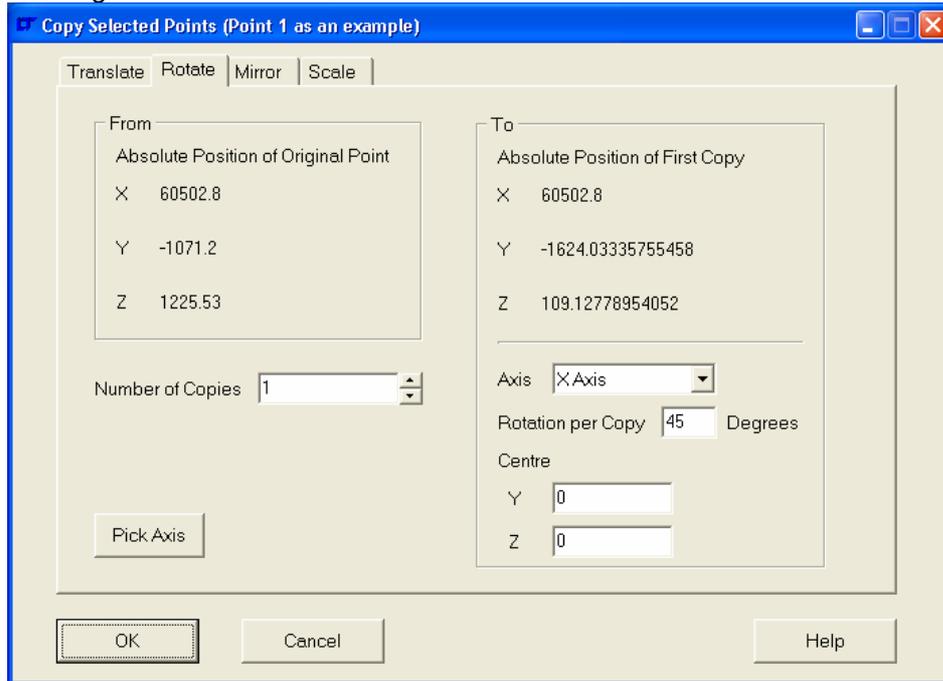
The Translate window always takes an entity key feature as an example. For a point, it is the point location. For a Line, the key feature is the line midpoint, and for a surface it's the surface centroid.

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.

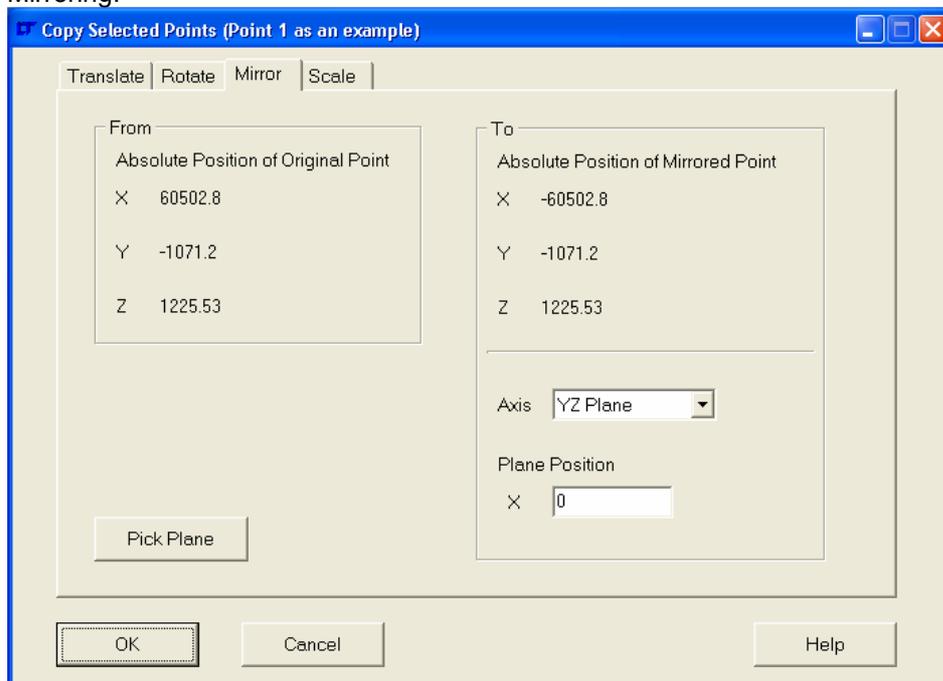
The 'Pick Points' button may be pressed to select a 'Copy From' Point and a 'Copy To' point. This will then be used as the relative displacement per copy. The 'Pick Points' button is equivalent to pressing the 'Copy'  button, but will *only* work on the entities initially chosen. For example, if you got to the translate window by '**Geometry| Points| Copy**', then only points will be copied, whatever else was initially selected.

Rotating:-



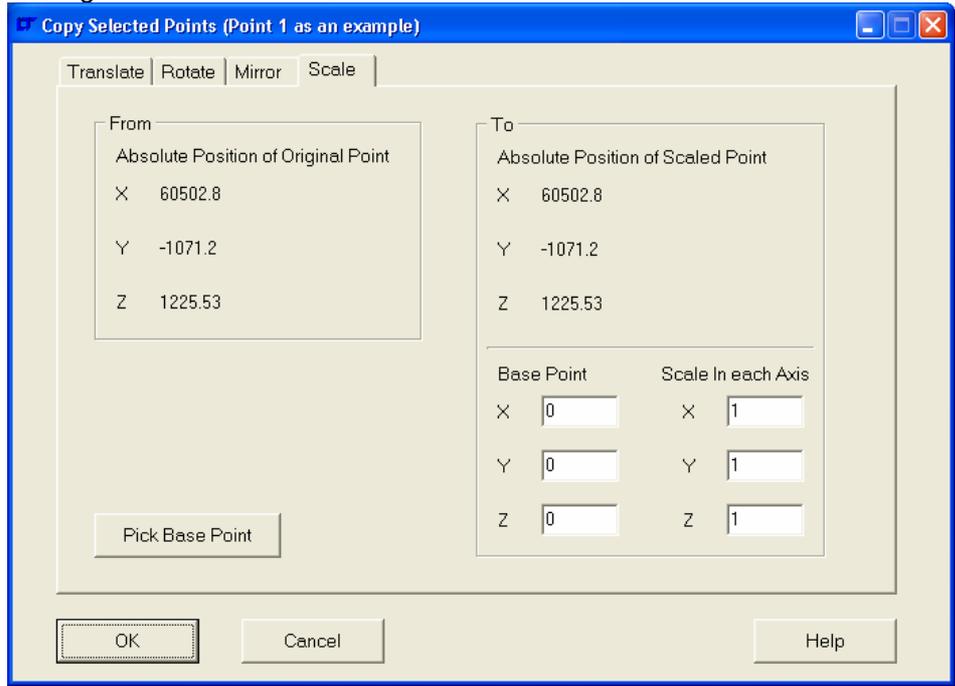
When rotating you specify the 'Number of Copies', the angle of 'Rotation per Copy' and the axis of rotation. Three axes of rotation may be entered on the form, the X, Y or Z axis. These axes may be offset to pass through a point other than the global origin. Alternatively, to specify an axis other than 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 two finite element entities remain to be discussed; Rigid Links and Nodal Pressures.

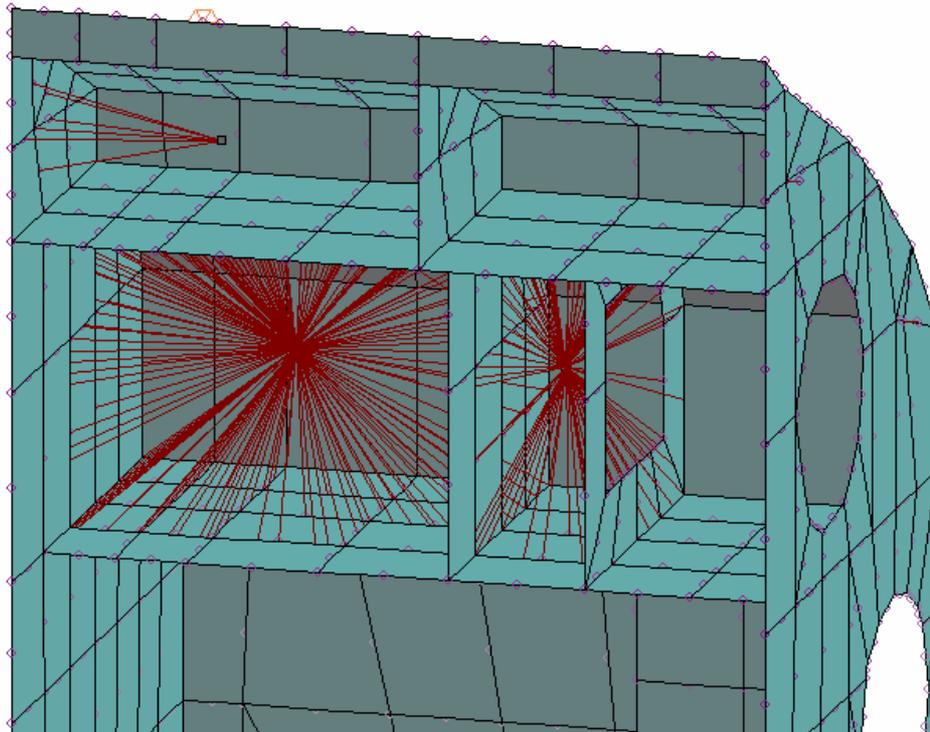
2.4.1 Creating Rigid Links

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

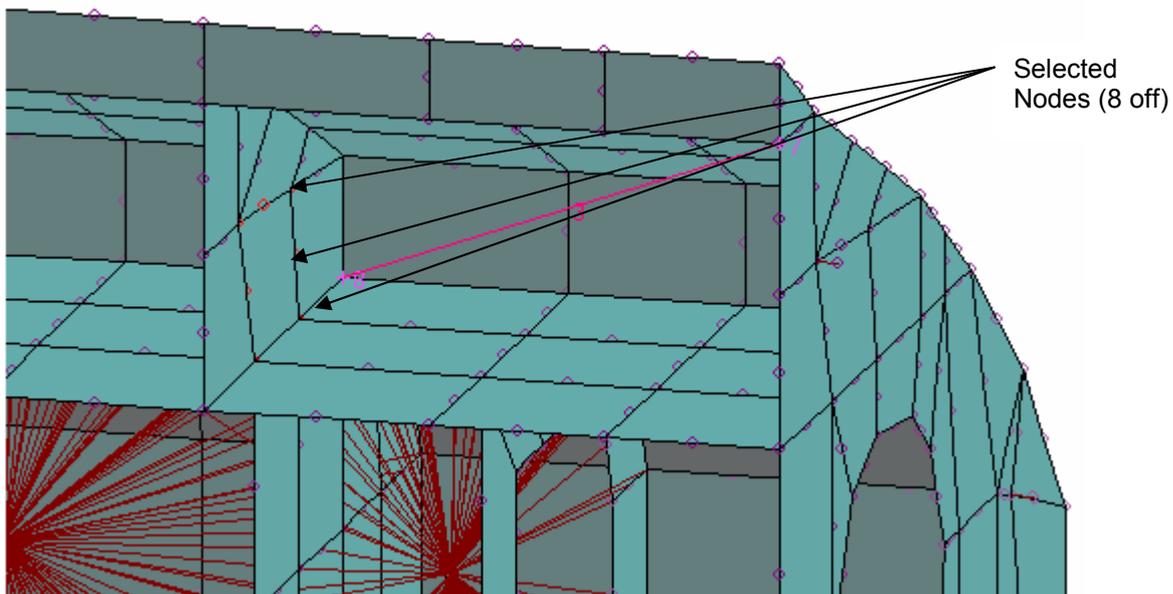
The way to create a Rigid link is described below.

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:-

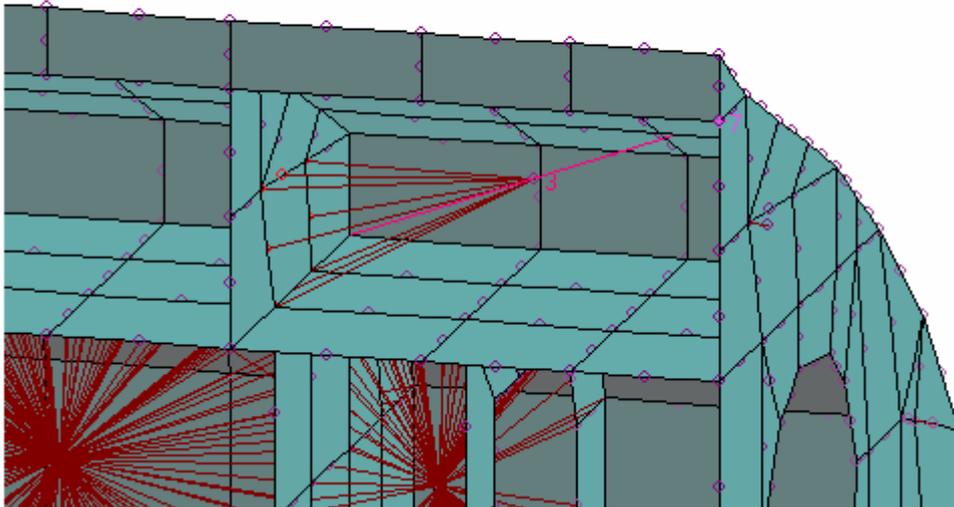


The default settings are all constrained. Set the constraints as you need. Now all future Rigid Links will be created with these slave settings until the settings are changed.

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



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:-



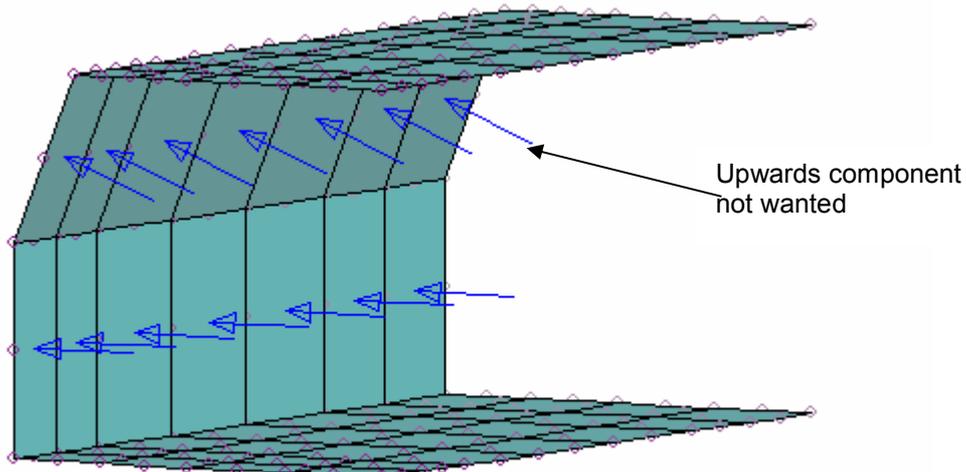
Now a the master node can be loaded to simulate a 'Remote Load', or a Mass element put at the node.

2.4.2 Creating Nodal Pressures

What is a 'Nodal Pressure'?

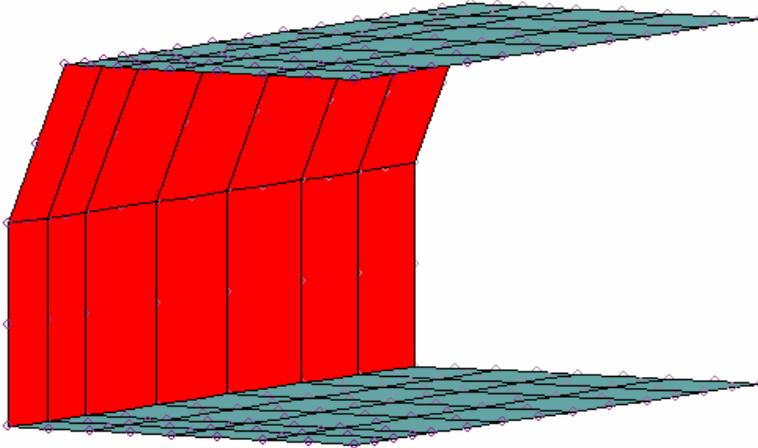
It's a slightly odd, but very useful, idea that must be credited to Mike Hoey at AIM Aviation (Jecco) Ltd in Bournemouth.

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:-

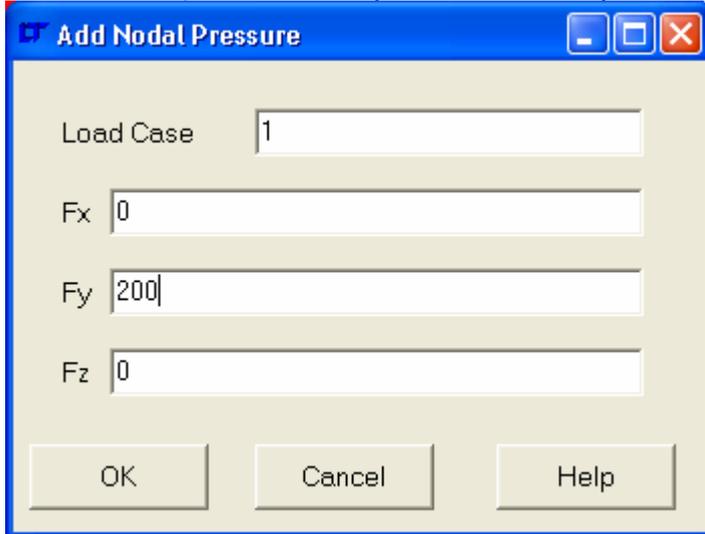


'Nodal Pressures' take the guesswork out of the calculation by applying loads to the nodes around each element in proportion to the projected area of the element in each orthogonal direction. The load is also proportioned in accordance with the element type. Instead of applying a pressure in the above example, a single load is applied. The process is as below:-

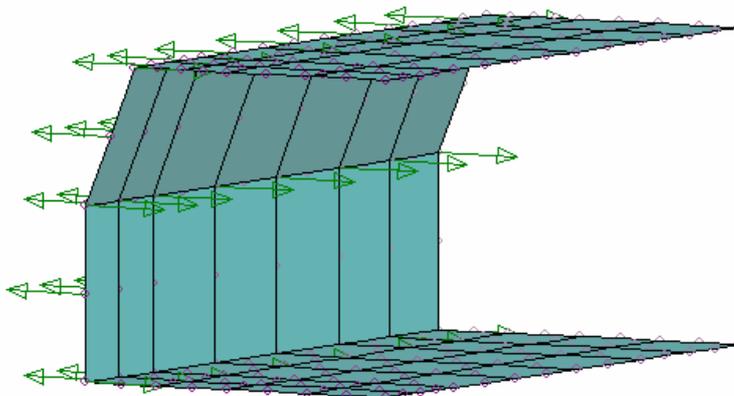
Select the elements..



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	NKTP	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.

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	Open a NISA File.....	15
3.2.2	Use Groups and Layers	16
3.2.3	Plan a Model.....	19
3.2.4	Verify a Model.....	21
3.2.5	Input FE Information	22
3.2.6	Create Laminates	24
3.2.7	Use the Load Combiner	27
3.2.8	Model for Rendering	32
3.2.9	Texture Map a Model.....	33
3.2.10	Export a Model for Rendering	43
3.2.11	Create a Morph Target	45

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

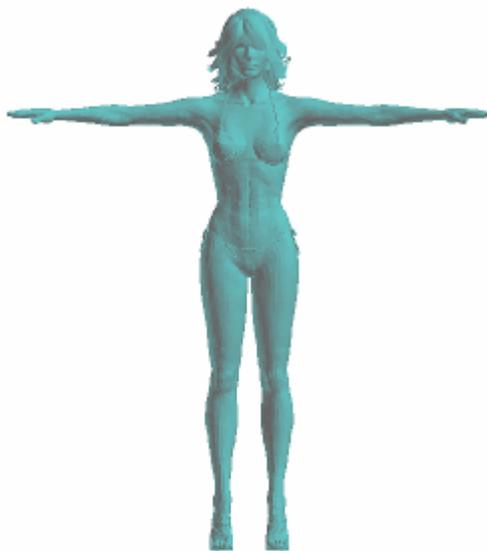
Groups and layers provide a convenient way of aggregating objects that have something in common so that they can be plotted, selected or erased in one easy action.

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'.

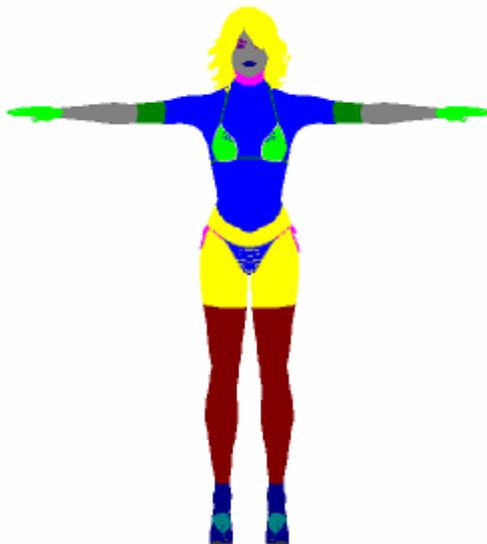
Elements can also be put into a group, which enables a second way of aggregating elements that have something in common. For rendering packages the Modelling Studio 'Group' corresponds to the rendering package 'Group'.

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:-

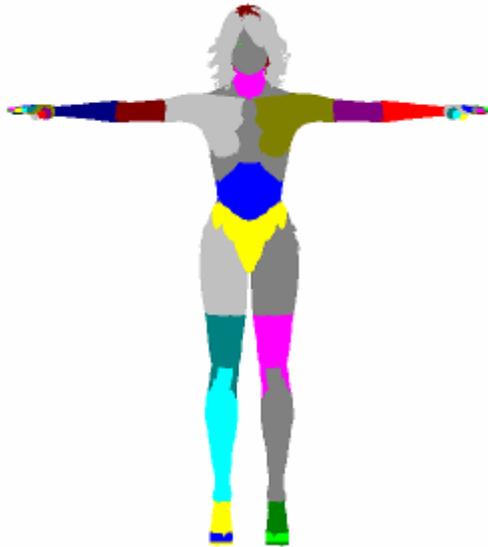
Normal View:-



Coloured by Layer:-



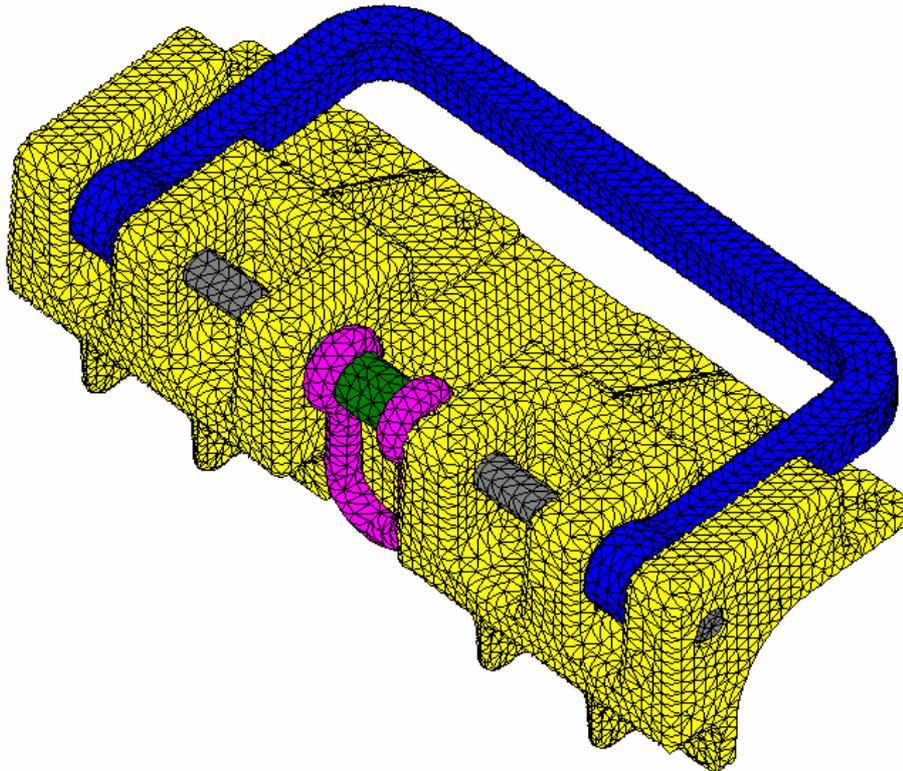
Coloured by Group:-



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:-



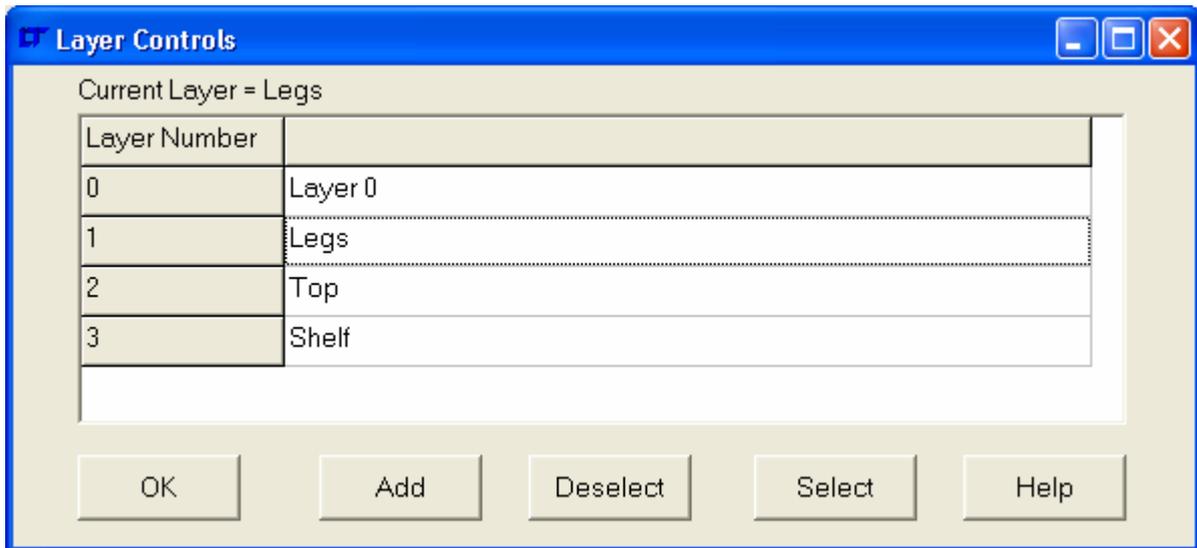
Now, to erase the shackle the only command required is '**View| Erase| Elements| Erase by Group Name**' and select the 'Shackle' group. This saves a great deal of messing about when wanting to view certain parts of the model without the clutter of other parts.

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.

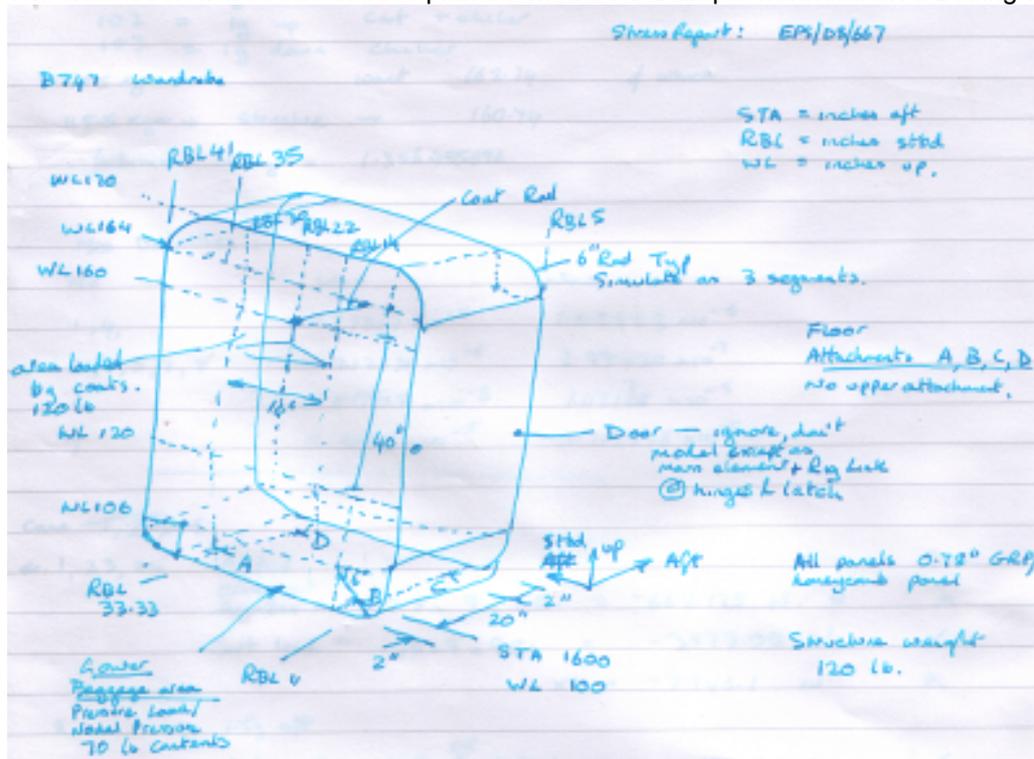
When you start to model, the **last** thing you need is a computer – it certainly **isn't** the *first* thing you need.

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.

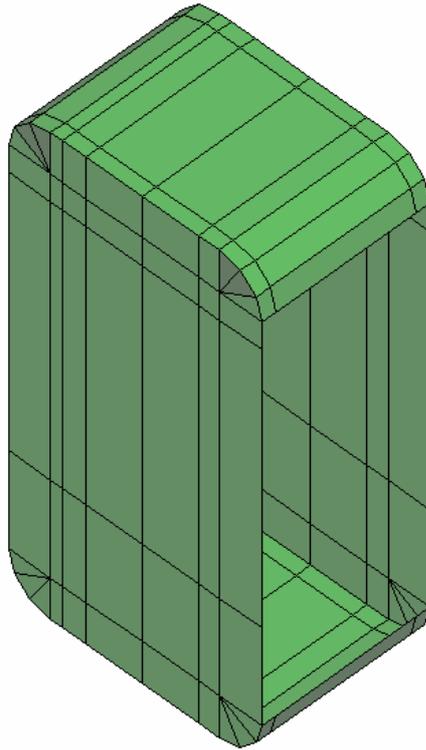
- 1) Get an adequate drawing of the object you wish to model. This may be an engineering drawing or a marked-up sketch.
- 2) Simplify the drawing as much as possible.
- 3) Mark out the key points of the model: edges, attachment points, openings etc.
- 4) Work out the centrelines of any flat panels.
- 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.

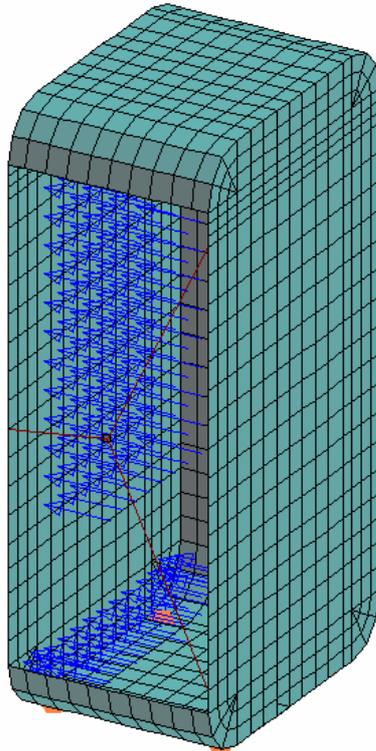
The sketch below was one used to plan the model for a simple wardrobe on a Boeing 747.



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

There are a couple of features which would help verify a model that may be added at some point in the future. These are:-

- 1) Boundary Line Check, to show which edges are not shared between two elements.
- 2) An analysis readiness check to see whether the model is adequately defined for each material, property and load case.

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 only be created by meshing points, lines or surfaces. This may be done using the **Mesh Points**, **Mesh Lines** or **Mesh Surfaces**    tool buttons.

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**', then every new load case referenced by a boundary condition will automatically create a new Load Case.

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'.

Pressures

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'.

Nodal Pressures

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'.

Forces

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'.

Accelerations

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

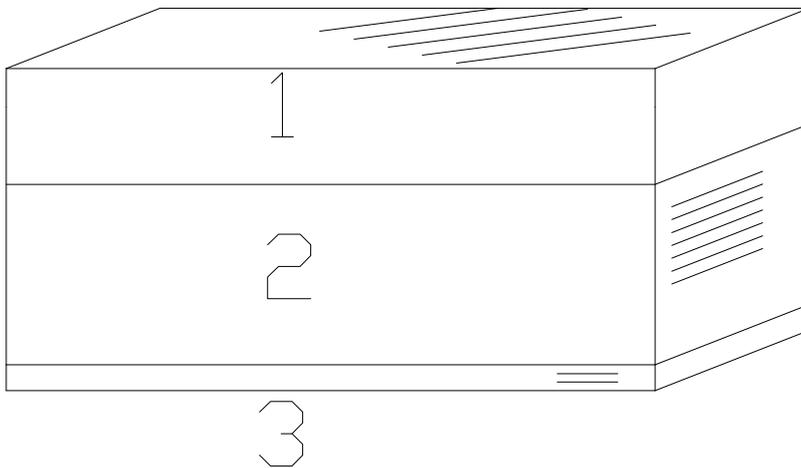
3.2.6 How to Create a Laminate

Laminate Creation is a NISA only function, and is not accessible if the target FE program is set to 'MYSTRAN'.

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.

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:-



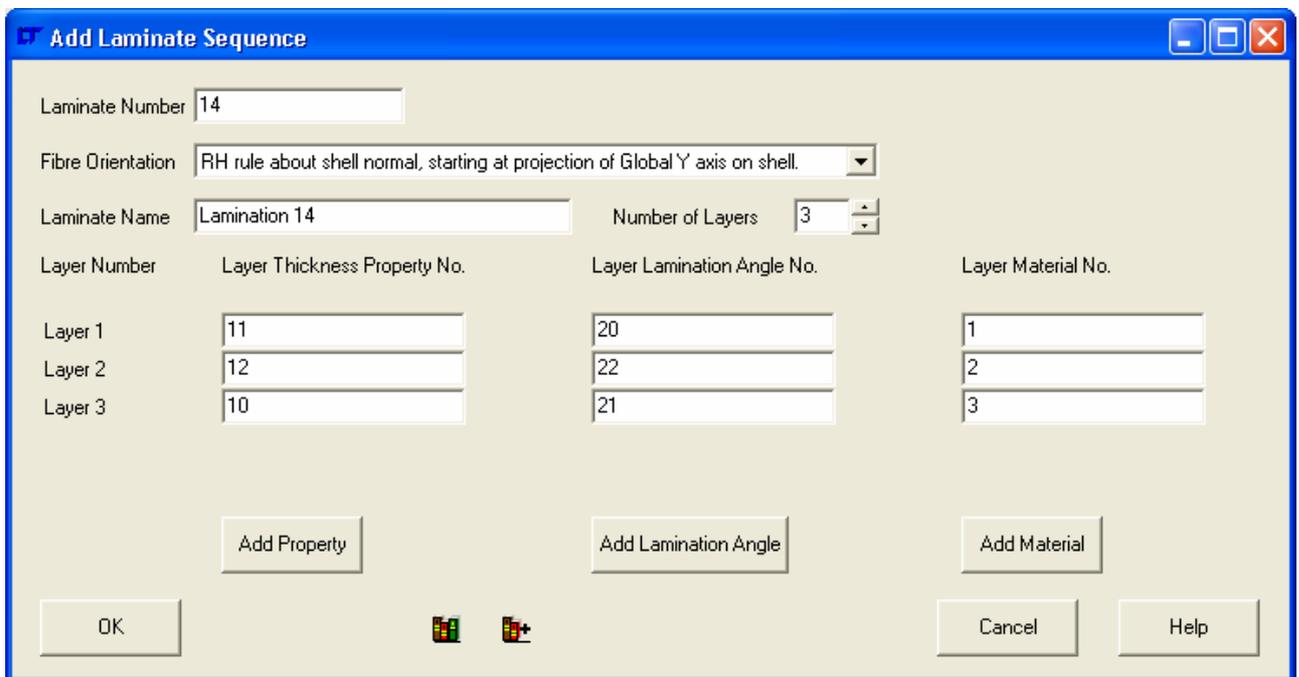
Layer	Thickness	Fibre Angle	Material
1 (top)	2.5mm	-45°	GFRP
2	7.5mm	90°	Nomex Core
3 (bottom)	1mm	0°	Aluminium

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'

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.

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

Fibre Orientation

The 0° axis for the fibre orientation is in the combo box. The 0° axis is in element local axes or global axes.

If any of the following definitions are used, then the element must **not** be exactly in the plane formed by the two axes mentioned or the NISA analysis will abort without any indication partway through the stiffness calculation.

- RH Rule about Global Z axis, starting at global X axis. (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!).

Material

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

Tapered Laminates

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 18 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 identical materials. 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.

3.2.7 How to Use the Load Combiner

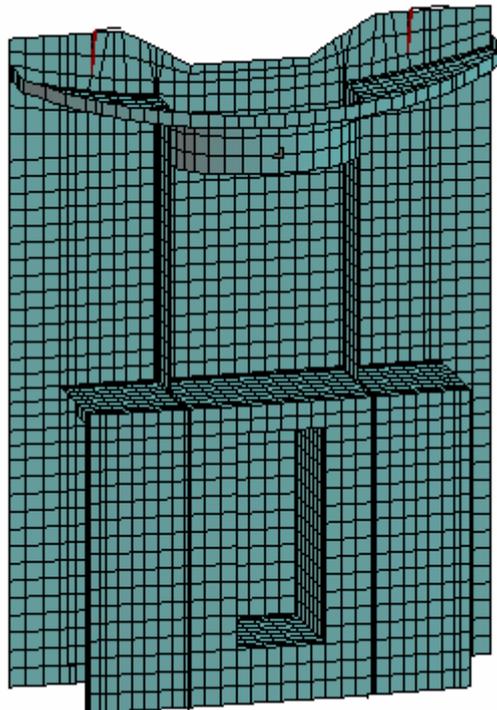
The load combiner is a very powerful tool; however, the best way to use it is not immediately obvious. This section of the ‘How to...’ guide attempts to explain how to best use the tool.

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.

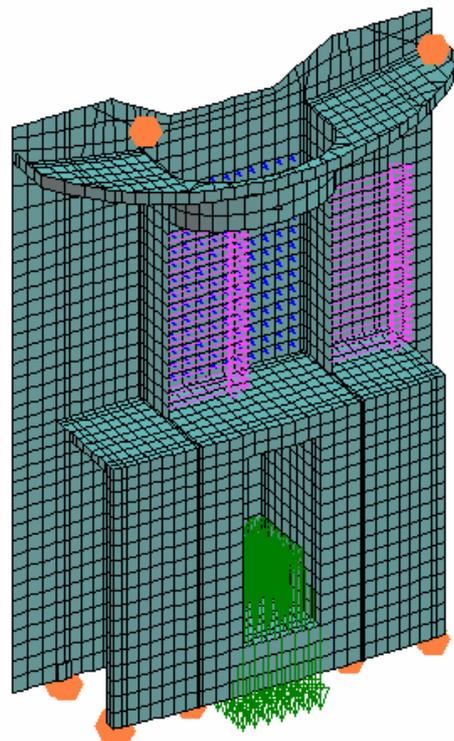
This is one case where automatic load case creation would be a disadvantage, so make sure that menu option '**FE Model| Load Case| Auto Create**' is not ticked.

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.

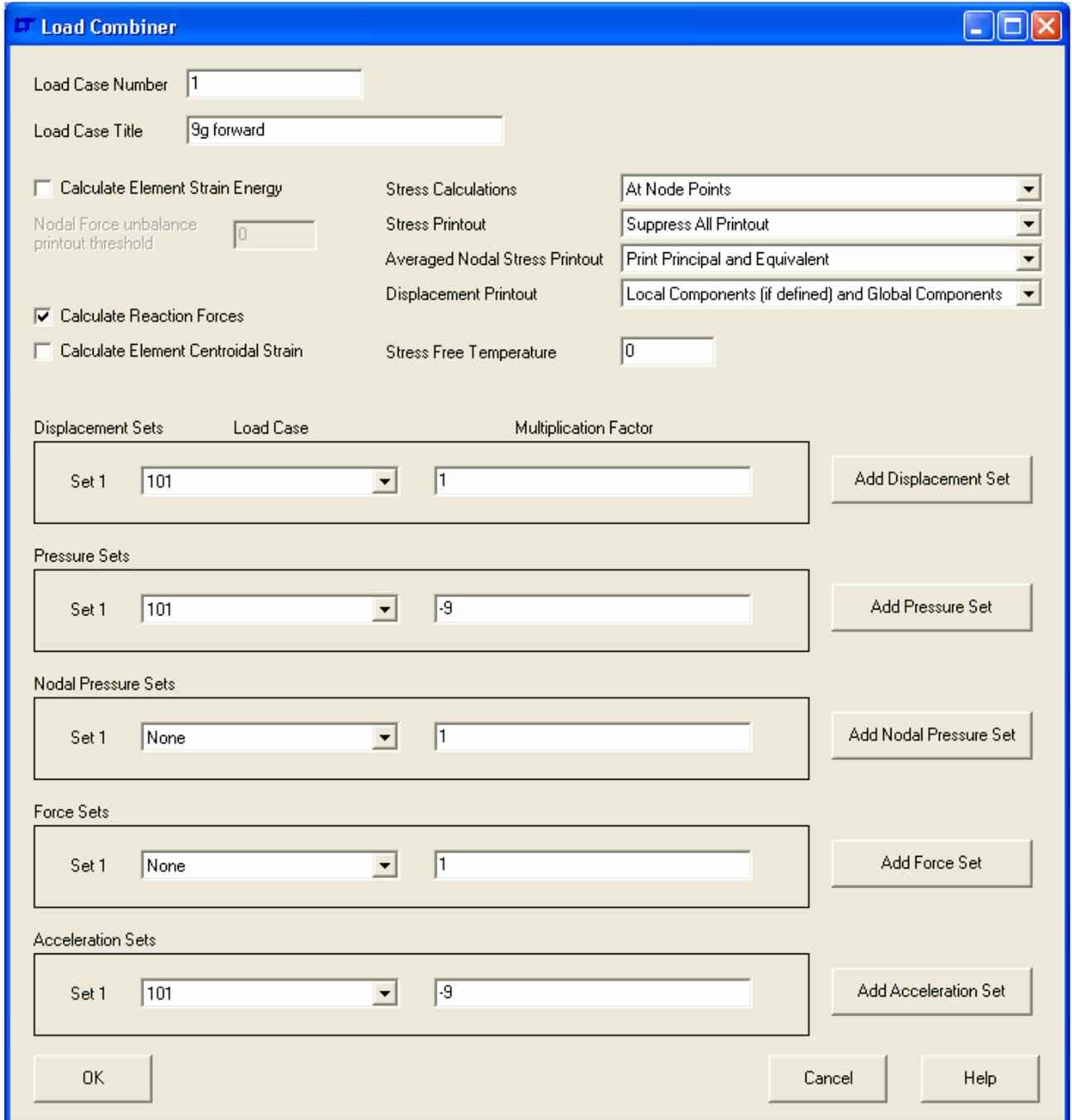
Right... what to do now?

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.

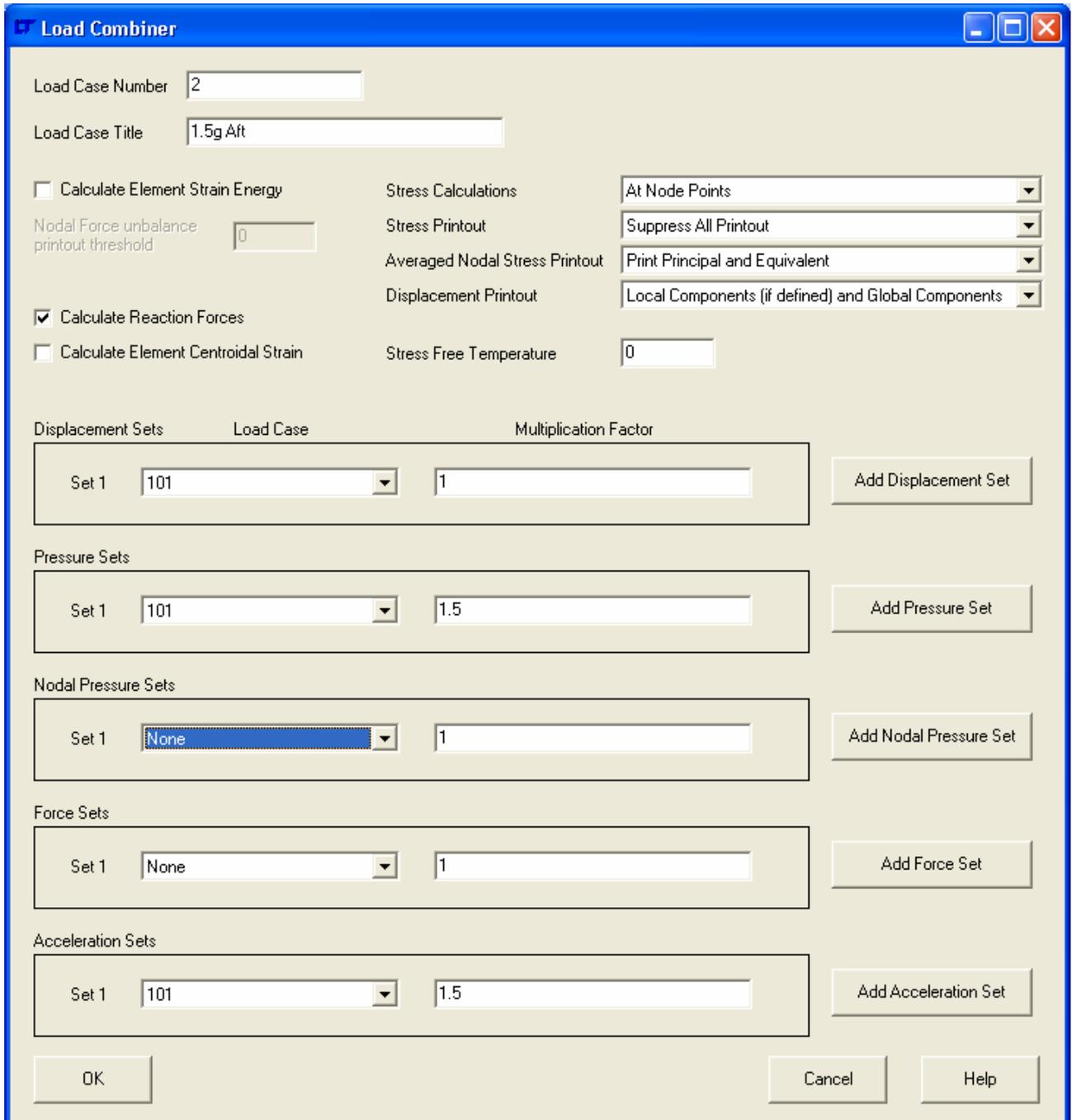


Now use the load combiner to create load case 1. Select menu option 'FE Model| Load Combiner', and make the window look like the one below:-



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.



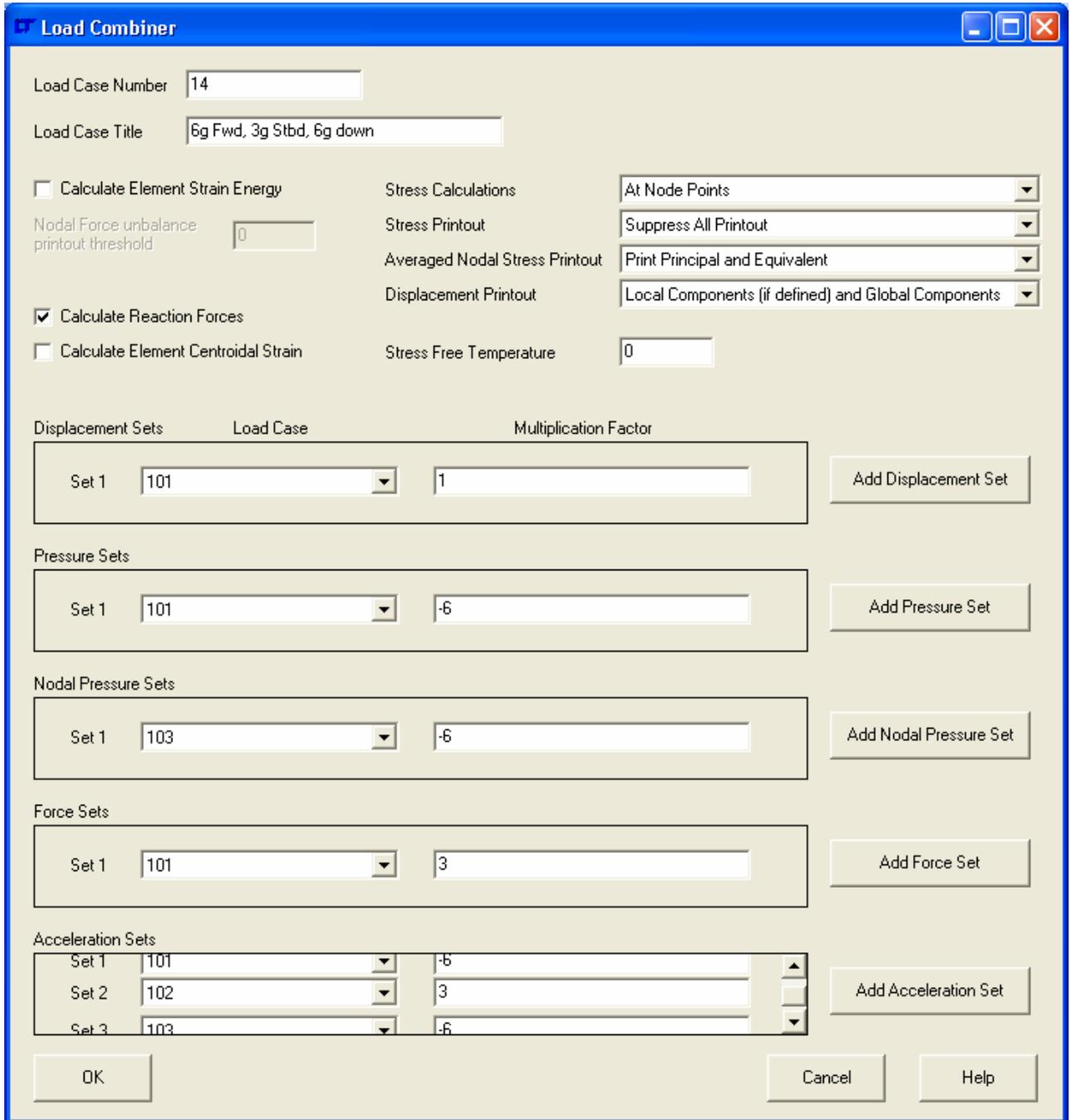
Once again press 'OK' when you're happy. Get the idea?

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 acceleration from set 101, set 102 and set 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.

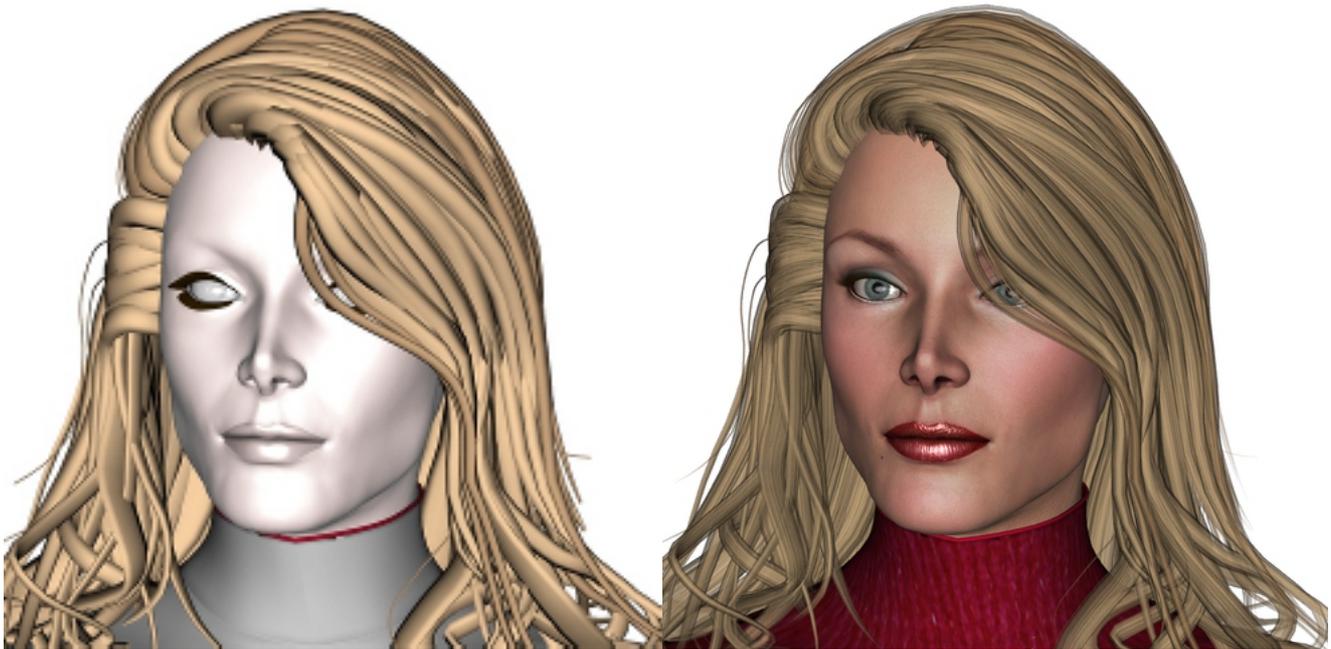
Creation of texture mapped models is a function used for rendering packages, and is not a core FE analysis package function.

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 and other effects can also be simulated. 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.



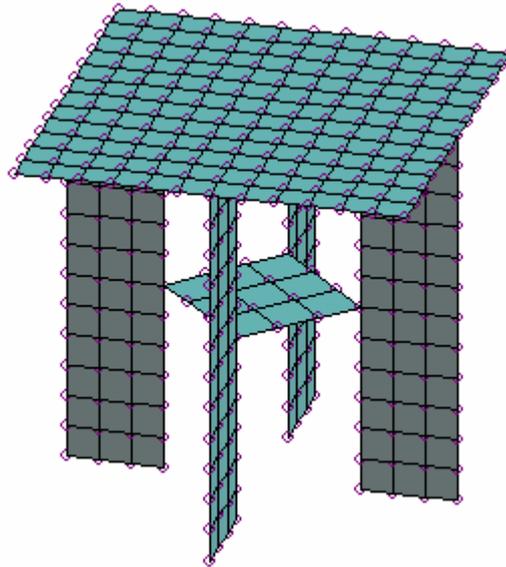
(both pictures rendered in DAZ| Studio)

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 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.

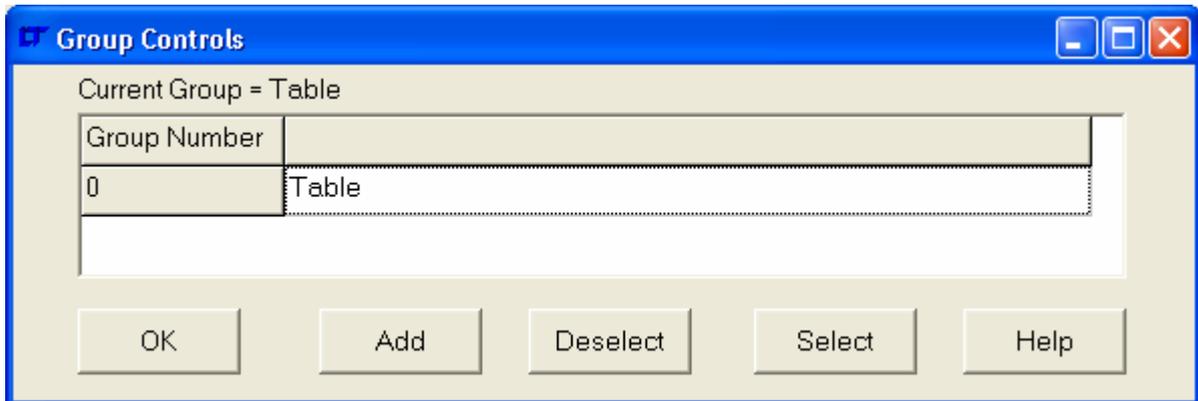


There are 4 legs, a top and a lower shelf.

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

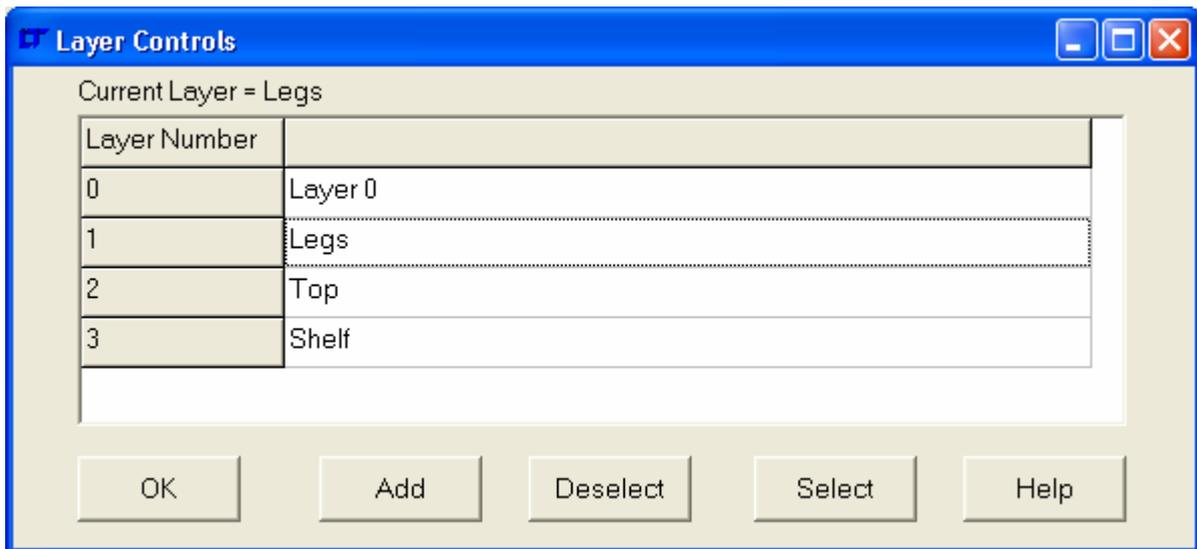
There will only be one group, so it's possible to rename the element group that is created by default. All elements are added to the 'Current Group' as they are created, so the default group will be sufficient. Select menu option '**FE Model| Elements| Groups| Group Controls**'. Click on the default group name and right click over it, to get the 'Rename' pop-up menu item. Rename the group as 'Table'.



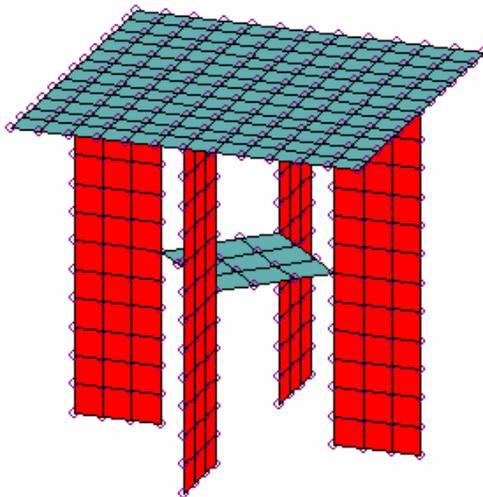
Creating the Layers

We want to create layers because each layer can have its own texture map, element colour, specular 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.

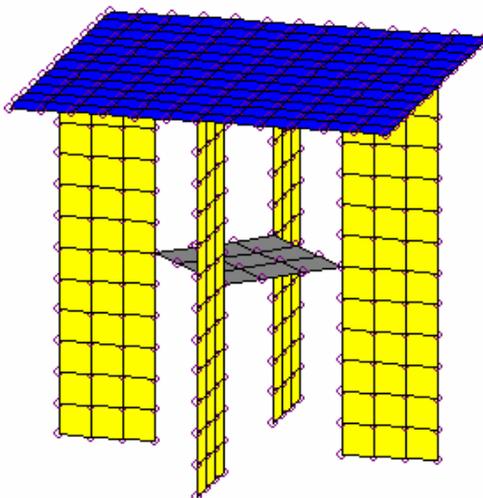
Select Menu Option '**View| Layers| Layer Controls**' and add layers by pressing the 'Add Layer Button'. As each layer is created, click on the default name to rename it, or wait until afterwards, click on the layer to rename, and right-click to get the 'Rename' pop-up menu item. Once finished renaming the layers, double click on the 'Legs' layer to make it the current layer. The final window should look similar to the one below:-



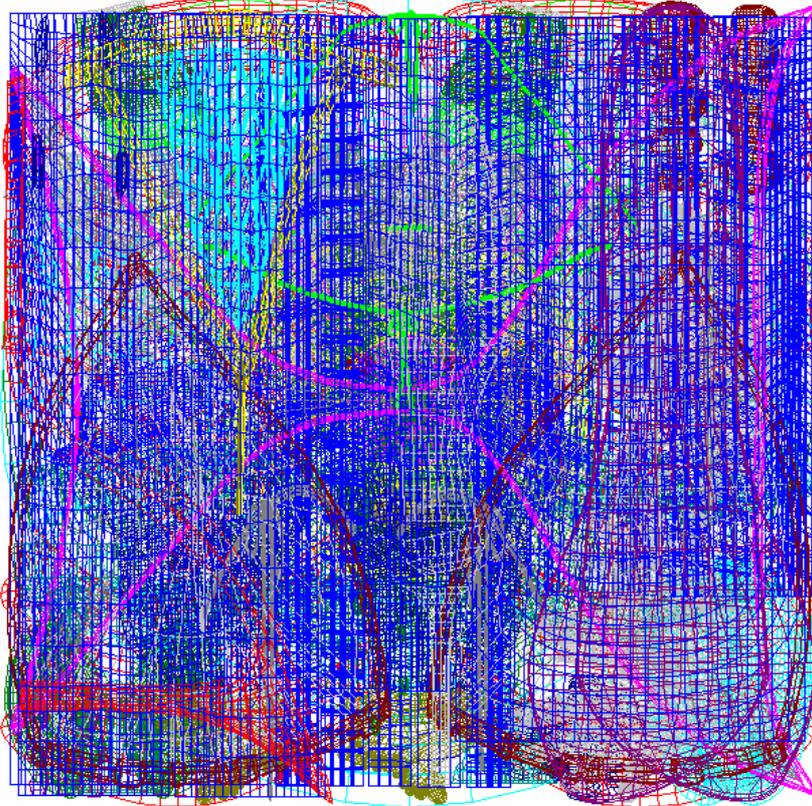
Now, select all the elements making the legs, as below:-



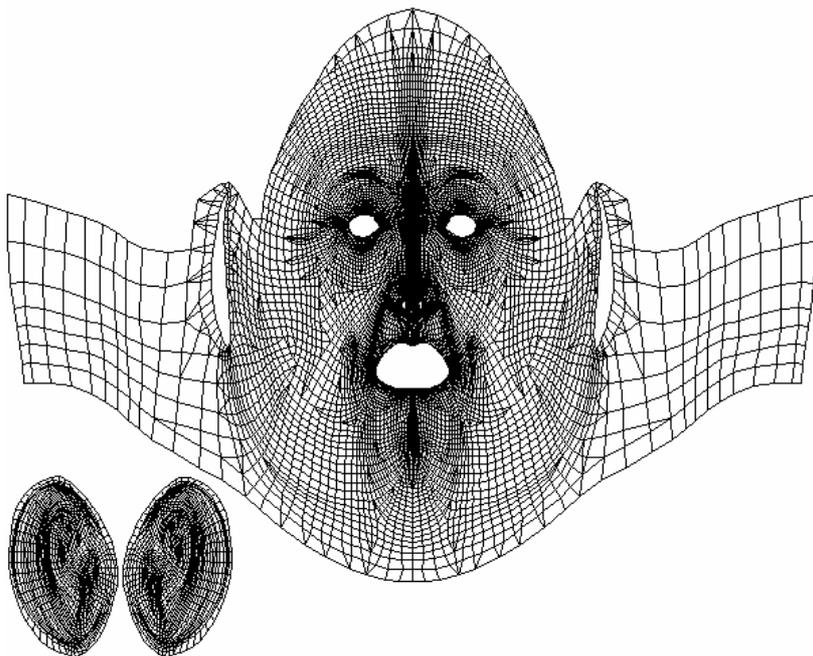
Select menu option '**View| Layers| Add to Current Layer**'. Now select '**View| Layers| Set Current Layer**' and set the shelf layer as the current layer. Add the shelf elements to the layer. Finally set the current layer to 'Top' and add the top elements. When you set the element colour to 'Colour 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:-



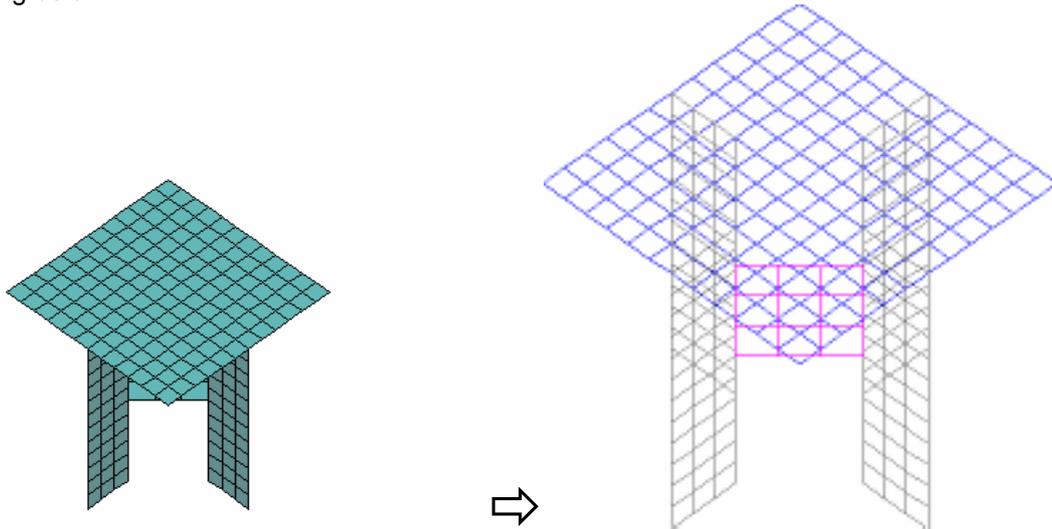
Under all that mess of hair, clothing etc. is the texture template for the 'SkinFace' (honest!):-



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.

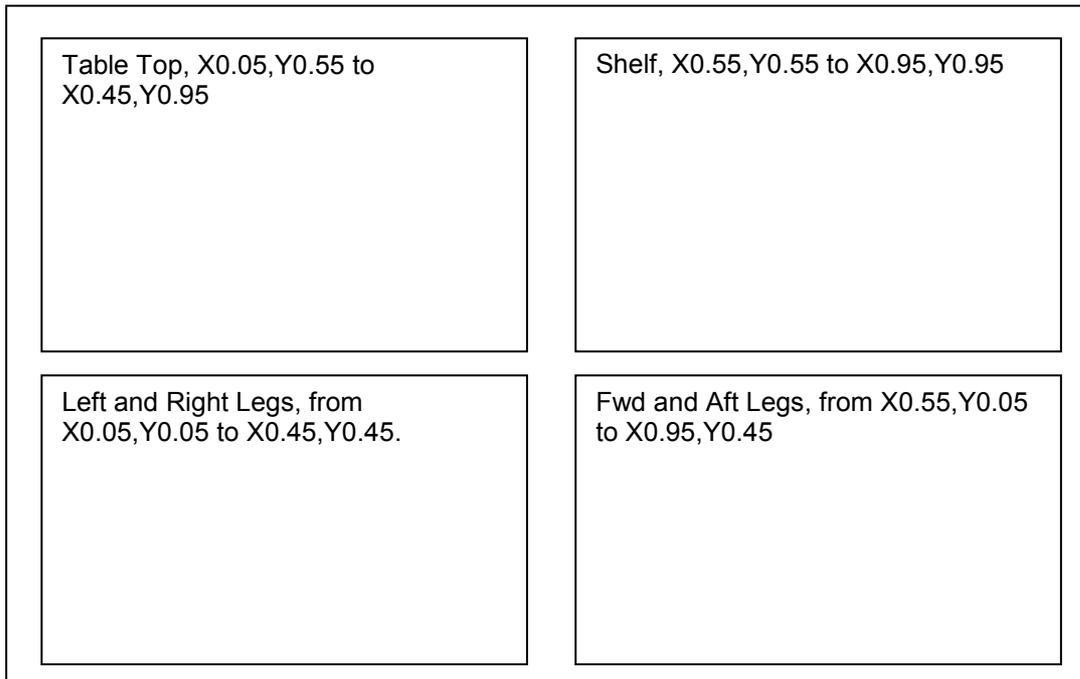
Note:- One thing to remember is that all elements on a layer have to be mapped on to the same texture map, because the rendering package will only open one texture per layer. If this is a problem, just create a new layer for the elements that will not fit on the first texture map and move them to that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

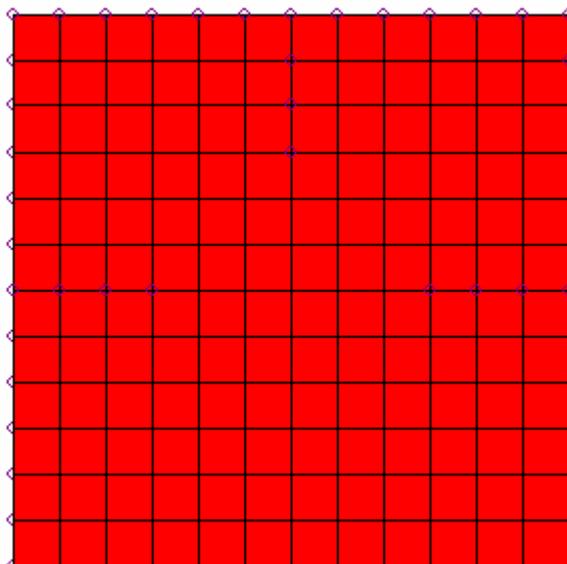
Now to plan where each set of elements are to be mapped. The texture co-ordinates are *always* 0,0 at the bottom left of the picture and 1,1 at the top right of the picture, with X across the picture and Y up the picture. Let's adopt the template layout below:-



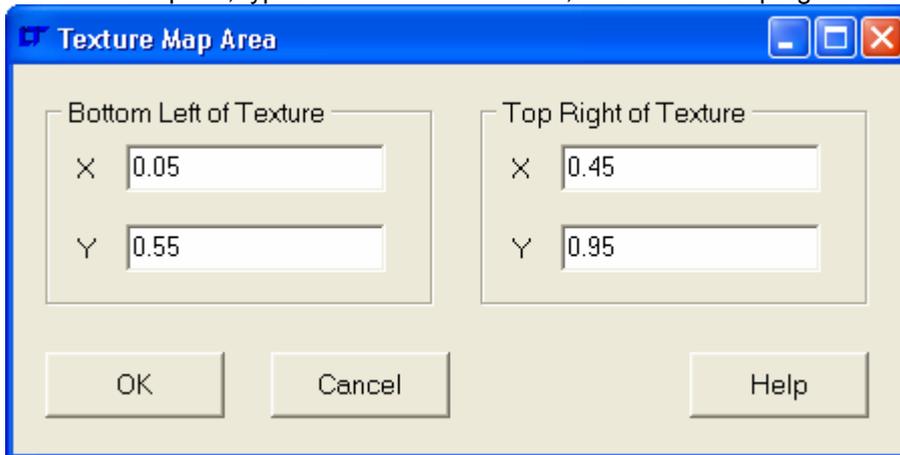
It will be noted that there are margins around each area, and that the scale will not be the same when mapped onto the texture. This really does not matter unless it's imperative to get the best detail possible on a map. If the margins cause problems, change the size of the notional area on the template 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.

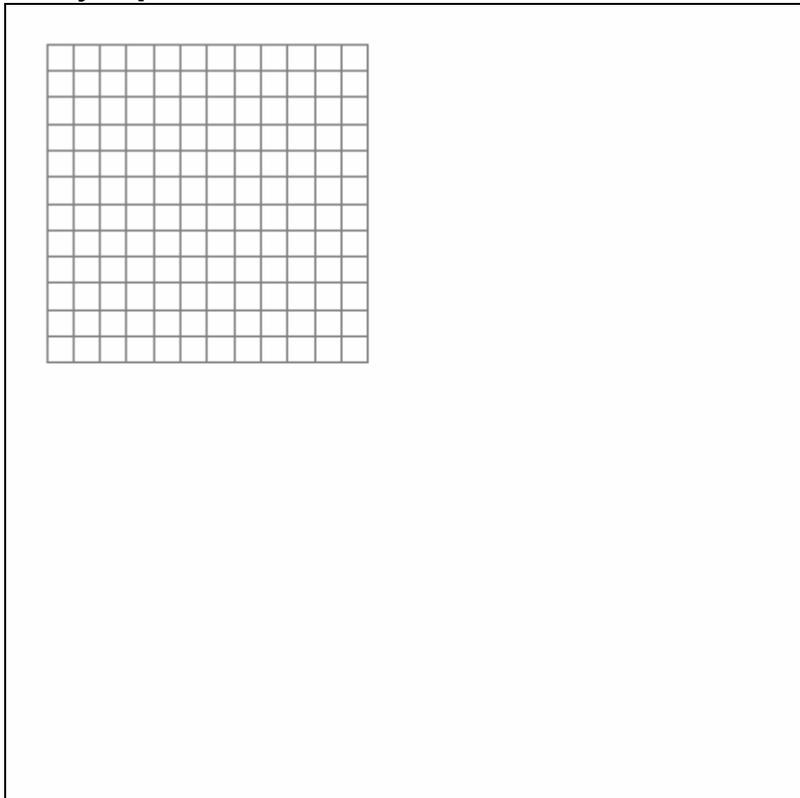
Getting back to doing it right... We cannot map the whole table at once, because the elements are orthogonal to each other and whatever view we take, it's impossible to get a perpendicular view of all faces. If we have an angled view we loose detail in the texture map by smearing the map over a larger 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:-



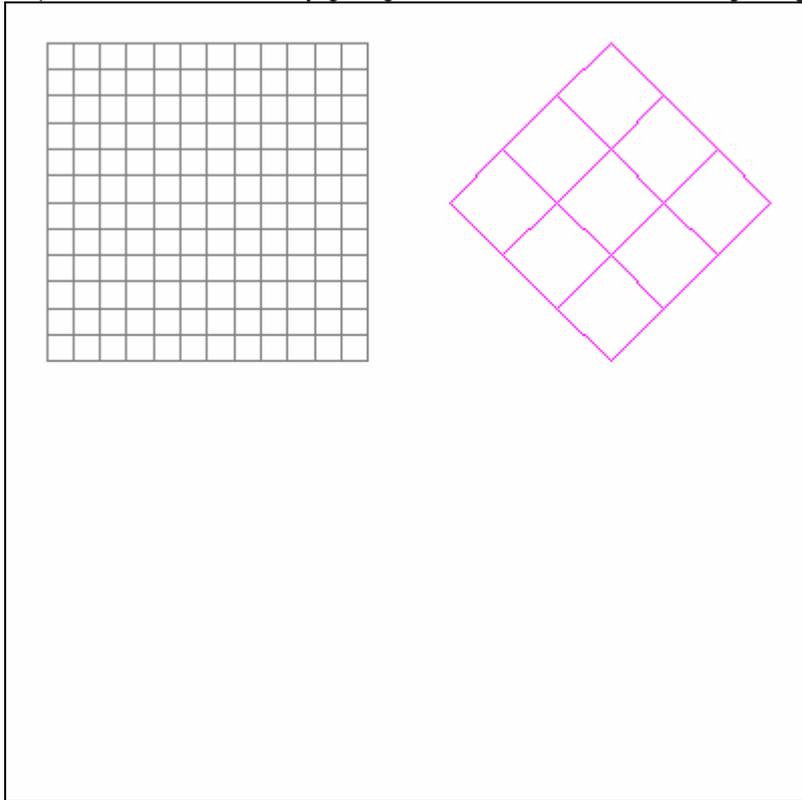
Do the mapping of the top by selecting menu item **'Mapping| Add| Selected'**. In the mapping dialogue window that opens, type the bottom left as 0.05, 0.55 and the top right at 0.45,0.95, as below:-



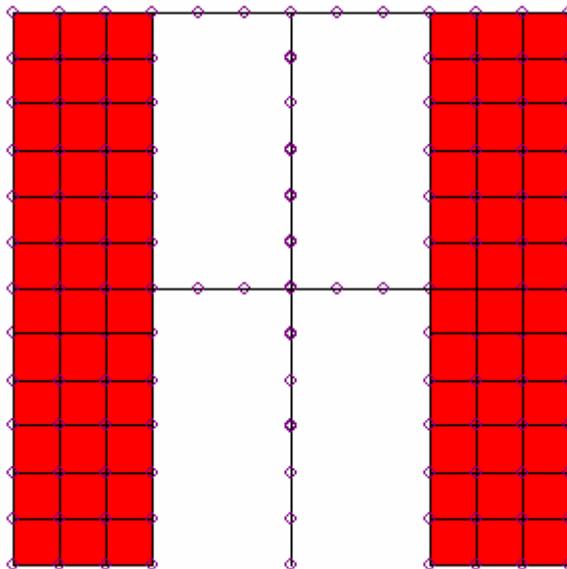
This gives the following texture template to show what we've done [menu item **'Mapping| Templates| All Layers'**]:-



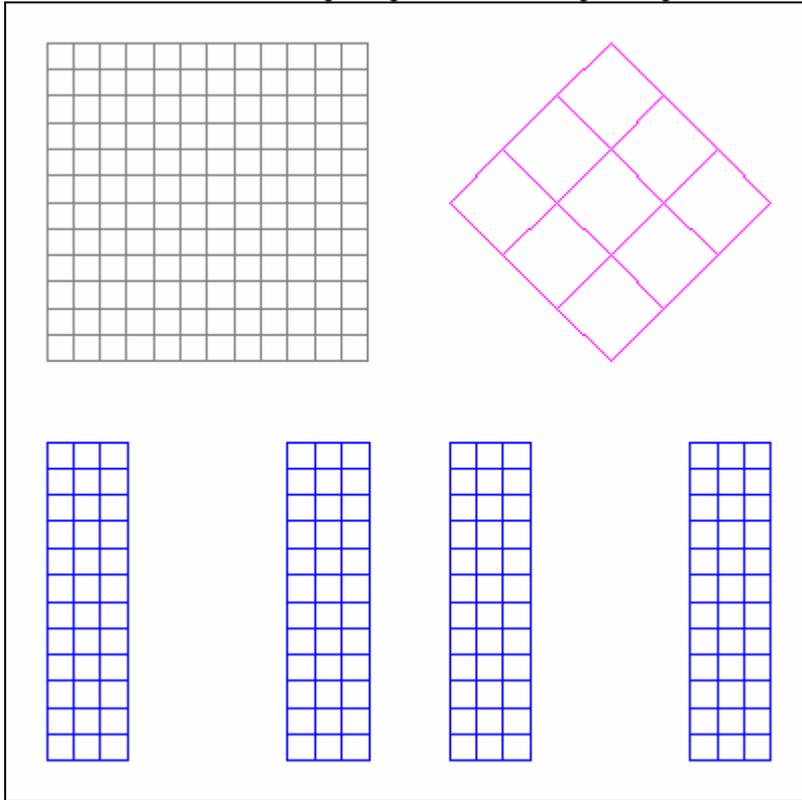
Repeat with the shelf, only giving the new co-ordinates for it, getting the following template:-



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.

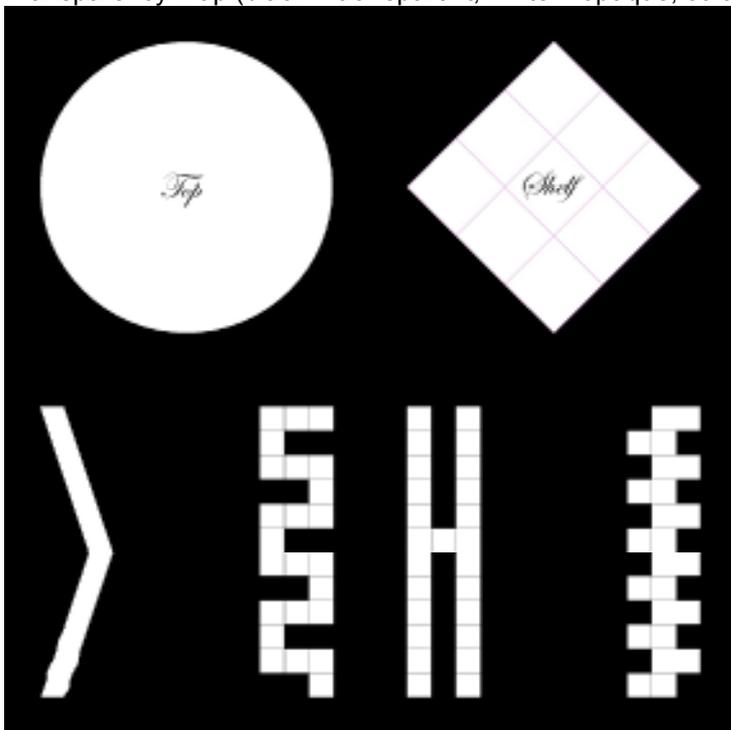


Do the same with the left/right legs after choosing the right view. The final texture template is:-



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!

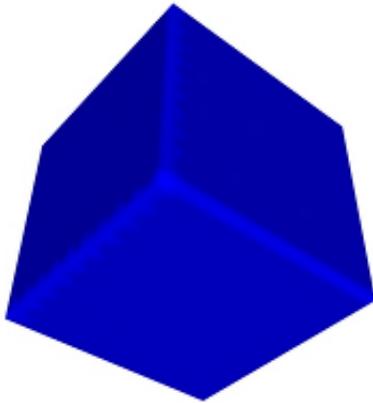
3.2.10 How to Export a Model for Rendering

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

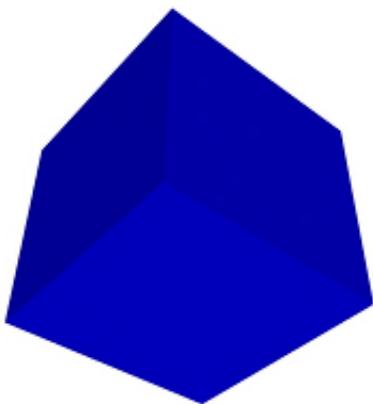
There are a number of causes of unsatisfactory results, each of which has a separate cure.

- 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) Symptom: The model appears to have rotated 90° forward on export.
Cause: Most rendering packages use the Y axis as the vertical axis, rather than the Z axis.
Solution: To resolve the problem, rotate the model by -90° about the X axis prior to exporting it.
- 4) Symptom: The model size/ scale is unexpected.
Cause: The rendering package being used has a default scaling factor on import.
Solution: Find the scale factor assumed by the package and scale the model in the Modelling Studio to match the assumed dimensions.
For example:-
Take a 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:-
$$\text{Factor} = 1/(96 \times 25.4) = 4.101 \times 10^{-3},$$

This gives a figure height of 0.75869 Poser Units, which is correct.
- 5) Symptom: The model in the rendering package is composed of triangles. The Modelling Studio modelled rectangles.
Cause: The Modelling Studio exports parabolic formulation faces as groups of triangles.
Solution: 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.

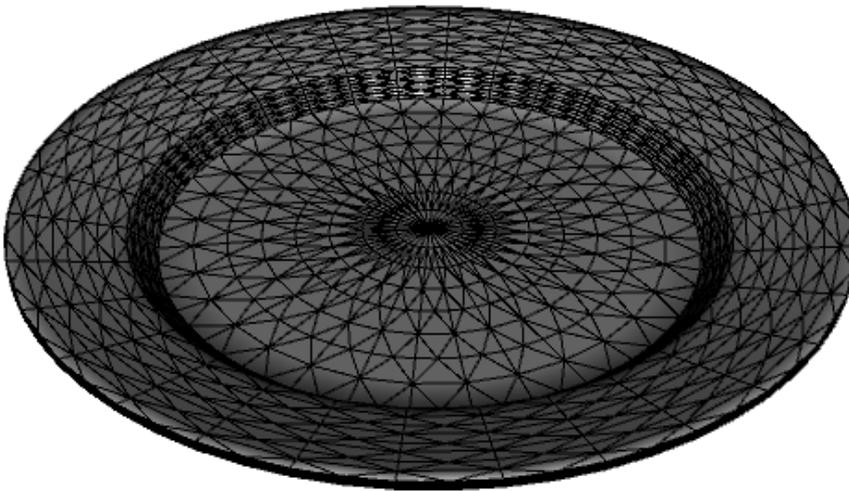
3.2.11 How to Create a Morph Target

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

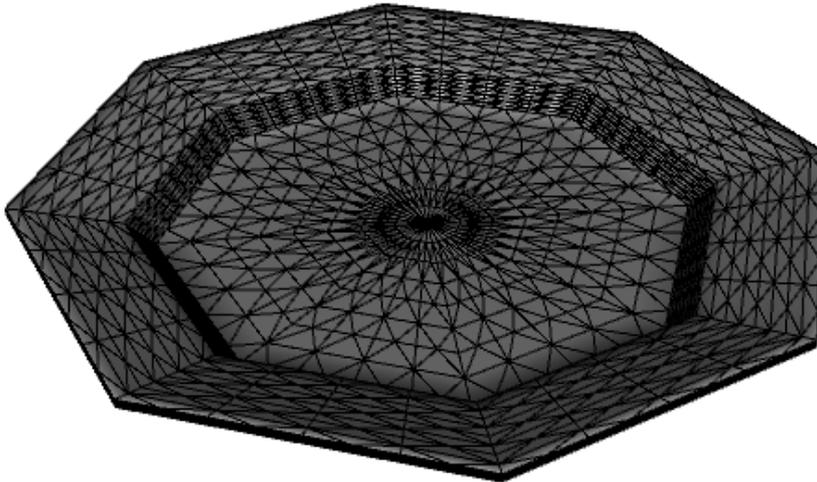
Firstly, what is a morph target?

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

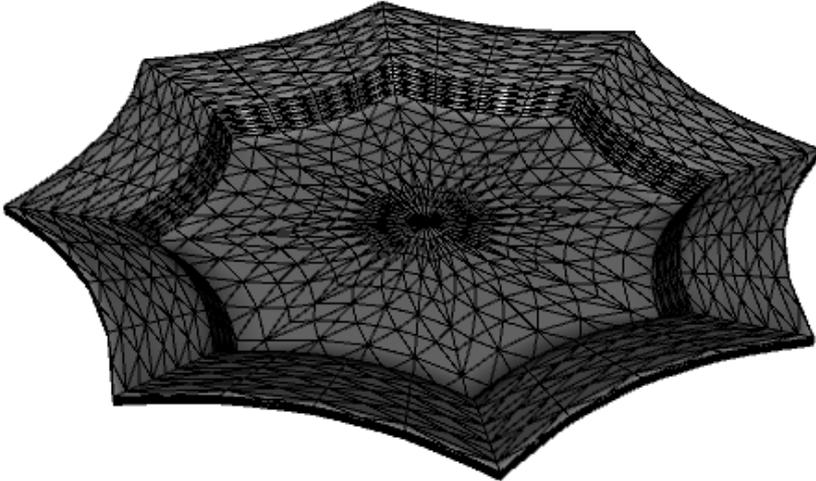


Morph Target Model (100% morph strength):-

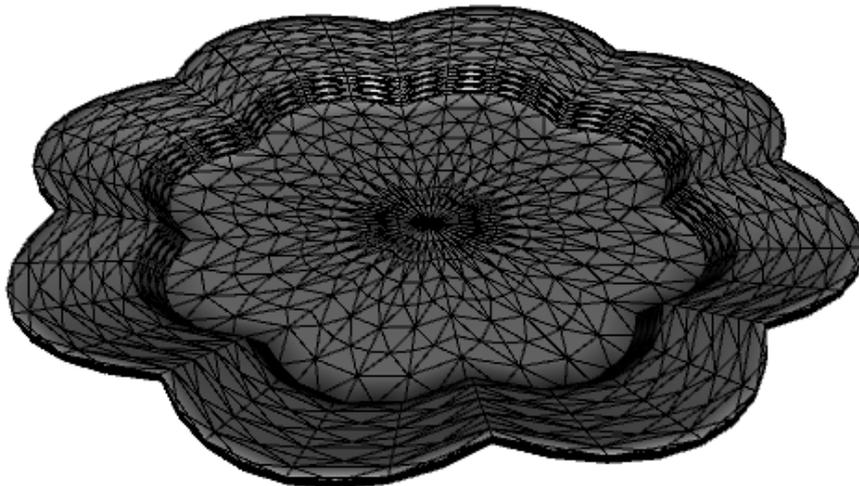


Unexpected, and frequently quite attractive, effects can occur when different morph strengths are used, as below:-

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.

4. File Reference

4.1 General Information

4.1.1 Section 4 is 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:-

<u>File</u>	<u>Location</u>	<u>Description</u>
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

<u>File</u>	<u>Location</u>	<u>Description</u>
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:-

<u>File</u>	<u>Location</u>	<u>Description</u>
Structures.h	<Program Directory>	Structures used by the Modelling Studio.
base.h	<Program Directory>	Word processor ancestor of the Modelling Studio.
base.dfm	<Program Directory>	
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**'.

<u>File</u>	<u>Location</u>	<u>Description</u>
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@epsit.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 (.mif)

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], [PropertyData] or [LamAngleData] section title.

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

One entry per line and one line per entry.

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	
CHEXA	
CPENTA	
CTETRA	
CBAR	
CROD	
CELAS	
CQUAD4	
CTRIA3	
CONM2	
RBE2	
GRID	
PBAR	
PSHELL	
PROD	
PSOLID	
PELAS	
SPC	
FORCE	
PLOAD2	
GRAV	
RFORCE	
MAT1	
MAT8	
PARAM ELFORCEN LOCAL.	Fixed output.
PARAM GRIDSEQ BANDIT	Fixed output.
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 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 does not support laminates, so any laminated elements (Types 32 or 33) export as shell elements of type CQUAD4 or CTRIA3, and the material used is specified by the Element Material [which properly refers to the laminate definition number]. No check is made to ensure that the correct material definition exists – however a warning reminder is given in the '**Progress Information**' box.
- 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 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).

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:-

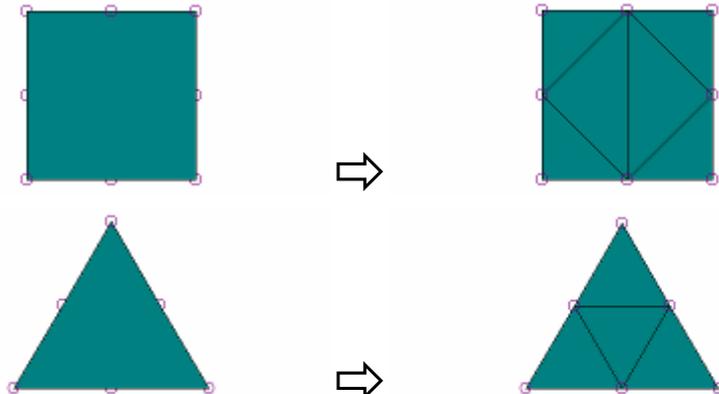
v	Vertex. Imported as a Node.
vt	Texture Vertex. Imported as a Texture Node. 2 dimensional texture maps only are catered for.
g	Group. Set current element Group.
usemtl	Use Material. Sets current Modelling Studio layer and, if the .mtl file exists, the element colour.
f	Facet. Imported as a shell element.

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, 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 confuse the issue - the Object 'Material' has nothing to do with a FE Model 'Material'. 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 opened or imported by the Modelling Studio. Both ASCII and Binary format files are supported.

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. However, when a STL model is imported as elements and nodes, the new nodes are not merged with the existing model. 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.

4.8 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 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.

5. 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.msfl' somewhere, and then close the text editor. Press menu button 'File' and select the 'Table.msfl' 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