**SIEMENS**

**GEOMETRIC SOLUTIONS**

# Femap 11.0 new features

## White Paper

Written for Siemens PLM Software by Tony Abbey, FEA Specialist.

**www.siemens.com/plm**

# Contents

# Femap 11.0 overview

Femap™ 11.0 software is now here and this paper takes a very close look at the new functionality rolled out. First off, a confession is needed. Femap has been one of my favorite tools for many years. A significant part of my professional life has been spent using the program to make a living! When it is that critical then you tend to be very demanding and not accept second best.

So what is my initial reaction to the new release? Well I think it is a winner – it develops further the rich vein of new tools that has been introduced over the last major upgrades. In short the new postprocessing options and in particular the completely revamped Chart toolbox would form the basis for a strong release. However there is much more than that in Femap 11.0.

There are several significant big ticket items such as Model Graphics performance and Externalization of results which keep Femap in the forefront as Finite Element Model size and data output quantities grow at an insatiable rate. It is no good having a very slick User Interface if it stalls when a one million element model is thrown at it! The Femap team within Siemens PLM Software is doing a great job of future proofing the product just when many in the FE industry are concerned that we are about to see a major data explosion as FE solver power continues to increase. The days of 'keeping it simple' seem to be behind us when it comes to FE model creation!

The Femap team is also doing a great job of bringing advanced analysis features into a typical engineer's toolkit by use of more friendly and straightforward User Interfaces. Examples in this release include External Superelement and Bolt Preload support. You may never have heard of External Superelements, or if you have, been daunted by the sheer complexity of the data setup. Well read on, the technique is simple in concept and the Femap 11.0 interface is very elegant and easy to use. You may become a convert! Similarly Bolt Preload is essential for correct modeling in many cases and again the interface is very easy to use.

I really like this Femap trend, complex data set up reduced to a simple process so that we can focus on understanding the engineering assumptions and explore the concepts. I am not advocating using the tools blindly; the reduced learning curve should allow Femap to be used as an aid to understanding.

There are many other analysis based enhancements in Femap 11.0, including additional Composites and Optimization support. A wide range of specialized applications such as variable acceleration fields and non-structural mass regions is also included. Now these may not be relevant to all, but just reflect on the way these have been implemented in Femap. They are not just 'bolted' in; they use what is now a rich set of ergonomic interface paradigms to make sure there is a consistent and elegant setup process.

Femap has a strong pedigree in mixing geometry based and free form meshing. This is one of the aspects of the program I find compelling as I often work with fabricated structures with solid CAD geometry unable to produce an easy meshing platform. It is great to see another important tool emerging here in the ability to create surrogate or local surface geometry from bare mesh using the Surface from Mesh feature. There is also a great new direct mesh editing tool, Mesh Edge Split. This allows a very fast introduction of mesh refinement, propagating the split across a defined zone of the model. I look at this and anticipate how much time it will save in future models.

Throughout this review you will find constant reference to the many new and enhanced aspects of the Model Info tree, the Entity Selection dialog box, the Toolboxes and so on. Over the last major releases these have been developed and honed into an extremely productive environment that allows the pre- and postprocessing job to be done powerfully and efficiently. Femap 11.0 continues this theme. I am sure I have not found all the subtle improvements yet and indeed they are often so seamless that they may not be obvious!

Read on and find a more detailed breakdown of the features in the review. I hope it whets your appetite and that like me you are looking forward to using Femap 11.0 in your next FEA project.

# External superelement support

Femap 11.0 has made a very important advance in opening up the use of Superelement technology in many applications through a very well thought out Graphical User Interface. This now means that a very powerful FEA technique has been 'tamed' and should be a useful analysis option to many engineers.

Superelement technology has been around for a long time in the Finite Element Analysis world. However it has been largely restricted to highly specialized analysts working on sophisticated assemblies of large system models such as aircraft, launch vehicles and the like. For the general user the advantages of the technique have remained somewhat moot as the technique was not simple to implement. In addition more powerful computing and software have made the original basic motivation of overcoming model size limitations less important.

In essence External Superelement technology allows an engineer to make a Finite Element Model of a component of a large system, such as a winglet attached to a full wing. The component model is translated from its usual 'physical' format of nodes, elements, properties, loads etc. to a set of structural matrices that represent the same information. Common boundary nodes exist between the component and the main system so that the component matrices can be attached to the main system. The complete system can then be solved and the results passed back to the component. Figure 1 shows the overall concept.

The advantages of the External Superelement approach include the fact that the company designing the winglet does not need to pass any detailed information to the wing manufacturer, only the winglet matrices. The wing manufacturer is able to run an overall analysis, including the influence of the winglet. The winglet designer can see the full response of his model when attached to the wing. Confidentiality is maintained between partner companies.

It can be an advantage to reduce model size for advanced analyses like nonlinear and dynamics for both performance and housekeeping reasons. For a large system with many components it can be very convenient to split up a model between parallel project teams. For example different components may have reached different design maturity, but as long as the interfaces are defined then a full system model can still evolve with the latest component designs.

The engineer still has to understand the basic ideas behind the technology to avoid misusing it. However the setup is now greatly simplified with Femap 11.0 and the learning curve is now less steep with the user able to focus in the results of the method rather than the complex data format.
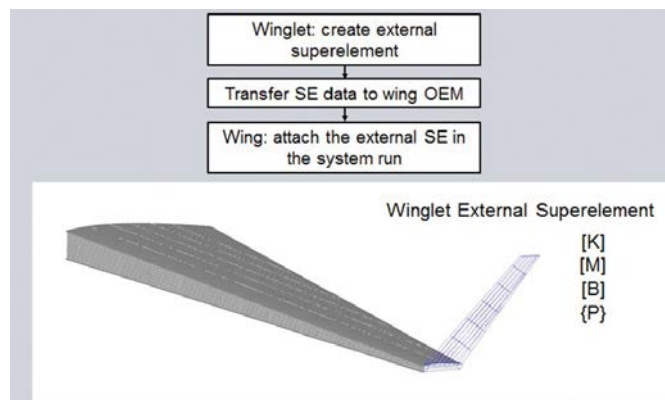


*Figure 1: Winglet external superelement.*

# Model performance

Finite Element Model sizes continue to grow inexorably with improvements in hardware and software power. The maximum feasible size of a model could reach the point where it is impractical to handle such a model in a traditional GUI environment. The Femap developers have this very much in mind with consequent performance improvements in Femap 11.0 Graphics architecture.

Users want to be able to zoom, spin and rotate a model in a preprocessor to enable fast visualization of complex geometry, mesh and analysis attributes. Technologies that are being harnessed to permit this include both hardware and software based approaches such as Graphics Processors, including parallel Processors, Graphics RAM, Open GL improvements and Vertex Buffer Objects.

Vertex Buffer Objects (VBOs) are Buffer Objects that are used for vertex data. Since the storage for buffer objects is allocated by OpenGL, vertex buffer objects are a mechanism for storing vertex data in "fast" memory (i.e. video RAM or AGP RAM, and in the case of PCI Express, video RAM or RAM), thereby allowing for significant increases in vertex throughput between the application and the GPU.

The Femap development team use large test models including an array of Boeing ISS Laboratory Module 'components'. It is important to use real models with their inherent complexity rather than arbitrary blocks of elements. Figure 2 shows a single model and the full array used in testing. The model has 4.5 million nodes and elements together with a large number of physical and material properties. A speed up of eight times was found in dynamic rotation performance between Femap 11.0 and 10.3 on a mid-range laptop with 16 Gb of RAM.
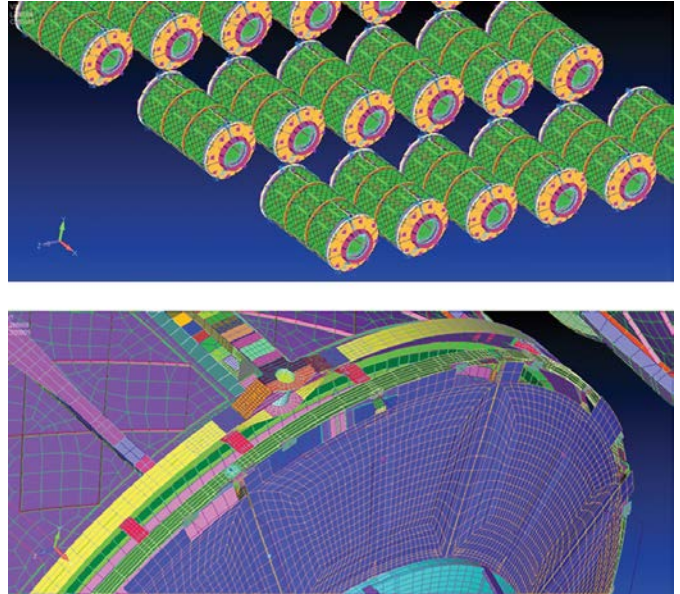


*Figure 2: Large test model.*

# Results attachment to the database

According to many in the finite element analysis community, organizations are bracing themselves for the next big data explosion. Driving factors include, FE model sizes increasing continuously as discussed previously, analysis types such as nonlinear and dynamics becoming more common, and increasing software efficiency and hardware performance. One of the limits to realistic model size is the ability to store and retrieve the vast amount of data generated in a Finite Element analysis. Femap 11.0 has taken an important step forward in dealing with this situation with the new ability to attach a results database to a Femap model, rather than storing it internally. The schema for achieving this is shown in figures 3 and 4, contrasting the old internal storage and the new attached storage approach.

The key feature is that the Femap 11.0 model file size is kept to a minimum as the results are not incorporated into the model database. In addition, the tendency for the model file size to grow with successive result import and deletion operations is avoided. Significantly increased efficiency has been found when accessing transient results when attaching, versus importing.

Many industries require very large numbers of results sets to be addressed and again the new schema is ideal for that application as shown in figure 5.
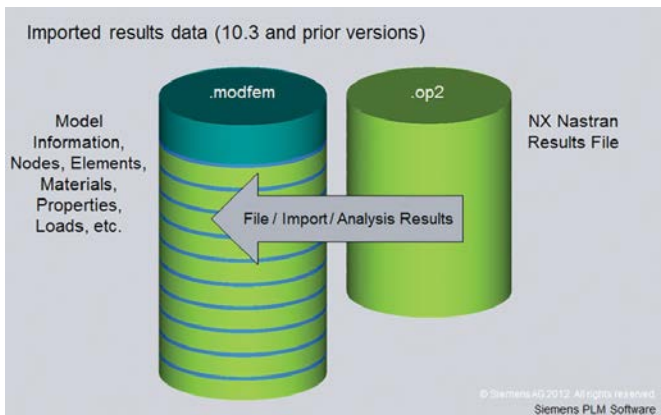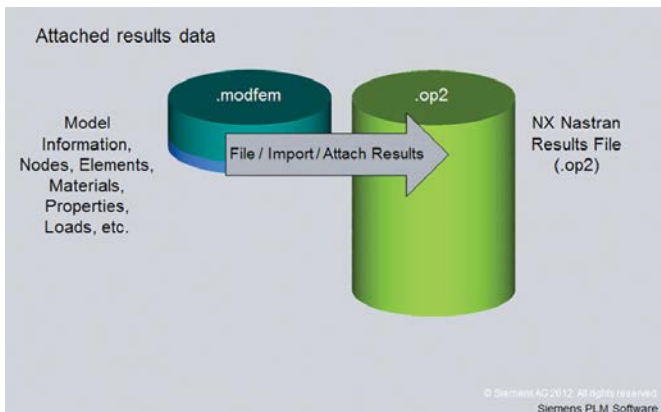


Figure 5: Attaching to multiple results files.



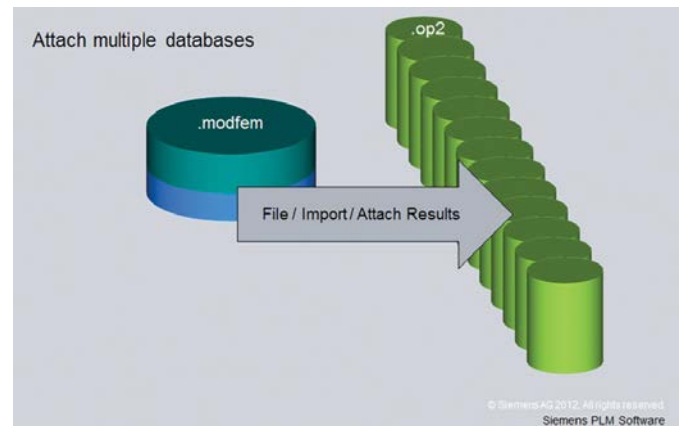Figure 3: Importing results data.

Results for Model size using the new Attach and old Import methods are shown in figure 6 and show very significant improvements in model size and results acquisition times.
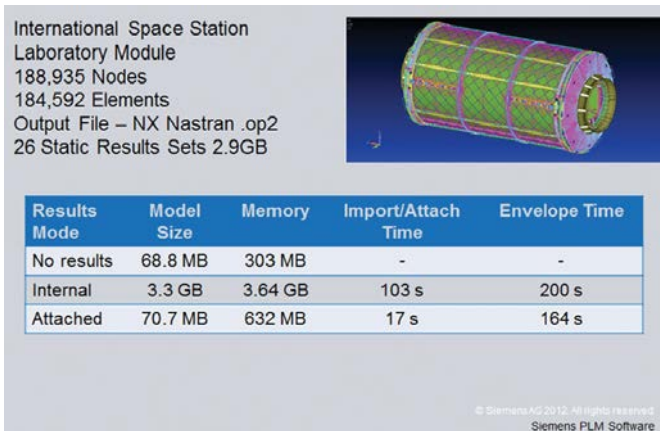


Figure 4: Linking to results data.

**A white paper issued by: Siemens PLM Software.** 6

International Space Station
Laboratory Module
188,935 Nodes
184,592 Elements
Output File – NX Nastran .op2
26 Static Results Sets 2.9GB

| Results Mode | Model Size | Memory | Import/Attach Time | Envelope Time |
|---|---|---|---|---|
| No results | 68.8 MB | 303 MB | - | - |
| Internal | 3.3 GB | 3.64 GB | 103 s | 200 s |
| Attached | 70.7 MB | 632 MB | 17 s | 164 s |

*Figure 6: Internal vs. external performance comparison.*

All postprocessing functionality does not change and the attached and Internalized Results can coexist.

Siemens stated rollout plan is for the Nastran 'op2' results file format to be addressed initially in Femap 11.0, followed by ABAQUS 'odb' results format, Nastran 'xdb' results format and ANSYS 'rst' results format.

# Creating geometry from mesh

A powerful new feature in Femap 11.0 is the ability to recreate geometry from existing Finite Element mesh. There are several reasons why you might need to do this. Imagine an analysis project where the parent geometry is unavailable and you inherit a bare Finite Element model input file – then this feature is a lifesaver. You can import this mesh and rebuild key parts, or even the whole model, by creating geometry based on the original mesh topology. This will allow you to then take full advantage of Femap database relationship between the new geometry and the old mesh. The new surface geometry inherits all the material and physical properties of the mesh.

Another scenario is where design changes are required in a Finite Element Model, but you do not want to rebuild the geometry and remesh the whole model. Local mesh regions can be used to build surrogate geometric surfaces, avoiding the need to disturb the global geometry model. This geometry is then used to model the new detailed features, such as cutouts and new surface intersections. There are many powerful tools available in Femap to manipulate geometry. Finally the new zones are remeshed with the new local detail included.

Figure 7 shows a simple flange which is imported as mesh only, with no geometry.
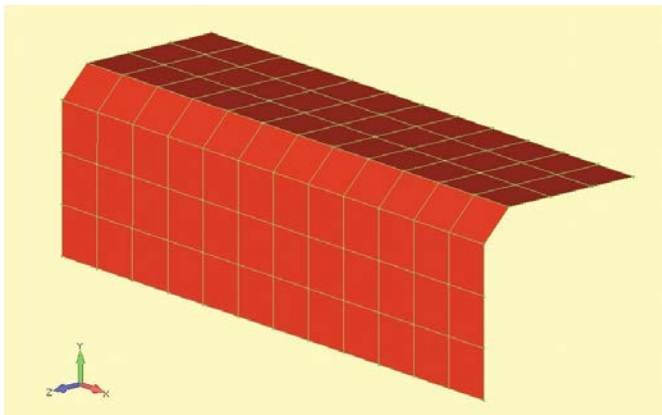
choose wireframe geometry. The default geometry fit is shown in figure 8 and as can be seen the geometry is based on the original mesh topology.
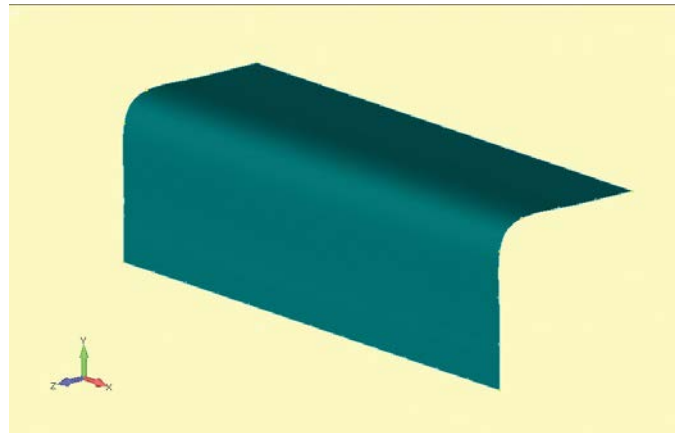


*Figure 8: Surface created from mesh.*

The boundary mesh 'seeding' or spacing is mapped onto the edges and the original shell element properties are inherited, which allows very fast re-meshing setup. In the case of the model shown in figure 9, the original mesh is deleted and the mesh is refined for the complete flange model.
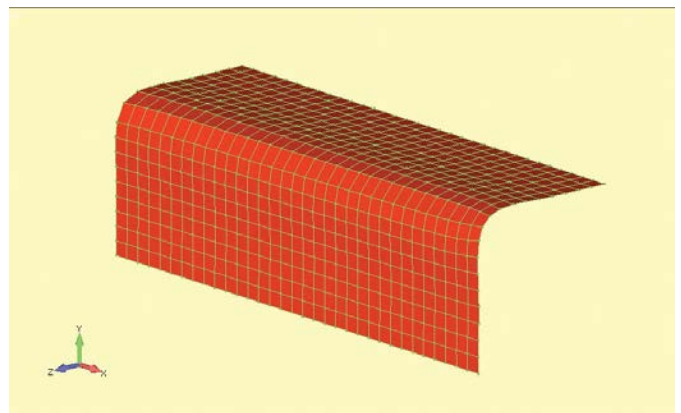


*Figure 7: Flange with mesh only.*



*Figure 9: Refined mesh fitted through rebuilt surface.*

Using the new Geometry/Surface/From Mesh command the whole mesh can be selected and converted to a single geometric surface. There are controls to define the surface tolerance, allow the existing mesh to be deleted and to optionally

Figures 10 and 11 show a typical process where you want to add new features into a mesh. Here three mesh regions are selected for geometry creation and subsequent manipulation to form a cutout of a hole, a further mesh region is selected on the fillet for geometry creation and subsequent blending of a fin to be attached. The ability to insert geometry into a mesh in this way, and then create virtually any local feature using Femap geometry tools, is very powerful.
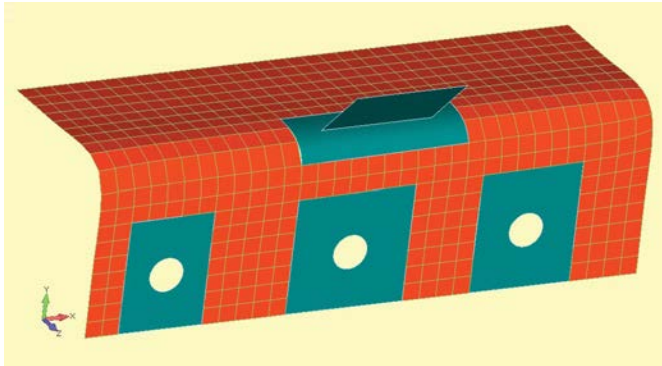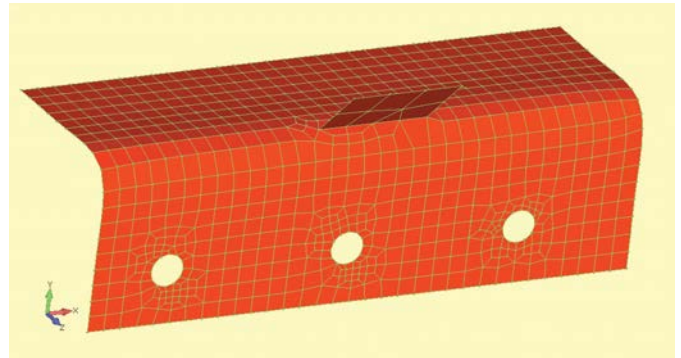


Figure 11: Surfaces meshed and new features embodied into flange model.



Figure 10: New surfaces created in mesh and modified to accept new features.

# Splitting a mesh from an edge

There is a great new direct meshing feature slipped into the Mesh/Editing options; Edge Split. This allows the edge of an element to be selected as the definition of a split in the mesh. It works for 2D shell elements and hex elements. In the 2D mode it will also split any associated line elements such as bars. How I wish I had this on previous models including aircraft fuselages and ship deck and hulls when refining the mesh. The usual scenario is that you need to introduce more features quickly, or that the mesh needs refining in a local region due to a stress concentration. Again, you want to do this fast to carry out a sensitivity study and can't afford the luxury of geometry modification and full model remesh. Figure 12a shows a simple example in a regular mesh, a starter edge is chosen and the mesh splits. Figure 12b shows the effect of limiting the region to be split. This is defined very easily by a zone of nodes. Figure 12d shows the effect on a general mesh.

A superb feature is that at junctions of skins or plates, where several elements are connected, the mesh split propagates in each direction as shown in figure 13. In complex fabrications visualization of this is a nightmare when splitting manually. This tool eliminates that situation completely. With use of the other direct meshing options such as delete and create pretty well any mesh can be sorted out for fast local refinement.
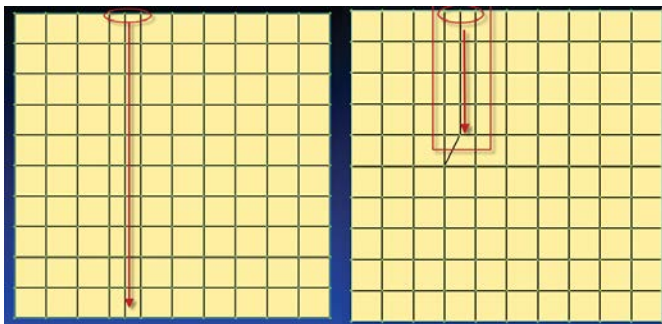


*Figure 13: Shell mesh splitting.*


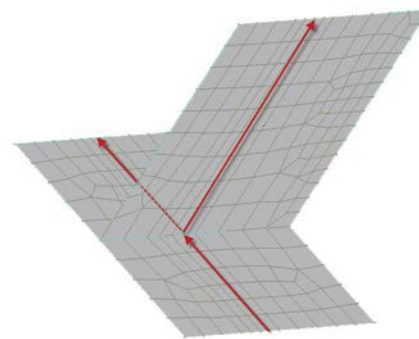
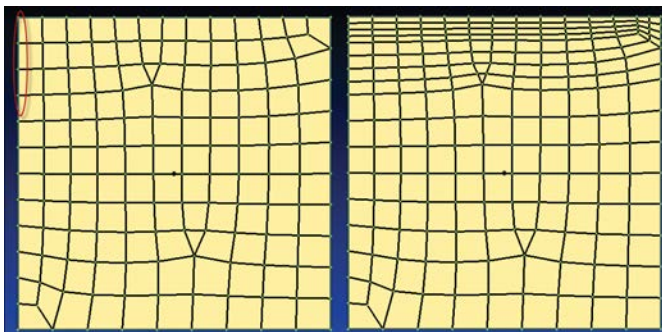*Figure 12a: Mesh splitting.*          *Figure 12b: Mesh splitting.*



*Figure 12c: General mesh.*          *Figure 12d: General mesh split.*

# New entity selection tools

The controls for selection of entities have always been extremely powerful within Femap, in fact if you want a quick boost in your level of productivity with Femap; take some time out to explore the richness of the direct, indirect group, list and pick features of the Entity Selection dialog box, along with the entity specific methods. A new addition in Femap 11.0 is the ability to add to a current element selection list by choosing 'Add Connected Elements'. There are many scenarios, but figure 14 shows an example. A rigid spider element has been connected to the edge of the channel section shown. The initial setup looks ok, but spider elements are notoriously fickle. By using the List/Model/Element command we can pick the spider – and add all connected elements to the selection automatically. The selection highlight – another great option, allows a clear view of the rogue 'leg' of the spider!
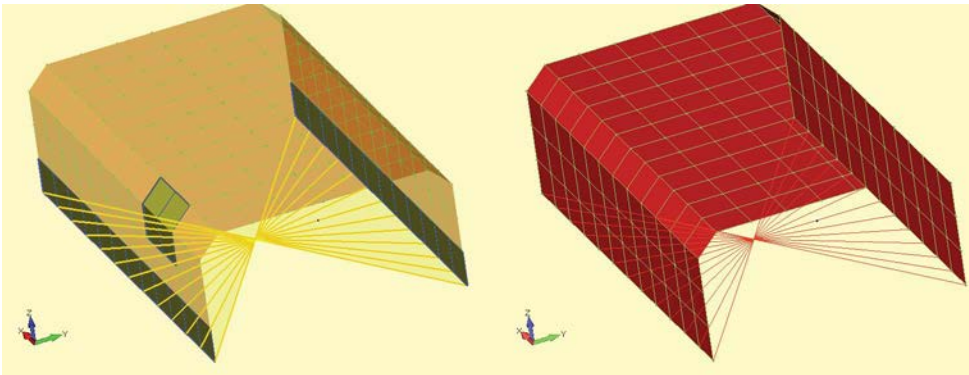


*Figure 14: List connected elements usage to spot error in rigid element.*

**A white paper issued by: Siemens PLM Software.**

# Model info tree updates

The Model Info tree view has been updated in Femap 11.0 so that whilst working in any open Dialog Box, the tree can be expanded or contracted using the +/- icons, and visibility icons are available for various entities in the tree. This is a good example of the sophisticated level of ergonomics which is such a strong feature of the Femap graphical user interface which has been continuously updated throughout its development.

Further improvements to the Model Info tree view include the ability to copy both constraints and loads between load sets as shown in figure 15. In this case a nodal load definition, set up for load set one is copied into load set two. The menu is invoked by right mouse click on the load definition entity and then the 'Copy To Set' command is chosen. This approach allows a rapid spawning of clones of the original, which can then be tailored to suit the individual loading. This is much faster than setting each up from scratch. If a complex set of loads is to be defined such as a set of unit load cases, and then combinations to form operational cases, then this is a very convenient way to do it and minimizing the likelihood of error.
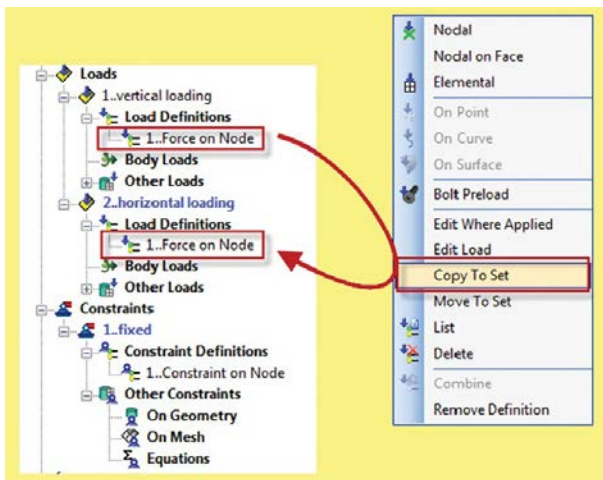


*Figure 15: The Move and Copy To Set commands in typical usage.*

# Composite layup creation new features

The definition of plies within a layup is controlled by the Layup Editor Dialog box as shown in figure 16. The Model Info tree also shown reflects the creation of the Layup Entities. This forms a very productive and elegant way of working with Finite Element Model entities and is common to all major features used in the analysis model definition. New in Femap 11.0 is the 'AutoCreate' option for use when setting up new plies in a layup. If selected then the newly created ply goes into a list of Global Plies. These can be re-used across different layups and avoids the tedium of inputting the same data across those layups. The global ply also has an important role in defining the definitive labeling of plies in a global sense. In this example, layup 2 has 2 plies, locally labeled as ply 1 and ply2. However across the whole composite structure they are defined as global ply 3 and 4. This is a very convenient way of allowing ply drop offs between adjacent layups, or totally different layup schemes to exist in the same model.

Finally, global plies may be used to create useful contour plots in Femap by concurrently displaying data from the same global ply in different layups.
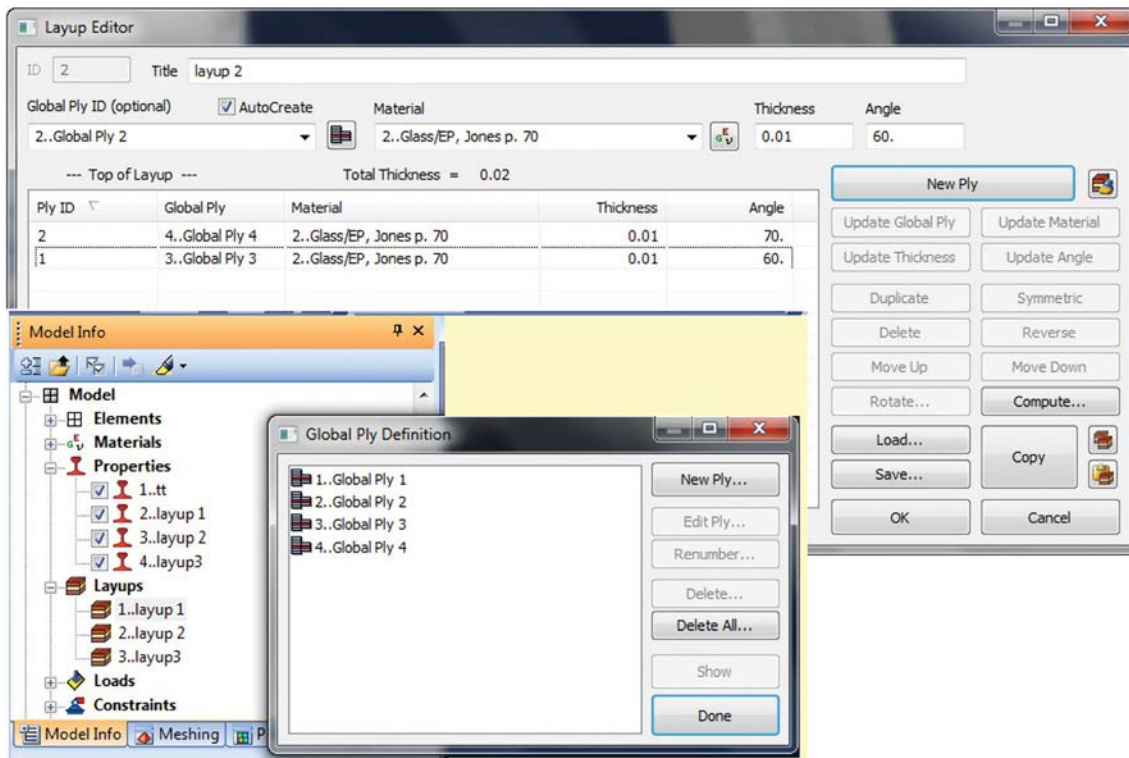


*Figure 16: Global and Local ply definition with new AutoCreate option.*

# Other new composite analysis features

Several other new features in NX™ Nastran® composite technology are now supported in Femap 11.0.

The Layup definitions described for 2d shell application can now be extended to 3D solid element application through the Laminate Solid property type. The same Layup can be used for a 2D shell; however the additional information on how the plies are stacked through the solid is defined by the plane in which the ply exists. For a solid element it can be in any of the global planes of the element, which is a great convenience as it means plies in solid composite elements do not have to be orientated in a fixed XY plane.

Support is now given for ply/bond failure values for solid laminates on several NX Nastran material types through the Ply/Bond Failure form. Supported material types include isotropic, 3D anisotropic and 3D orthotropic.

# Contour Model Data tool

Continuing on the theme of increased productivity a new tool available from the view toolbar is the Contour Model Data tool. This pulls together a lot of functionality that was available before into a common Dialog Box as shown in figure 17.
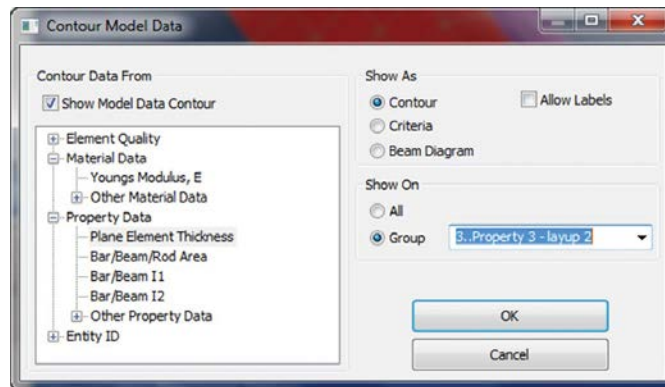


Figure 17: Contour Model Data Dialog Box.

This allows physical and material properties of elements to be displayed via color contour plots, labeled color fill plots or beam plots as appropriate. It allows for quick checking of mesh properties from one place. The other very attractive feature is that the color plotting or labeling can be restricted to a specific group. For detailed regions, or large models with many variations in property, this is a huge bonus. An example of usage is shown in figure 18. In this case there are three composite layup regions, with two having very similar thicknesses. The actual regions are defined via groups, so using the group filter it is easy to distinguish between the three regions.
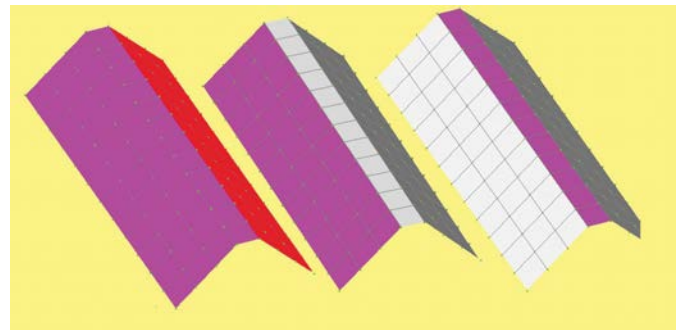


Figure 18: Property color contouring with group filter option.

Various other toolbars are new groupings of very convenient utility functions, this includes the Measure toolbar option as well as charting and post data, of which more later.

# Bolt preload

Preload in bolts is essential is most real world applications. Bolts simply do not work if there is no preload applied to form clamping load paths with increased break out load levels, frictional contact etc.

The development of specialized bolt modeling tools has steadily evolved in NX Nastran, the latest release now includes the option to model the bolt, together with the preload via a group of Hex elements. Femap 11.0 fully supports this important functionality. This is a very useful addition to the beam type method which requires a 'spider' connection to adjacent solid or shell elements.

Figure 19 shows a demo model of a bolt in isolation, with a preload added, but clamped at both ends. In practice the bolt would be part of an assembly using linear contact to provide appropriate load paths. The bolt shows correct overall axial loads, with more complex distribution at the clamped ends.

Definition of the bolt preload is very straightforward in Femap. Essentially a special bolt Region type is defined through the nodes on the cross section of the hex mesh forming the bolt. The Region also defines the bolt axis. This Region is shown in figure 20. This Region is then referenced by a Loading Entity – the bolt preload, which defines the value of the preload.

From this information NX Nastran is able to determine the cross sectional area and the bolt orientation and apply the correct pressures to each of the region faces.

The concept of using a bolt Region and bolt Loading Entity in Femap, together with the simple Model Info tree layout enables fast and accurate set up and checking of multi-bolt situations where various families of bolts can have different preload levels.
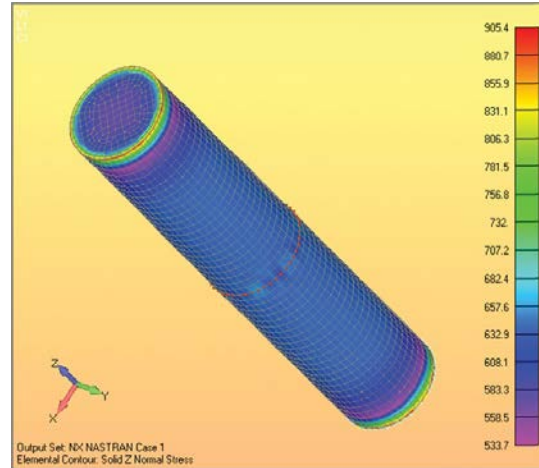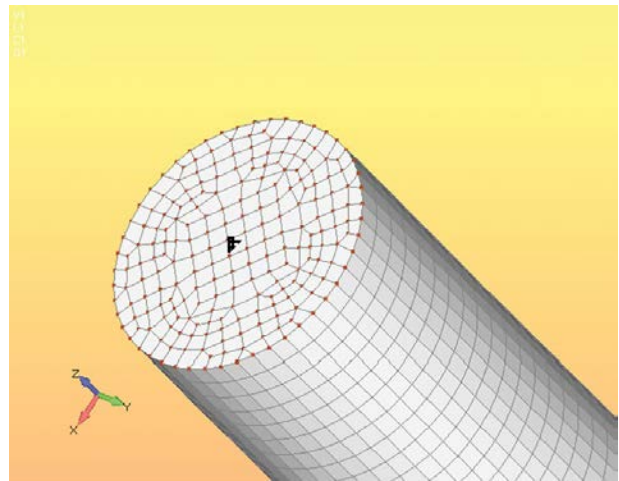


Figure 19: Solid bolt.



Figure 20: Solid bolt region.

# Result postprocessing improvements

In Femap 11.0 the Select Postprocessing Data form has been updated and streamlined and new functionality added. The new form is shown in figure 21.
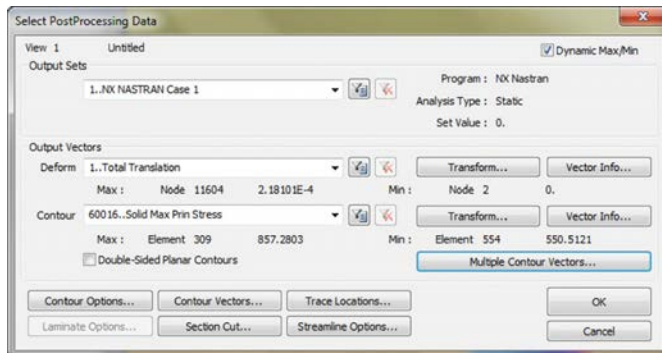


*Figure 21: Select Postprocessing Data form.*

A new filter Icon appears after the Deform and Contour Output Selection drop down lists. This Icon and functionality now appears consistently throughout Femap 11.0 wherever an Output selection is used. The filter allows a text string input which will be used to search for corresponding Output Set titles. In figure 22 a simple example selects only results sets in the '11.xxx' range of Frequency Response Sets. Useful text entries would include component directions, Load Case names, Max and Min designation etc. Again, with some user ingenuity in naming conventions, a very powerful data processing scheme could be set up very quickly.

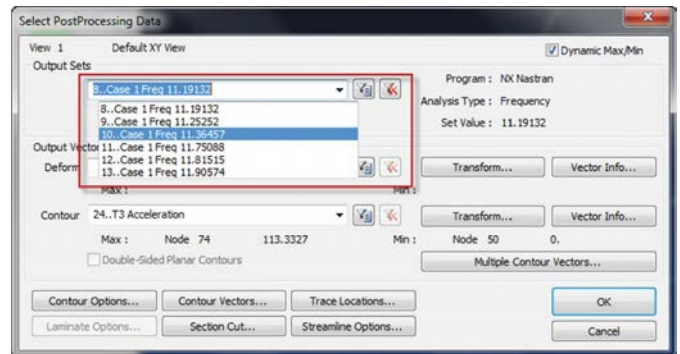The filter can be instantly switched on and off and updated.



*Figure 22: Select postprocessing data.*

The Select Postprocessing Data form also allows for Dynamic display of Maximum and Minimum results in each Output Vector set. This very fast preview is great for checking the sense of the data values quickly. If user error means a result of 1200 mm instead of 1.2 mm, then this will be spotted instantly before any serious postprocessing starts.

The Transform feature is now streamlined and easily accessible in Femap 11.0. The feature allows output sets to be mapped into any local coordinate system on the fly. It is extremely useful for checking local stresses for example. Stresses around a hole can be mapped to a local cylindrical system to instantly give bearing and hoop direction values. A component oriented in a local axis can have local axial, transverse and in-plane stress results plotted. If you want dig deeper into your results to understand what they mean then this is an essential tool.

The Contour Options feature also has new choices for contour plotting based on "All Data/Full Model", "Visible Group(s)" or "Contour Group". Dependent on what is selected the results are plotted on the appropriate node and element set and the contour range is also updated accordingly. A contour group is a specially selected group. Alternatively all currently visible groups can be used. This is very powerful as now all Femap groups have a visibility option so that multiple groups can be seen at once.

There is an additional option on the Select Postprocessing Data form in Femap 11.0 labeled 'Multiple Contour Vectors'. This enables output from multiple entity types to be plotted at the same time, with up to two additional output vectors available. This means that results on line, shell and solid elements can all be shown at the same time. Figure 23 shows a very simple example. For results visualization with complex structure with mixed element types this is a great improvement in productivity and avoids multiple plots of each type.

Another very nice enhancement is the ability to show results on different types of elements with the same topology such as beams, bars and rods, in a single plot.
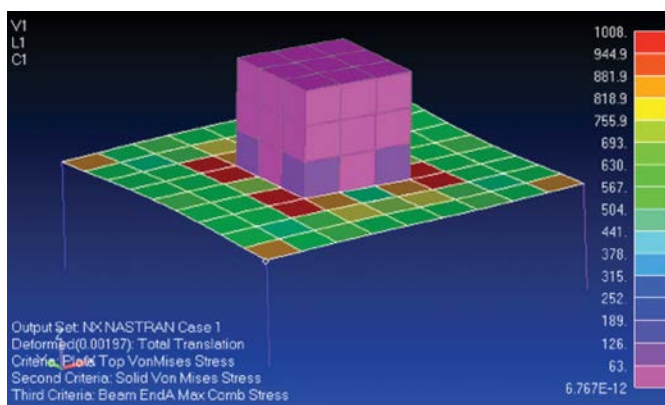


Figure 23: Multiple contour vectors.

Femap 11.0 has added ability to create "Envelopes" of results on the fly by selecting multiple Output Sets in the Model Info Tree. A right Mouse click on the group of output sets invokes the menu which includes the Envelope option. A choice of Maximum, Minimum or Absolute Maximum criteria can be chosen. Once selected this spawns a new Output Set which contains the relevant criteria value for each output vector available in the results (Displacements, stresses, forces, accelerations etc.). A second new Output Set is spawned which includes the Output Set Index associated with each criteria value. The menu setup is shown in figure 24.
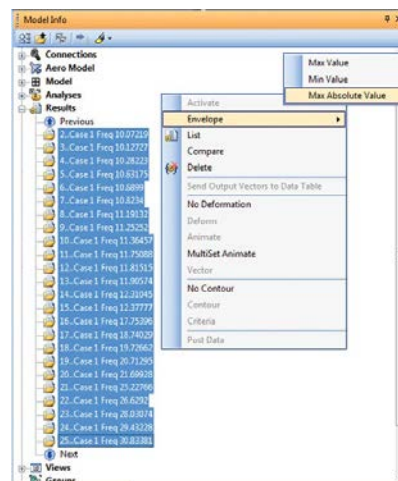


Figure 24: Results enveloping.

The two spawned output Sets can be used in combination to find, for example the absolute maximum value of acceleration across a range of Frequency Output Sets in a Frequency Response analysis, and the Output Set it occurs in. Any of the directions x, y or z can be quickly selected. Furthermore the two sets can be used in contour, deformation or criteria plots – as well as in the Chart Toolbox. That makes for a very rich set of tools for identifying key areas of model response in a quick and effective manner. The individual engineer will be able to develop a process tailored to his or her needs from within this toolset.

An example is shown using the Frequency Response analysis Output Sets in figure 25. The Chart curves show the maximum accelerations in x, y and z. The Envelope Output Set is used in the Criteria Plots to check the response at the same Node – and the results agree. A Spatial and Frequency based view of the model peak results is quickly established.
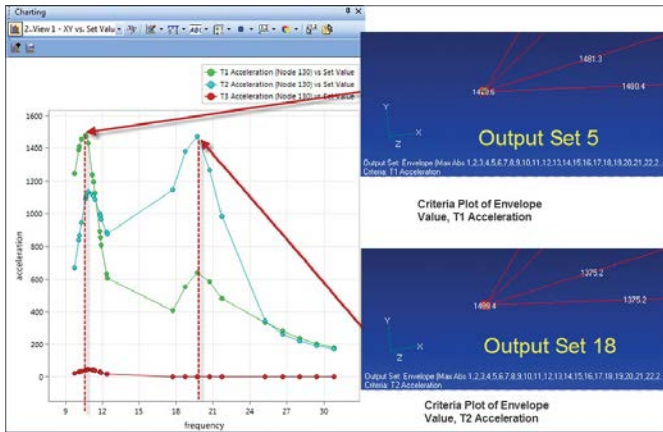
WHITE paper | Femap 11.0 new features



*Figure 25: Frequency response output sets.*

The List/Output command includes a very versatile set of options for filtering, enveloping and similar 'batch' type operations. The processed results can be sent to the Data Table, or directly to an output list – for quick 'on the fly' checks the Message window can be used. New in Femap 11.0 is the added ability to select the "Multiple" button for List/Output/ Compare option. This allows selection of any number of Output Sets for comparison, rather than just two. In addition, if the "Entire Set" option is "Off", then any number of Output Sets AND individual Output Vectors may be selected for comparison. Each case will be compared against each other selected case.

This provides another powerful way of gaining insight into the response of a complex model and gives the engineer wide scope to use ingenuity in producing appropriate envelope, filter and ranking type operations.

**A white paper issued by: Siemens PLM Software.**                                           **19**

# New charting toolbox

The new charting toolbox in Femap 11.0 is a major break-through in ease of use and elegance. For many engineers who rely heavily on XY plotting to interpret results, this feature alone will make 11.0 an important release! There is little reason to leave the Femap environment now with such a rich feature set available.

The toolbox is laid out logically so the user can get straight to work. Selecting analysis sets is via a very clear menu, allowing dynamic picking of single or multiple nodes or elements as appropriate. Filtering options, now consistent throughout 11.0, allow very powerful selection methods even when faced with significant amounts of data – for example concatenated modal, frequency response and random response data can be easily separated.

Figure 26 shows ten sets of frequency response data picked interactively from the model and presented in a clean and modern format. Modifying chart settings for items such as labels, markers, and exes are as close as a right-mouse click away and allow the user to quickly and easily tweak the appearance to their liking.
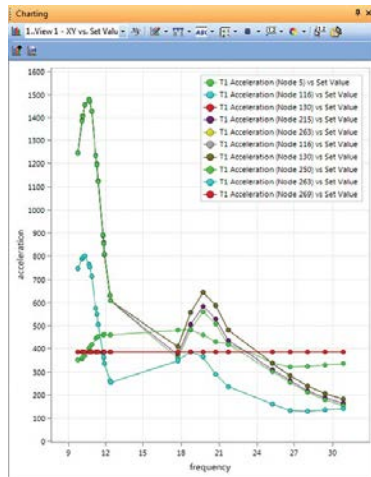


*Figure 26: Frequency response curves.*

Another intriguing aspect of the chart functionality is that individual curves can be selected as shown in figure 27. This curve can be highlighted, removed from the chart or deleted. This makes de-cluttering busy sets of curves to gain insight into the nature of a model response a very simple task.
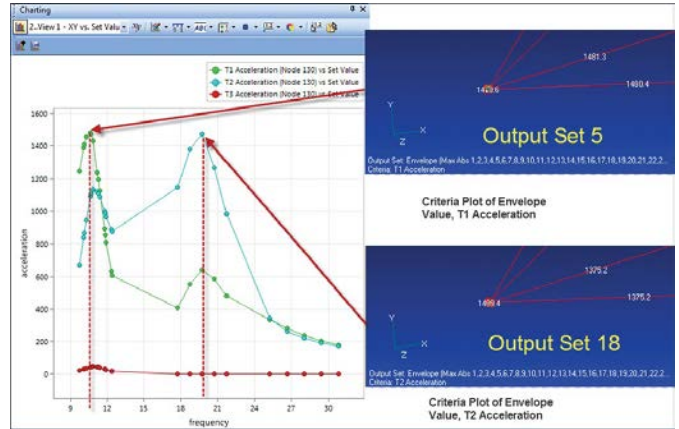


*Figure 27: Individual curve selection.*

Interactive selection and operation on curves from the chart window is very powerful, and is geared towards the need of the analyst. In figure 28 a curve has been selected from the chart and various options are available such as Marker, Color, Label etc., but also powerful features such as creating a function on the fly and copying data to the clipboard. For example, a shock spectra response curve generated from a transient analysis could be converted to an input function and captured for later use.
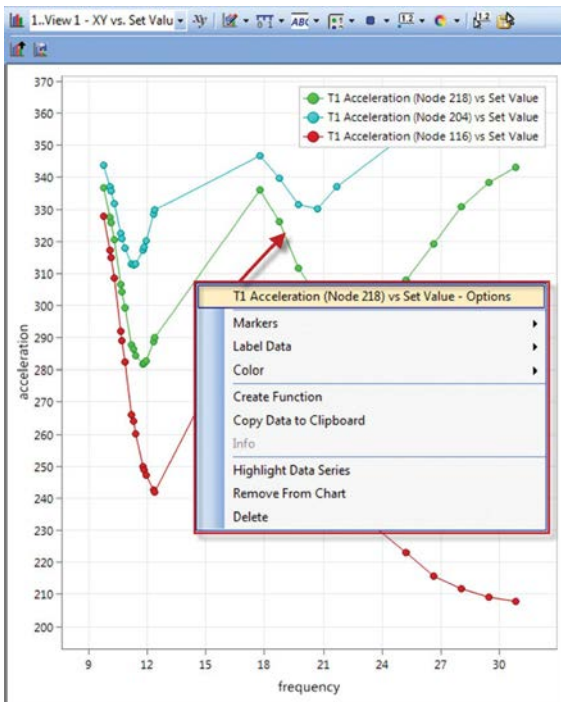
*Figure 28: Curve options.*

Siemens have said that this is an ongoing enhance-
ment program and that they are looking for user
input to help guide the future direction. Many of us
will have a shopping list based on the great founda-
tion laid here in Femap 11.0.

# Varying translational acceleration via body loads (ACCEL entry)

As analysis applications get steadily more sophisticated, older technologies have to be rethought. A case in point here is the application of inertial body loads to a structural mesh. In normal usage a component will see a single resultant vector of translational and rotational acceleration. However, some analyses now demand a varying acceleration applied throughout the body. NX Nastran has developed the ability to define that in an analysis using an ACCEL entry format. Femap 11.0 has an elegant new input paradigm to support this.

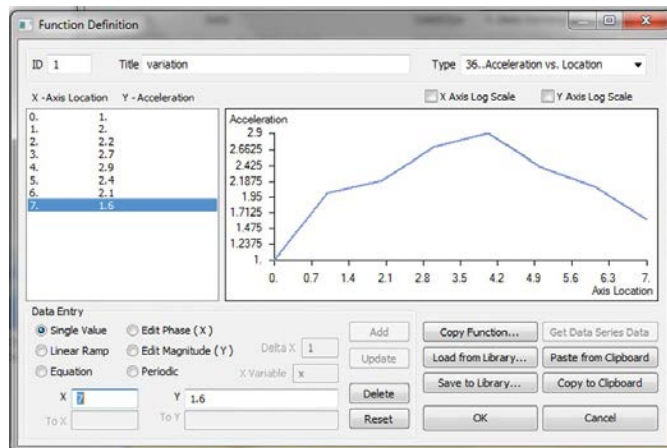The spatial variation of the acceleration field is defined via a function field, as shown in figure 29.
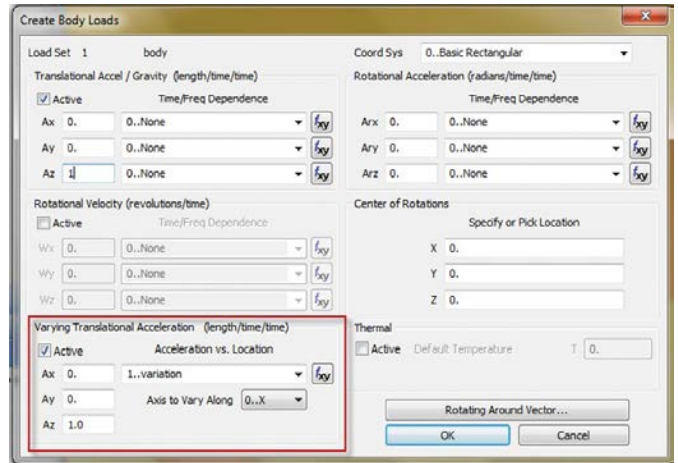


*Figure 30: Body load creation.*

An extremely welcome feature is the visual confirmation of the application region shown in figure 31. The function application direction and scale value is shown on the coordinate axes triad and the values and location of the function are plotted on the model. Data input of this sort is notoriously difficult to check so inclusion of this feature reflects the real-world appreciation the Femap development team consistently demonstrate.
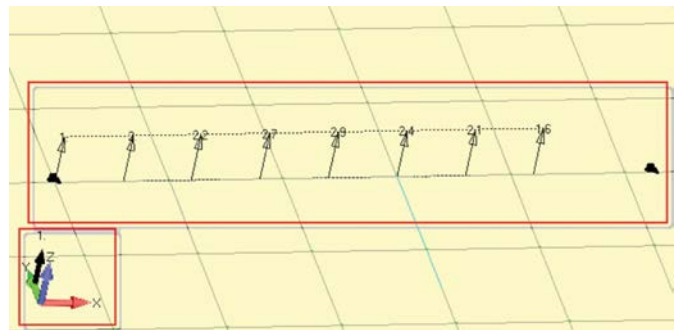


*Figure 29: Varying acceleration load function.*

The function is referenced from an extended body load dialog box as shown in figure 30 in a very logical format.



*Figure 31: Load application region.*

**A white paper issued by: Siemens PLM Software.**

# Nonstructural mass

A finite element model representation makes many simplifications of a real structure, including the mass, or weight distribution. Each element simply takes its volume, multiplied by the density to find its contribution to the total. Consider a real structure such as a building, a bridge, an electronics chassis or an aircraft. There are many items which just don't get included in the structural representation such as furnishings, fittings, paint, lagging, weldments, rivets, electronic components etc. If we are concerned with inertia loading, or doing a dynamic analysis then the total mass of the structure – including these nonstructural mass (NSM) items, is crucial. The normal approach is to smear the mass over 1D or 2D elements to get the right total and distribution. In many disciplines controlling this data is very labor intensive. NX Nastran has developed the concept of nonstructural mass

regions which are very effective in aiding the setup of this data. Femap 11.0 now supports this concept with an additional Region type specifically for nonstructural mass. Figure 32 shows the Model Info tree with four regions designated; each has a different NSM from 5 psi to 25 psi. The region was set up by reference to parent geometric surfaces, the associated elements then inherit the NSM terms. Alternatively the elements can define the NSM region directly. The figure shows region 3 expanded to check visually using a right mouse click in the Model Info tree. The NSM Region icon is shown. Editing of NSM value and associated surfaces or elements is very straightforward and visual, which really gives a boost in productivity to this important set up operation.
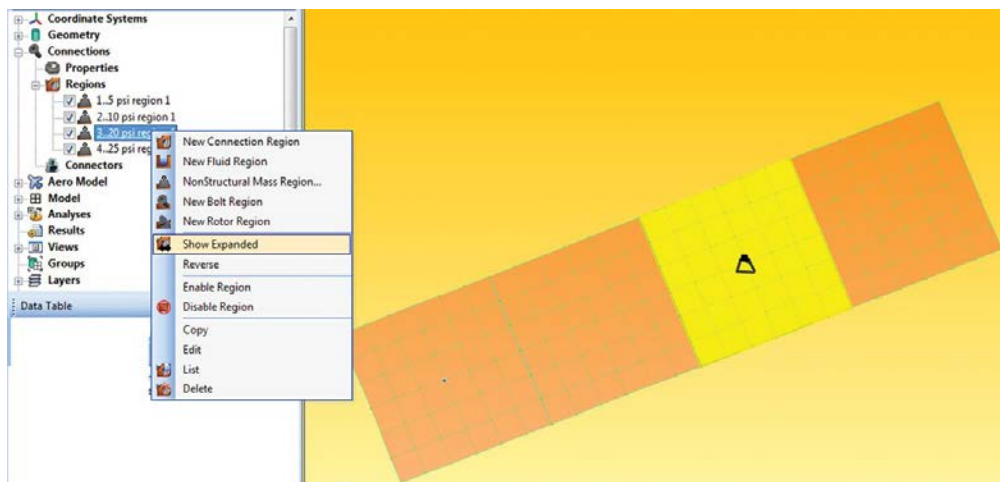


Figure 32: Nonstructural Mass region display and control.

# Conclusion

Well it has been a long and detailed review, but I make no apology for that. The scope of the new features is very wide and it is important to see them in the context of the superb Femap user interface paradigm. As stated in the overview, I hope the information has whetted your appetite, both for the new features and the existing rich set of features which grows larger with each release.

If you have never used Femap then you will be able to appreciate the strong ergonomic background to the product which provides a straightforward route through a range of very advanced Finite Element Analysis options. It is time to start looking at the product more closely.

If you are a Femap 10 user then I am sure you will be looking at the new features with enthusiasm, eager to try them out. But you will also be reflecting on the range of existing features and deciding that you too should be a Femap 'power user' with all the increase in productivity that will bring to your pre- and postprocessing work.

Prepared for Siemens PLM Software by Tony Abbey, FEA Specialist, December 2012.

**Siemens PLM Software**

**Headquarters**
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

**Americas**
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8287

**Europe**
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

**Asia-Pacific**
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

**About Siemens PLM Software**
Siemens PLM Software, a business unit of the Siemens Digital Factory Division, is a world-leading provider of product lifecycle management (PLM) software, systems and services with nine million licensed seats and 77,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software helps thousands of companies make great products by optimizing their lifecycle processes, from planning and development through manufacturing and support. Our HD-PLM vision is to give everyone involved in making a product the information they need, when they need it, to make the smartest decisions. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

GEOMETRIC SOLUTIONS

| Solution Partner | SIEMENS | Gold Smart Expert |
|---|---|---|
| PLM | | Channel |

844-GEO-SUPT
support@geoplm.com
geoplm.com

**A white paper issued by: Siemens PLM Software.**