

Femap Express for Solid Edge

www.siemens.com/solidedge

white paper



- ▶ Preconfigured, best-practice, process guidance for simple finite element analysis.

PLM Software

Answers for industry.

SIEMENS

Table of contents

Integrated FEA for Solid Edge	1
Femap Express – built-in analysis for engineers	2
Using Femap Express to analyze Solid Edge parts	4
Reports	7
Advanced analysis	8
Conclusion	9

Market pressures to reduce design cycle time, increase speed-to-market and improve quality are driving growth in the use of digital simulation throughout the product lifecycle. Femap® Express software allows designers to perform first-pass analysis and validate that parts are 'fit for purpose' while keeping costs low. By moving analysis to an earlier stage in the design cycle, Solid Edge® software users can make sure their designs will be strong enough and function as intended in order to avoid costly recalls.

Solid Edge Femap Express features

Built-in analysis for part and sheet metal models

- Stress analysis
- Modal analysis (natural frequency)
- In-context (within assembly) part analysis
- Factor of safety
- Deflection
- Sheet metal mid-surface support
- Adjustable accuracy settings (coarse to fine)
- Simplified part = less complex mesh = faster performance

In-window, pre- and post-processing

NX Nastran, included as standard

- Provides functional advantage over competitive products
- Proven reliability
- Adds credibility

Results options

- HTML reports
- Save animation – avi
- Show/hide contours

Created specifically for design engineers, Femap Express provides pre-configured, best-practice, process guidance to the user for simple finite element analysis. Using the same process-based approach found within other Solid Edge capabilities, finite element technology is presented to the user in an easy-to-follow workflow using Solid Edge's patented SmartStep approach and detailed analysis tasks, which are undertaken within a single Solid Edge window. Analysis settings and results are stored with the component; if the part changes, the results are modified without needing to repeat the process of applying loads and constraints. Detailed HTML reporting capabilities and animations capture response-due-to-analysis conditions and full documentation of the results, containing meaningful images of stress contours.

Femap Express flexible analysis tools

Two main analysis types are available – 'static' where stress, safety factor and deflection can be analyzed and 'modal' to determine the natural frequencies components operate at. Both can be used to analyze a 3D solid component or a sheet metal part using its mid-surface.

Finite element analysis:

In its application, a 3D model is represented by a geometrically similar model consisting of multiple, linked, simplified representations of discrete regions. These discrete regions are finite elements. There are several stages to perform a finite element analysis. Femap Express guides engineers through the process by means of an easy-to-follow wizard, via the familiar SmartStep ribbon bar.

Functional advantages

Due in part to its heritage, Femap Express enjoys several capabilities that provide functional advantages over seemingly comparable analysis tools.

Including NX™ Nastran (the well-respected, proven and recognized solver) as a standard feature of Femap Express ensures that users can be highly confident in their results.

Individual parts can be analyzed either independently or in the context of an assembly, where surrounding parts can be used to help determine where a force vector or holding constraint should be applied.

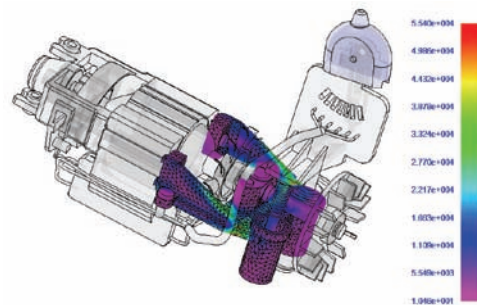
► Femap Express – built-in analysis for engineers (continued)

The ability to predict component strength in a static condition is only one of the problems engineers face. In machine design, many components operate at high speed; these components can start to resonate at a particular speed (RPM) or frequency (Hz), which can lead to catastrophic failure. This stress is hard to predict and may not be recognized until the machine is operational, resulting in some costly consequences. The ability to predict the four major natural frequencies (modal) of a given component makes Femap Express the first practical embedded solution for engineers to solve real problems early in the design, leading to higher quality products at a lower cost.

Analyzing disproportional components that are extremely thin in relation to their length (like sheet metal components), using a normal 3D finite element mesh approach, is both very resource hungry and yields highly inaccurate results. Femap Express uses 2D 'plate elements' to analyze an automatically extracted sheet metal mid-surface delivering fast, accurate results.

In addition, Femap Express can take advantage of the Solid Edge concept of 'simplified parts,' ignoring design details that are not important to the analysis. Accurate results are returned in less time.

Detailed HTML reports, images and animations are quick and easy to produce. The reports include detailed information on part and material properties, constraint information, as well as the results for modal or static analyses. Stress contour images graphically show stress levels, displacement and factor of safety.



Solid Edge Femap Express key advantages:

- *Modal analysis*
- *Sheet metal mid-surface*
- *NX Nastran, included as standard*
- *In-context part analysis*
- *Works with simplified models*
- *Automatic HTML reports*

► Using Femap Express to analyze Solid Edge parts

The remainder of this document explores some of the specific analysis commands and step-by-step workflows that make Femap Express so intuitive and easy to use, allowing engineers to add value to their design process by doing first-pass analysis and eliminate problems early in the design stage when they are much less expensive to deal with.

Process-oriented workflow

One of the areas that allows Solid Edge to stand out from the competition is its process-specific workflow philosophy to simplify design tasks. Femap Express uses SmartStep to guide users through the steps required to accomplish a finite element analysis.

Body selection

Through the SmartStep ribbon bar, users are first prompted to select the body (most models have only one, however Solid Edge is capable of creating multiple bodies in one file); simplified parts can be used instead of the as-designed parts. This is particularly useful as much of the detail is insignificant, has no real influence on the results and unnecessarily complicates the analysis.

Types of analysis

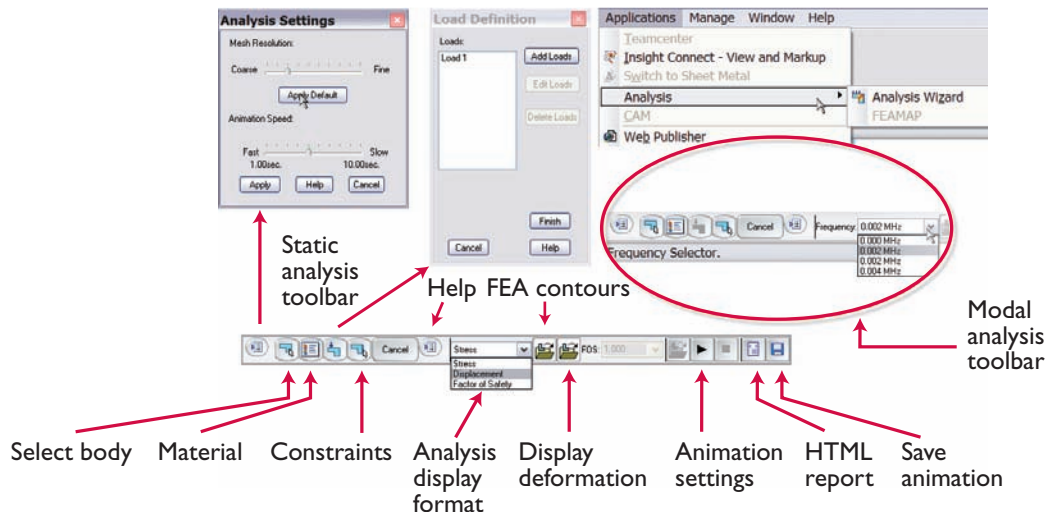
Femap Express is able to solve two types of analysis problems faced by engineers: static is used to find the stress levels within a component under certain load conditions; displacement indicates how much a component will displace or flex under load. Modal analysis predicts the shape of a component when it is vibrating at its four major natural frequencies, important for determining the dynamic characteristics of the design. Modal analysis is used to determine natural frequencies and shape of the part as it vibrates at these frequencies. If the part is operating in an environment where it experiences loading at a

frequency close to a natural frequency, the part can vibrate with an increasing amplitude and fail – just like an opera singer and a wine glass.

Static analysis (stress) – Static analysis is used to calculate the stresses in a component to help predict strength and durability for given loading conditions. For example, when a wire is pulled tight, it stretches (undergoes strain). The amount it stretches is proportional to the load divided by the cross-sectional area of the wire: $\sigma = F/A$. Failures occur when the load exceeds a critical value for the material – the tensile strength multiplied by the cross-sectional area of the wire, $F_c = \sigma_t A$.

Modal analysis – Nearly all objects when hit, struck, plucked, strummed or somehow disturbed will vibrate. If you drop a yardstick or pencil on the floor, or pluck a guitar string, these objects will begin to vibrate. When each of these objects vibrates, they tend to vibrate at a particular frequency. The frequency or frequencies at which an object tends to vibrate is known as the natural frequency of the object. The shape of the part when vibrating at one of these frequencies is the mode shape. Modal analysis is used to determine the natural frequency and shape of the part at this frequency. If the operating environment contains a loading condition at a frequency close to a natural frequency, the part can vibrate with an increasing amplitude and fail.

Sheet metal analysis – For thin bodies, a different type of meshing approach is more efficient. So for sheet metal parts, a mid-surface can be extracted and meshed using 2D plate elements instead of 3D solid elements. These types of structures can be represented much more efficiently using 2D elements without compromising accuracy and minimal solution time and use of computer resources. Sheet metal mid-surface can be used for both static and modal analysis.



▶ Using Femap Express to analyze Solid Edge parts (continued)

Choosing the material

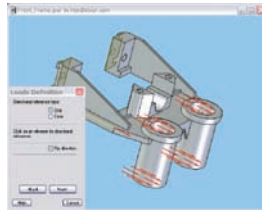


Accurate material data is critical for analysis and has a huge influence on the results. Choosing the correct material is also important for designing parts that are suitable for their application. Material properties available in Solid Edge are accessed directly from the part. The material step allows materials to be easily changed, if the designer wants to test

alternative materials. If the material property is not detected in the part, users will automatically be prompted to choose a material during this process.

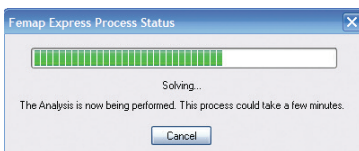
Placing a load

Loads (pressure or force) are placed on edges or evenly across a face respectively. Multiple loads can be placed at the same time. Because parts can be analyzed in context of an assembly, other components within the assembly can be used to set the force vectors, as well as edges of the part itself. This capability is not present in comparable analysis tools. Units such as N, Nm and lbf are set in File Properties.



Constraining parts

For loads to have an effect on a part, it needs to be held in place or constrained. Next the part is constrained along faces or edges by simply choosing where to hold the part. Once all the analysis criteria have been gathered, users are notified that an analysis can be performed. Loads are applied to the model and then underlying Femap technology automatically creates the mesh. NX Nastran is a standard feature of Femap Express and is used to solve the analysis.



Processing

The system has now collected all the information and input from the user it requires to perform the analysis. Loads,

constraints and material properties have been applied to the finite element model and the process invites the user to 'analyze' the part. The model is automatically meshed and the analysis performed.

Meshing

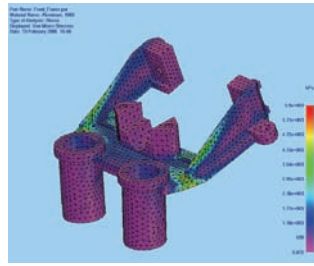
The mesh contains the data on material and structural properties that define how the part will react to certain load conditions. An optimized 3D mesh is automatically applied directly to the Solid Edge model, based on model proportions and level of detail to determine the mesh density. The same model, when simplified, will have a different mesh size than as designed. The mesh can be refined for greater accuracy. Femap technology is used to mesh a model.

Results

During the analysis, von Mises stresses for the component are calculated. The results display format can be presented in different ways depending on what criteria need presenting. Deflection and finite element contours can be turned on or off in the display. Results can be animated with any combination of these settings. Animation speed is adjustable and the animation can be saved as a separate avi file. The model window displays the FEA criteria and a gradient scale to help read the results.

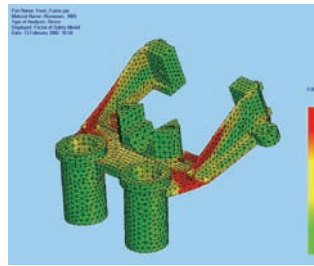
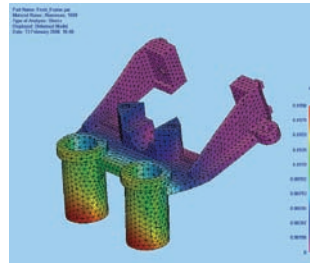
Stress in general is a complicated six-dimensional tensor quantity. Von Mises stress reduces this to a single number (scalar) for the purposes of calculating yield criteria. Von Mises stresses, named after Hungarian applied mathematician Richard von Mises (1883-1953), are typically presented in finite element solutions.

► Using Femap Express to analyze Solid Edge parts (continued)



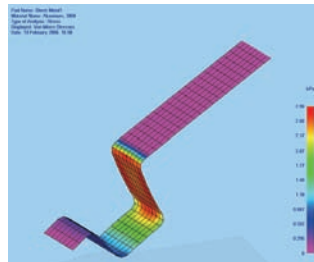
Stress – Results show the mesh and colored stress contours which show areas of stress buildup, and a scale which shows the values using units that have been set for the stress analysis, such as kPA, PSI and BAR plus many others.

Displacement – Results show mesh and colored displacement contours which show where the model deflects and a scale which shows the values using units that have been set for the stress analysis, such as millimeters, centimeters or inches.



Factor of safety – Results show mesh and colored factor of safety (FOS) contours indicating areas where the model exceeds the determined factor of safety and a scale which shows by how much a component is within or outside its FOS.

Natural frequency – Results show the four major natural frequencies of the part and the shape of the part at those frequencies.



Mid-surface – The sheet metal mid-surface is a method of analyzing parts using 2D plate elements to make the analysis more efficient. The mid-surface results can be displayed using both static and modal analysis.

“The new FEA tool inside Solid Edge v18, Femap Express, is powerful. I was able to work through my design with six different material size selections and configurations to ultimately find the right solution. I did all of this in less than an hour. I have to say – WOW! Femap Express, which includes NX Nastran is very cool!”

Shawn Oxley, Orbital Systems, Ltd., Irving, Texas

Results are quick and easy to save using the Femap Express report generation tool. A detailed HTML report contains in-depth analysis information on part and material properties, constraint information, modal and/or static results. Stress contour images graphically show where stress is concentrated, maximum deformation occurs or where parts exceed a factor of safety.

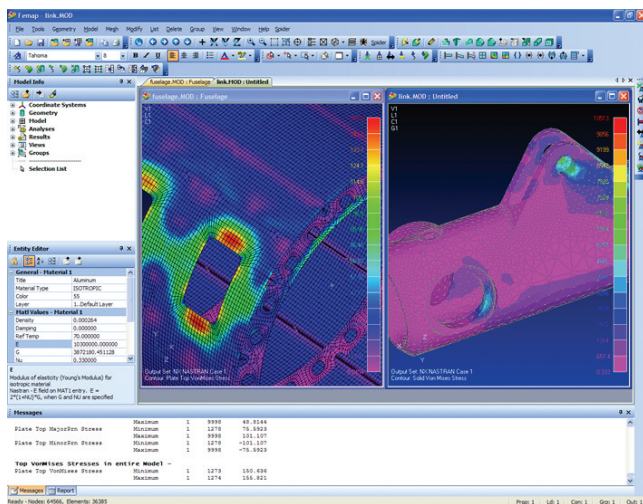
Animations/images

The Save Animation tool allows users to save individual pictures in bmp, jpg or tif format. A second option saves the currently displayed results animation as an avi file.



► Advanced analysis

Models can now be associatively passed to Femap from Solid Edge for more detailed and advanced analysis types including statics, modal, buckling, heat transfer and nonlinear studies for parts and assemblies. Design detail that is not necessary for the analysis can be removed by simplifying the parts, when parts (including sheet metal) and assemblies are transferred to Femap. If simplified part representations or sheet metal mid-surfaces are present in the Solid Edge files, they are automatically recognized and users can choose to use them in place of the designed body, which will speed up the meshing and processing operation to return faster results. Full associativity is maintained between Solid Edge and Femap; all material properties and colors are transferred with the Solid Edge model.



Femap is the world's leading Windows-based engineering simulation tool for finite element analysis. Engineers worldwide use Femap to model and simulate everything from simple solid components to entire spacecraft assemblies throughout a broad range of engineering disciplines. From simple linear static analysis right through to advanced solutions-based computational fluid dynamics, engineers and analysts use Femap to virtually simulate a complete range of product behavior before committing to expensive product development plans. This ensures higher product quality, lower development costs and reduced product development time.

▶ Conclusion

Today's analysts and engineers no longer have to build separate, meshed geometric models based on their CAD files in order to run FEA analyses. Femap Express is an FEA application closely integrated to CAD that allows CAD and entry-level FEA technologies to work together within a common user interface while providing design engineers a quick, easy way to see if their designs will meet specifications. When they need to run complex, high-end analyses, analysts can still choose to do more in-depth analysis with Femap.

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 5.5 million licensed seats and 51,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software's open enterprise solutions enable a world where organizations and their partners collaborate through Global Innovation Networks to deliver world-class products and services. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

► For more information, contact your local Solid Edge representative:



DESTEK
YAZILIM

Destek Yazılım Ltd. Şti.
Barbaros Mah. Denizmen Sk.
No:4/2 Koşuyolu – Usküdar
34662 İstanbul
Tel: (0216) 651 98 01
Fax: (0216) 651 98 02
www.destek-yazilim.com

SIEMENS

Division headquarters

United States

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
972 987 3000
Fax 972 987 3398

Solid Edge – www.siemens.com/solidedge

Americas

800 807 2200
Fax 314 264 8922

Europe

44 (0) 1202 243455
Fax 44 (0) 1202 243465

Asia-Pacific

852 2230 3308
Fax 852 2230 3210