DIGITAL INDUSTRIES SOFTWARE

Simcenter Femap version 2022.1

Breaking new ground with advanced analysis types

Benefits

- Better representation of real-world conditions when simulating assemblies with moving parts
- Eliminate need for geometry simplification and manipulation typically needed to create a high-quality solid mesh
- Take advantage of advanced analysis types via improved support for specialized entities needed to run certain simulations
- Enhanced graphics performance and helpful user interface features designed to improve overall User Experience

Summary

Simcenter™ Femap™ software is a standalone finite element modeling (FEM) pre- and postprocessor for engineering simulation and analysis. The software is computer-aided design (CAD) independent and can import geometry from all major CAD platforms. It supports most CAD data formats. Simcenter Femap also works in combination with a wide variety of finite element analysis (FEA) solvers, including Simcenter™ Nastran software.

The latest release provides a variety of enhancements that will improve your productivity across the simulation workflow. Building on the advanced meshing capabilities added for Simcenter Femap 2021.2, this version sees the addition of automated hex-dominant meshing technology which eliminates the need for geometry simplification and subdivision typically required to create a high-quality hex-dominant mesh. Support for Kinematic Joints and Flexible Sliders, available in Simcenter Nastran's Multi-Step Nonlinear Kinematic solution, has been added to better capture the real-world behavior of assembly models which

Features

- Automated hex-dominant meshing for streamlined creation of models representing solid parts
- Kinematic Joints and Flexible Sliders required to perform flexible body dynamic analysis, without the need for a motion solver, are now supported
- Enhanced Simulation Monitor provides live feedback of simulation progress and data, including displacements, energy, contact, and time steps
- User Interface improvements provide access to a modern Online Help system, along with new options to control display and assist in entity selection

contain parts which move relative to one another. A new solution monitor optimized to provide valuable real-time feedback for Simcenter Nastran's Multi-step solutions is also now available. Finally, updates to the user interface designed to improve overall user experience in various areas, such as controlling display options, entity selection, grouping, and accessing the modernized Online Help System, have been implemented.

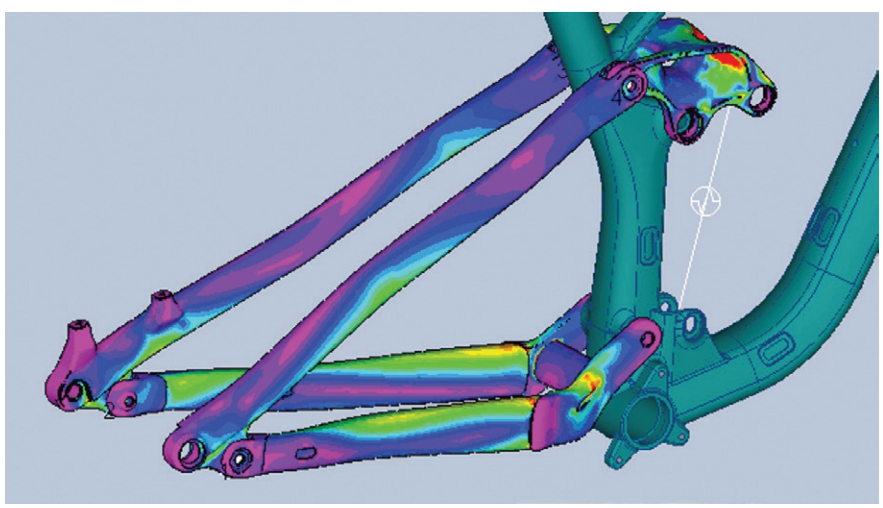
Preprocessing enhancements

Simulation Entities – Kinematic Joints and Joint Connections

Simcenter Femap 2022.1 provides support for advanced simulation methods, such as flexible body dynamics, using Simcenter Nastran's Multi-Step Nonlinear Kinematic solution, SOL 402.

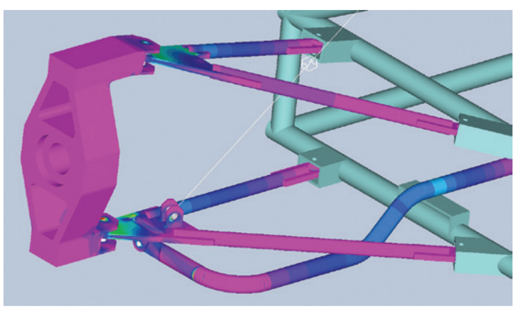
Kinematic Joints connect two nodes and allow a certain relationship of relative motion or rotation between the two depending on what type of joint is established. The types of Kinematic joints include revolute, inline, slider, spherical, cylindrical, and other highly specialized types which can be used for the analysis of models containing moving parts, such as aerostructures, helicopters, deployable structures in space, gas turbines, and machine tools. The addition of drive loads as a new load type can be used to enforce displacements or rotations on kinematic joints.

To accelerate the process of creating kinematic joints, Joint Connections have been added as a new unique entity type in Simcenter Femap. Joint Connections allow the user to establish how a kinematic joint will be connected to geometric entities and/or existing mesh of the model and are then expanded to the necessary nodes when the input file is exported.



Simulation Entities – Flexible Sliders

Simcenter Femap's support for kinematic analysis of mechanical systems is enhanced with the addition of "Flexible Slider" simulation entities. This capability includes several parameters for customization such as different slider types, driver loads, and additional friction options. The available slider types allow different constraints for relative rotation of the nodes sliding along the track. This includes spherical, prismatic, cylindrical, and universal types. For driver loads, Flexible Sliders can be driven with both external load sets, and forces/displacements applied directly to a specified driver node. Furthermore, for friction, available types include options for validating motion (no friction), infinite friction, and displacement or velocity dependent frictional forces.



Entity Display toolbar

The Entity Display toolbar has been enhanced to include an icon which can be toggled to select if the toolbar is currently controlling the overall display of the various entity types available on the toolbar or the labels for those entity types.

Grouping

New Group Commands to include nodes or elements in a group based on being associated to geometric entities which are part of a Solid have been added. For nodes, these include Group, Node, on Points of Solid; Group, Node, on Curves of Solid; and Group, Node, on Surfaces of Solid. The same commands are also available for elements.

Also new for grouping, is the addition of the Add to Copied Entity Groups option to the various Copy, Rotate, and Reflect commands for geometry and finite element entities, which places any duplicated entity into any existing group where the original entity resides at the time of the command.

Finally, the ability to evaluate any number of groups using the Model Info tree is now available by using the Evaluate command on the context-sensitive menu for Groups.

Selection of nodes and elements by geometric entities used by solid geometry

Like the commands mentioned in the Grouping section, options were added to Method^ menu in Standard Entity Selection dialog box for Node and Elements to select these entities based on their association to geometric entities used by Solids.

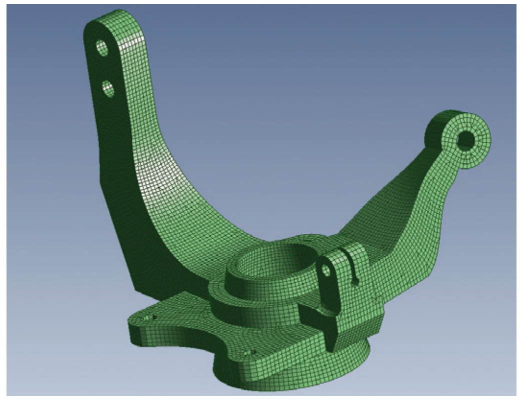
Meshing

Mesh, hex mesh bodies

Hex-dominant meshing of solid geometry with little to no simplification or subdivision into smaller and simpler regions has long been desired by the finite element analysis community. Simcenter Femap has collaborated with other development organizations in the Siemens Simcenter Portfolio to offer this exciting technology to Femap users for the first time for v2022.1.

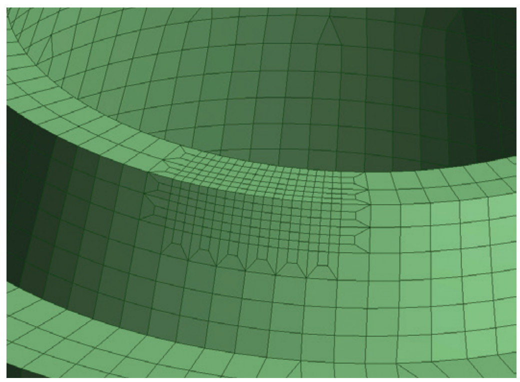
To accomplish this goal, the hex-dominant mesher first fills a solid volume with as many hexahedral elements as possible, then fills the remainder of the volume with wedge, pyramid, and tetrahedral elements, as needed. This process creates high quality elements which can be sent directly to the Simcenter Nastran solver with no additional interaction from the user or manual refinement of the mesh.

In addition to being able to mesh single parts, the hex-dominant mesher can work on multiple parts to create a single continuous mesh of the assembly. There are also additional options to control various aspects of mesh sizing, mesh associativity, and if all elements should be given midside nodes during the meshing process.



Mesh, editing, mapped hex refine

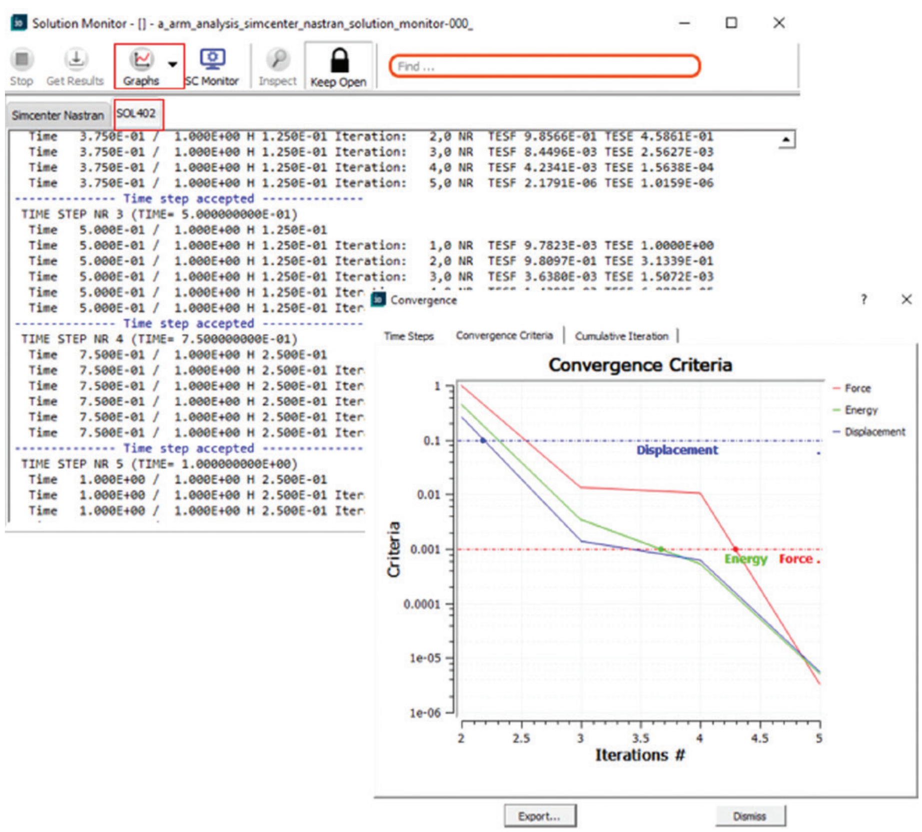
This command offers the ability to refine solid elements in regions of the mesh are fully mapped. Similar to the Mesh, editing, element refine command, the elements to refine are first highlighted, then are spilt when the user clicks OK to commit to the element splitting. Any elements which are needed to properly transition the fully refined elements back to the original mesh will also be split with a specialized transitiion pattern.



Solver support

Simcenter Nastran Enhanced Solution Monitor

Displays curated output from Simcenter Nastran including detailed warnings, information messages, and errors for any type of analysis. Separate SOL401 or 402 tab to display time steps and number iterations requested to converge the model. In addition, can be used to display graphs and information for time steps, iterations for each time step, and other items such as convergence criteria, cumulative number of iterations, elements which have experienced plastic deformation, and many other quantities.



Kinematic Joints – Joint Time Constraint

Dialog box in Analysis Set Manager to specify fixation time(s), liberation time(s), and/or removeable link(s) for Kinematic Joints in SOL 402 using groups of Kinematic Joints. Default is to set time for all Kinematic Joints in the model using a single entry in T/T1 field, but multiple entries can also exist. In either case, the appropriate entries will be written to the Nastran input file.

Flexible Slider Selection

Specialized dialog box controls which Flexible Sliders will be included for a particular Multi-Step Nonlinear Kinematic analysis run. Regardless of the number of Flexible Sliders has been selected, the appropriate entries are automatically written to the Nastran input file. The dialog box can also be used to highlight which Flexible Sliders are currently selected in the graphics window as well as to create new Flexible Sliders, edit any existing one, or delete any number of existing ones.

DDAM Enhancements

Added support for streamlined approach to specify control options for the Dynamic Design Analysis Method (DDAM). Previously, many of the control options were specified in an external file, but these can now be specified via the DDAMCTR entry in the Nastran input file. In additions, the base excitation had to be defined using a number of different entries, but certain aspects can now be defined using a SPDIR entry written to the Nastran input file

Support for Tension-Only Quad Property

Support for the PSHLPNL entry is now available by specifying Property Extensions for a Plate Property in Femap, which offers the ability for a shell element to a shear panel element when certain user-specified criteria have been met when using Simcenter Nastran's Multi-Set Structural Analysis solution. Once conversion has occurred, the element will continue to act as a shear panel for each Sequentially Dependent subcase following an initial Sequentially Independent subcase. In addition, a parameter which can be used to disable the conversion behavior, PARAM,TENSOQD, can be specified using the Analysis Set Manager.

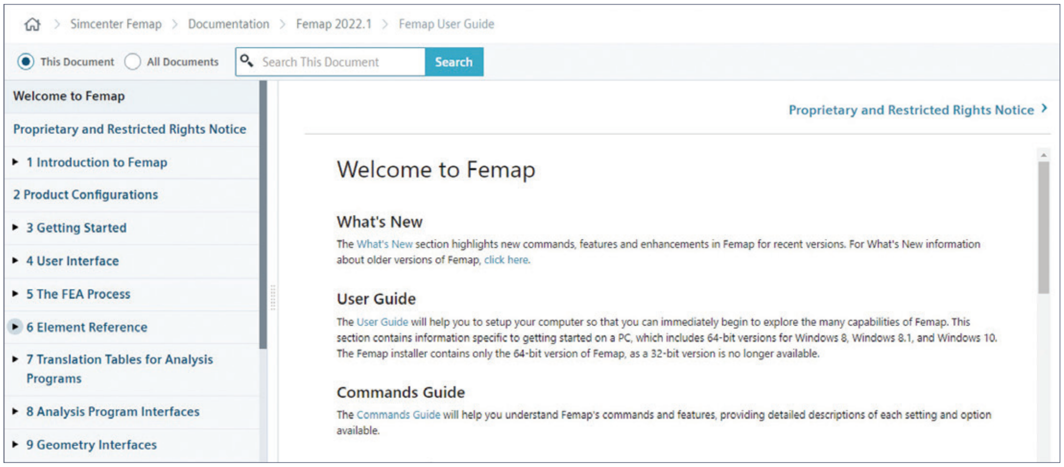
Support for DTEMP entry

Support for time-dependent temperature set definition for SOL 401 and SOL 402 has been added. In Femap, this is accomplished by creating a new set with Type set to Nastran DTEMP Sequence, then using the Referenced Set dialog box, Model, Load, Combine command, or the Load Set Combination Table in the Function/Table Editor to specify the desired time values to the appropriate Load Sets.

Nastran Solvers

Nonstructural Mass Axis for Beam Property

It is now possible to specify the Y Axis Offset and/or Z Axis Offset at End A and/or End B to define a Nonstructural Mass Axis for Beam Properties in the new Nonstructural Mass Property Values section of the Define Property dialog box for Beam elements. For consistency, Nonstructural Mass/Length has also been moved to the Nonstructural Mass Property Values section. Also, if values for Nonstructural Mass Axis are defined, they are used any time the Center of Gravity is calculated inside Femap, including for mass properties.



Help System

Femap's Online Help System has been modernized to provide the content in HTML format. It can now be access either online through the Siemens Support Center website or the help content can be downloaded locally and accessed by also installing the Siemens DI Software Documentation Server to the user's machine or a server machine which can be accessed by an entire organization. This new system is fully searchable and offers the ability set Browser Bookmarks as well.

Application Programming Interface (API)

Added Joint Object and FlexibleSlider Object, along with applicable Properties and Methods, to allow programmatic access to new Simulation Entities added for 2022.1

Added properties and methods to existing Objects to reflect new options added to the user interface for 2022.1

feCheckCoincidentNode5 can be used to access options in the Tools, Check, Coincident Nodes command which could previously not be specified programmatically.

Siemens Digital Industries Software

siemens.com/software

Americas
1 800 498 5351

Europe
00 800 70002222

Asia-Pacific
001 800 03061910

For additional numbers, click here.

84339-D3 12/21 H