Solid Edge Simulation

Powerful simulation functionality embedded inside Solid Edge

Benefits

- Reduce the need for costly prototypes with virtual testing
- Get your product to market faster with reduced physical testing
- Innovate more by experimenting with your design virtually
- Reduce recalls by finding out if your product fails before it reaches the customer
- Optimize material usage and minimize product weight
- Execute redesigns faster with synchronous technology

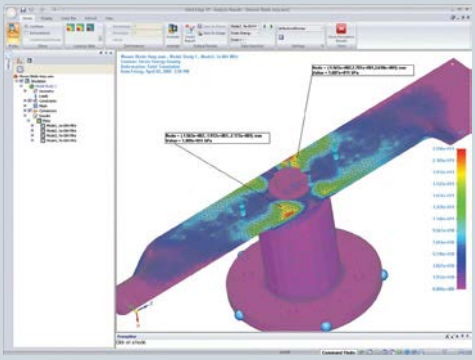
Features

- Embedded finite element analysis for design engineers
- Part, sheet metal and assembly contact modeling with realistic component interaction
- Automatic finite element model creation with optional manual override
- Realistic operating environment modeling with full complement of loads and constraint definitions
- Ability to maintain loads and constraints during model changes
- Synchronous technology for fast model redesigns

Summary

Solid Edge® Simulation software is an easy-to-use, built-in finite element analysis tool that enables design engineers to digitally validate part and assembly designs within the Solid Edge environment. Based on proven Femap® finite element modeling technology, Solid Edge Simulation significantly reduces the need for physical prototypes, thereby reducing material and testing costs, while saving design time.

For use by design engineers



This optional, built-in Solid Edge application uses the same underlying geometry and user interface as all Solid Edge applications. Solid Edge Simulation is easy enough for any Solid Edge user who has a fundamental understanding of FEA principles, yet robust enough to service most of your analysis needs. By enabling engineers to perform their own simulation, more analysis gets done in less time – improving quality, reducing material costs and minimizing the need for physical prototypes – without incurring the high costs of outsourced analysis. The layout of the user interface is designed to guide the user through the whole analysis process, with on-hand help if required, which makes it easy to learn initially, and revisit if necessary.

Automatic finite element model creation

You can create and refine finite element meshes where required to improve accuracy of results. Solid Edge Simulation supports solid meshes (using tetrahedral elements), two-dimension shell element meshes on mid-surfaced sheet structures, hybrid models that contain both 2D shell and 3D solid elements, as well as 1D beam elements for frame structures. A mesh size slider bar that makes element size adjustments to the overall finite element mesh is available with additional control of the number of elements on individual edges and faces. You can leverage a mapped mesh capability to take advantage of certain geometry topologies and create a more orderly and well-shaped mesh. In addition, the mesh size will automatically adjust to

Solid Edge Simulation

Features *continued*

- Ability to understand model behavior easily with comprehensive graphical postprocessing tools
- Full simulation scalability from Solid Edge Simulation Express or Solid Edge Simulation through to Femap
- Proven Femap modeling technology with the industry standard NX Nastran solver

accommodate detailed model features. In addition, you can fine tune the mesh with manual edge and face element sizing to generate an efficient simulation model that will deliver accurate results. Prior to creating the finite element model, you can use synchronous technology, with its ability to make history-free model changes, to prepare and simplify the geometry model quickly and easily.

Full complement of load and constraint definitions

Solid Edge Simulation provides all of the boundary condition definitions that you'll need to define realistic operating environments. The constraints are geometry based and include fixed, pinned, no rotation, symmetric and cylindrical variations. The loads are also geometry based and include mechanical as well as temperature loading for thermal analyses. Mechanical loads include forces, pressures and effects caused by body rotation and gravity. Solid Edge Simulation facilitates load and constraint application with Quick Bar input options and handles for direction and orientation definition.

Analyzing assemblies

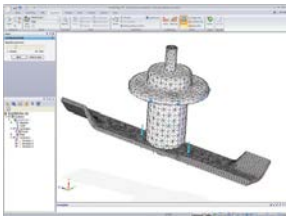
You can quickly connect assembly model components together. Assembly component interaction can be a glued connection between components or linear contact based on an iterative linear solution. Contact detection between components can be determined automatically, or connectors defined individually through manual face selection. Assembly materials and properties can be applied manually, selected from a material library or inherited from the geometry model by default. The included NX™ Nastran® solver assures realistic assembly component interaction that facilitates a robust solution.

Analysis types

You can obtain the results caused by a static loading, find the natural frequencies of vibration or determine buckling loads of a design using the industry standard NX Nastran solver. Re-use of finite element model loads and constraints is as easy as dragging and dropping from one study to another.

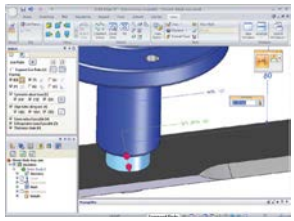
Analysis scalability

Simulation functionality within the Velocity Series™ software portfolio scales from Solid Edge Simulation Express for individual parts to Solid Edge Simulation, which extends analysis to assemblies, all the way to Femap with NX Nastran, thereby enabling you to define and analyze complete systems. This complete line of products provides a scalable upgrade path for users who need to solve more challenging engineering problems. Complete geometry and finite element models with boundary conditions and results can be seamlessly transferred from Solid Edge to Femap, where more advanced analyses can be employed if desired.



Postprocessing

You can interpret and understand the resulting model behavior quickly with the comprehensive graphical postprocessing tools. Model results can be displayed in a number of forms, including color and contour plots, which can be continuous, displayed as distinct contour bands or by element and displacement and mode shapes that can be animated. Min/max stress markers and a probe tool with result call-out boxes also are available. With Solid Edge Simulation's comprehensive postprocessing functionality, you can quickly identify problem areas for potential design revision and generate an HTML report of simulation model information and final results.



Design updates

You can quickly and easily make any required design update during post analysis. Synchronous technology lets you make history-free feature-based model changes at will, which drastically speeds up the model refinement process. In addition, Solid Edge Simulation maintains associativity between the CAD and FE models, while making sure that your applied loads and constraints are maintained for all geometry model changes.

Contact
Siemens Industry Software
Americas +1 800 807 2200
Europe +44 (0) 1202 243455
Asia-Pacific +852 2230 3308

www.siemens.com/solidedge

© 2012 Siemens Product Lifecycle Management Software Inc. All rights reserved. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. Nastran is a registered trademark of the National Aeronautics and Space Administration. All other logos, trademarks, registered trademarks or service marks used herein are the property of their respective holders. X15 16835 6/12 B