

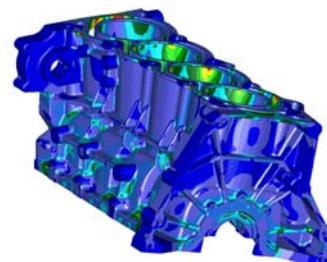
PTC Email Newsletter

January 14th, 2002

PTC Product Focus: Pro/MECHANICA (Structure)

Tip of the Week: Creating and using Rigid Connections

Upcoming Events and Training Class Schedules

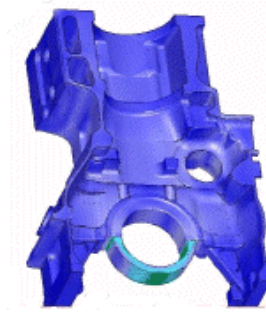


PTC Product Focus: Pro/MECHANICA (STRUCTURE)

Pro/MECHANICA lets engineers test and optimize designs for structural, dynamic, thermal and durability performance. By maximizing the parametric, feature-driven associativity of Pro/ENGINEER to quickly and accurately simulate the mechanical performance of a design, Pro/MECHANICA minimizes hardware prototyping, increases creativity and helps deliver better products in less time.

Pro/MECHANICA Structure allows design engineers to evaluate, understand, and optimize the static and dynamic structural performance of their designs in a real-world environment. Precise representations of CAD geometry, and unique adaptive solution technology provides fast, accurate solutions automatically - solutions that help to improve product quality, while decreasing costs associated with extended development times and ineffectual prototypes. Pro/MECHANICA Structure also offers specialized analyses that automatically create fully associative FEA meshes for products like NASTRAN and end-to-end interoperability with other PTC products and data management tools, including other CAD data.

By operating directly on the Pro/ENGINEER model, Pro/MECHANICA Structure eliminates data transfer issues while delivering the power of parametric optimization. Multi-disciplinary simulation capabilities additionally enable simultaneous structural, thermal, and motion design optimization. Pro/MECHANICA provides engineers with associative, parametric, interoperative, and internet centric modeling and simulation capabilities.



Capabilities

- Pro/MECHANICA Structure includes powerful, feature-based parametric modeling capabilities that allow you to automatically drive associative design changes from Pro/MECHANICA analysis and simulation. This powerful combination of functionality enables more design alternatives to be considered in a shorter period of time and allows changes driven from specific mechanical behavior to proliferate exponentially.
- Structural response simulation is the ideal tool for interrogating the static and dynamic structural performance of a design in real-world conditions easily specified by the engineer. Use the program's advanced vibration simulation tools to ascertain natural frequencies and vibration modes of a design, including transient dynamic and frequency response simulations - without a hardware prototype.
- Apply loads and constraints directly to the model geometry to simulate a real-world environment that mimics actual design performance
- Define the level of solution accuracy up front by specifying convergence settings before running a simulation, and watch as Pro/MECHANICA automatically checks for errors, converges to an accurate solution and generates convergence information for verification. See Figure 1 and 2.

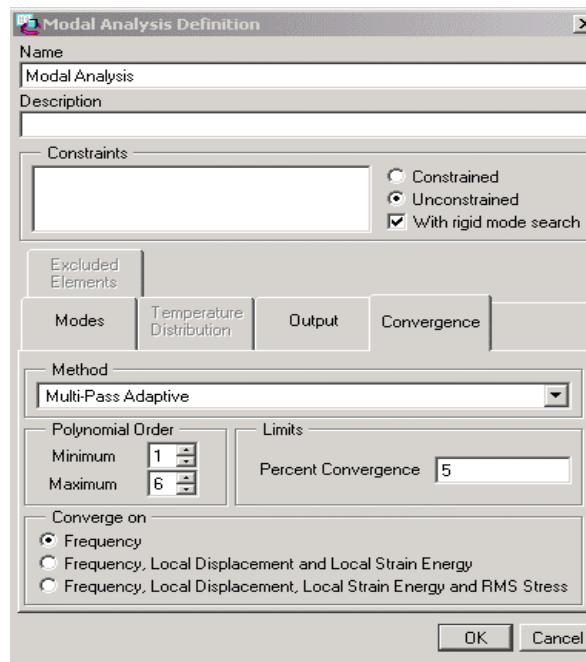


Figure 1.0

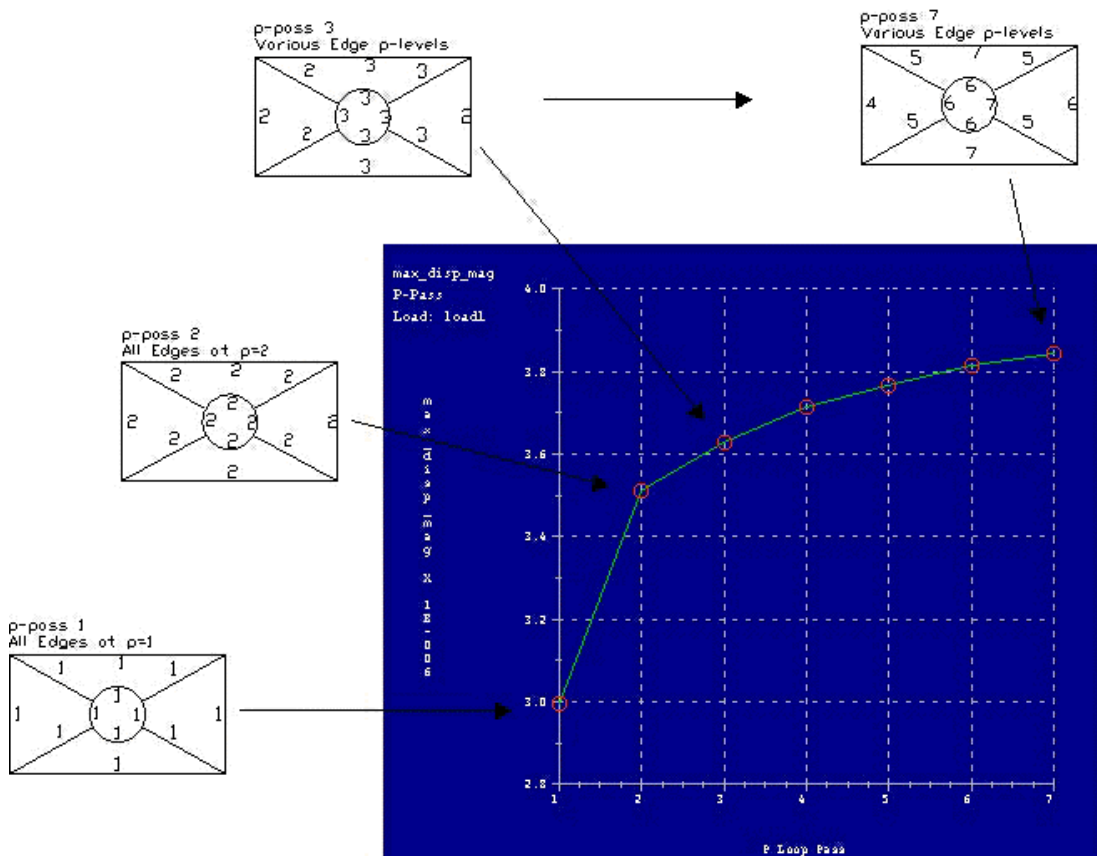


Figure 2.0

- Select one or more sensitivity parameters to vary over a range and then review graphs of desired outputs as a function of that changing parameter
- Calculate local sensitivity results at a small offset from the nominal parameter value and visualize whether varying a certain parameter has a significant effect. See Figure 3.0

Sensitivity Studies

Early Insight into Functional Behaviour

- “What-if” Studies
 - Design Insight
- Evaluates Different Designs Automatically**
- Guides Engineers Towards Better Designs**

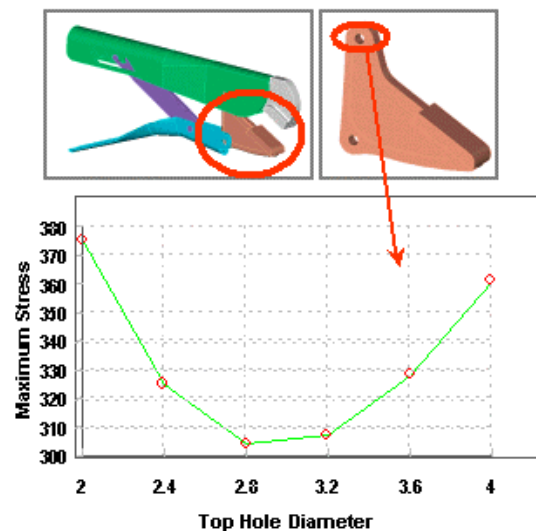


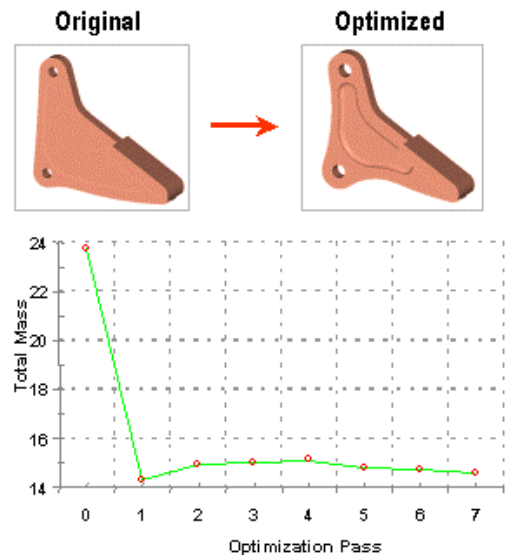
Figure 3.0

- Optimize designs by specifying multiple design parameters and a design goal in terms of cost, mass, displacement, stress, reactions, strain, frequency, or any other aspect of the design. For example, the mass of an assembly could be minimized while keeping stress, first modal frequency, and maximum displacement within limits

Optimization Studies

Driving Design Improvement

- **Goal** Automatically Establishes Best
- **Limits** Combination of Design Variables
- **Parameters** Produces Optimal Shape to Perform Intended Function



- Store and view displacement, stress, and strain over selected model entities as fringes, contours and query plots
- View vector plots of displacements and principal stresses as well as results on standard beam sections, animations of displacements, mode shapes, and optimization shape history
- Reserve and review results of displacement, velocity, acceleration, stress, and RMS quantities as fringes, contours, and query plots
- Evaluate graphs of any measure "at each step" in linear, semi-log, and logarithmic format Obtain summary values for all "single value" evaluation methods (minimum, maximum, maximum absolute value, and RMS)
- Finite element meshes for solid models and thin-walled solids automatically in the design environment or according to a designer's specifications. Interactive refinement capabilities enable designers to quickly evaluate different model configurations under a variety of environmental and geometric conditions. These features:
- Decrease analysis time

- Permit the investigation of more design variations
- Improved design flexibility and product quality
Once meshed, models can then be output to finite element analysis programs including ANSYS, COSMOS/M, MSC/NASTRAN, and MSC/PATRAN to:
- Leverage the high-end analysis capabilities of these best-in-class solvers without ever leaving the Pro/MECHANICA environment
- Enhance productivity as there is no longer any need to run multiple programs or to translate files between systems
- Post-processing capabilities provide a high level of functionality via an easy-to-use interface. Results are obtained in the context of the original part geometry, not as a meshed representative of the design. Engineers familiar with Pro/MECHANICA will be proficient immediately with the Post Processing functionality.

Benefits

- FULLY INTEGRATED WITH PRO/ENGINEER
 - EASY TO USE/DEVELOPED FOR DESIGN ENGINEERS
 - SENSISTIVITY AND OPTIMIZATION STUDIES
 - EARLY INSIGHT AND IMPROVEMENT OF DESIGNS
 - REDUCES RELIANCE ON PROTOTYPES
 - MINIMIZES COSTLY OVERDESIGN
-
-

Tip of the Week: **Creating and using Rigid Connections**

Overview

A rigid connection connects geometric entities, such as surfaces, curves and points, so that they remain rigidly connected during an analysis. When you connect entities in this way,

they move together as if part of a single rigid body

they do not deform, but the rigid body can move as a whole

Because Pro/MECHANICA uses linear constraint equations to enforce the rigid rotations, rather than equations with sines and cosines, you should use rigid connections only for small rotation angles of rigidly connected entities. Use rigid connections in this way, even if you intend to use them in a large deformation analysis.

In both integrated and independent mode:

Pro/MECHANICA supports rigid connections for 3D models only.

You can create, edit, and delete rigid connections.

You can turn rigid connection visibility on and off:

In integrated mode, select or deselect Rigid Connections on the Visibilities dialog (View>Simulation Display>Visibilities).

In independent mode, hide or show Rigid Connections on the Hide Entities pulldown (Display>Hide Entities). Or, you can use Display>Master Visibilities, and toggle Rigid Conns on or off.

In integrated mode, you can additionally use rigid connections to connect two or more parts at selected surfaces.

Details About Rigid Connections

Introduction

Rigid Connections were introduced with Pro/MECHANICA 2000i2. A Rigid Connection connects geometric entities, such as surfaces, curves and points, so that they remain rigidly connected during an analysis. When entities are connected in this way, they move together as if part of a single rigid body and they do not deform.

One application of this functionality is for a shell model, where the mid-compressed planes cannot be connected without modifying the geometry.

Procedure

1. The simple model of a support plate shown in Figure 1 will be used to illustrate how Rigid Connections are created in Pro/MECHANICA Integrated mode.

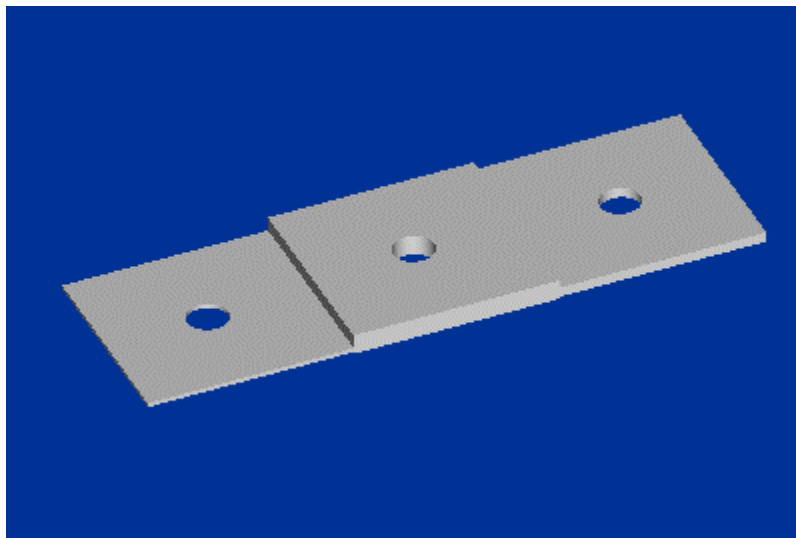


Figure 1

Begin by creating the model shown in Figure 2 in the unit system [mm N s]. The plate is extruded to a depth of 100 mm.

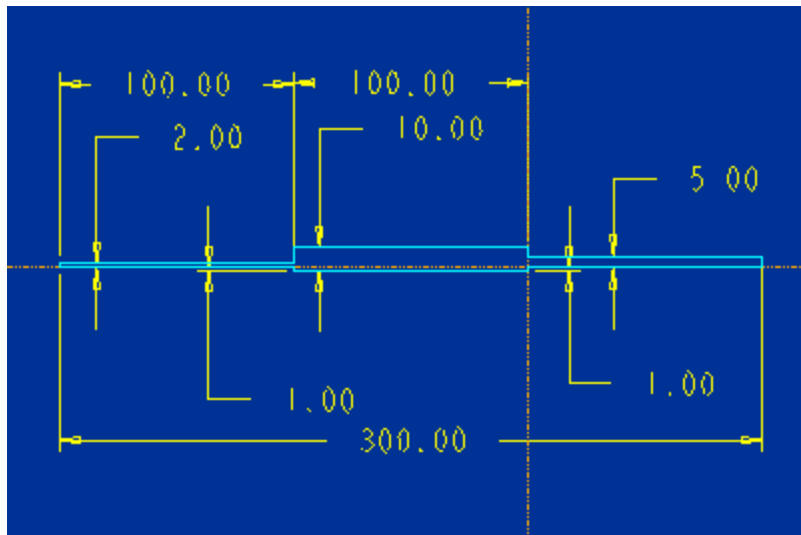


Figure 2

The three sections on this plate all have different thickness and since there is no common midplane, the resulting shell compressed model shown below has gaps between the shell compressed planes, see Figure 3. This makes it impossible to analyze this part as a shell model when compressing each single section to its own midplane. Alternate techniques in earlier releases of Pro/MECHANICA would have been to modify the sections slightly and to compress all sections to the same surface.

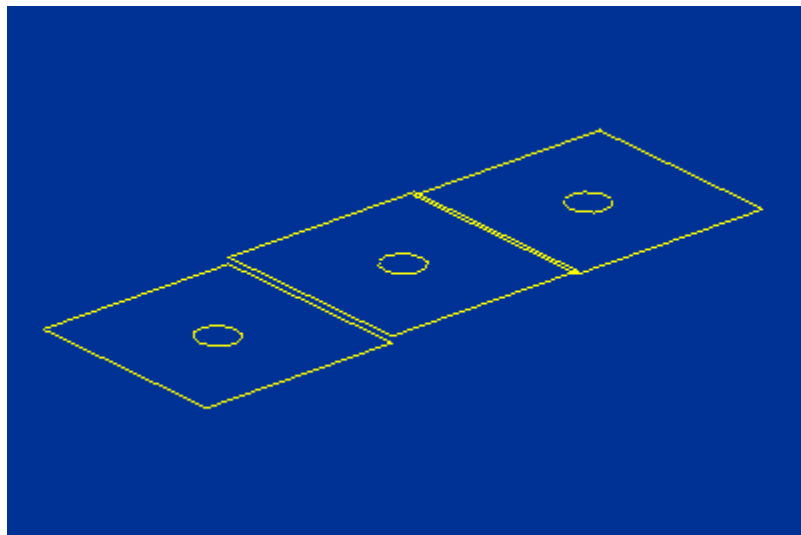


Figure 3

In this model, we would like the ends of the different mid compressed planes to have the same deformation and rotation.

2. Since this model is made of one single part, we have no separate surfaces on the ends of each plate to connect to each other. Therefore, we first need to create a Volume Region in

order to split the part into three separate volumes, enabling us to choose the end sections of each plate.

The procedure for creating a Volume Region is very similar to creating a protrusion or cut in Pro/ENGINEER. Select **Model, Features, Volume Region, Create** and then, for this model, select **Extrude** as type. Select one of the sides of the part as Sketching plane and sketch a box outlining the middle portion of the model, see Figure 4. Extrude the section up to the opposite surface. This will effectively split the part into three volumes.

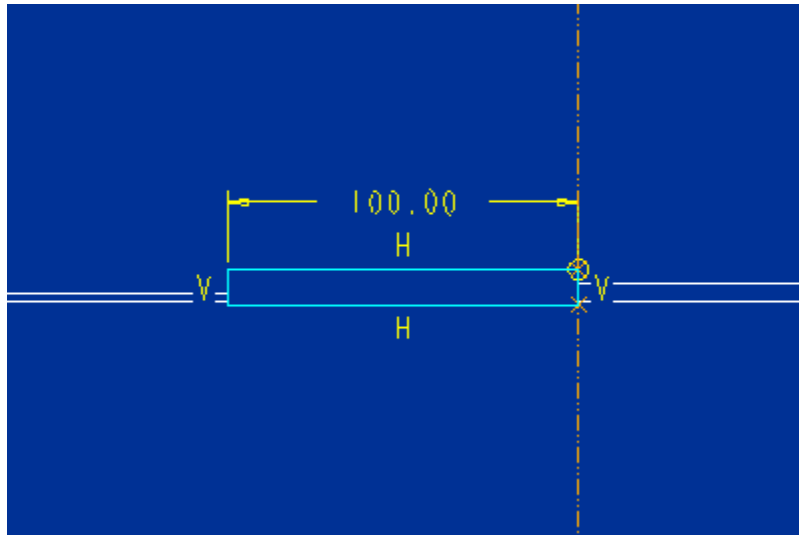


Figure 4

-
3. Now we can create the Rigid Connection. Then menu picks are **Model, Idealizations, Rigid Connections**. This will bring up the Rigid Connection definition window, see Figure 5. It allows giving a separate name to each Rigid Connection. Select the arrow to choose which entities that should be connected. Geometric entities, such as surfaces, curves and points may be chosen.

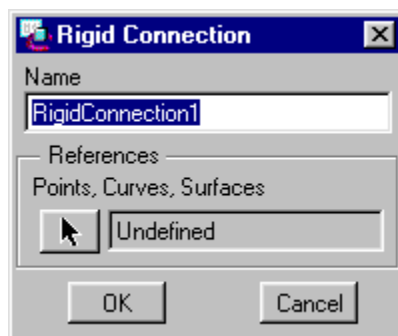


Figure 5

After selecting the arrow choose **Query Select** from the Menu Manager to select the two surfaces at the end of the Volume Region, see Figure 6 and 7 below. After the two surfaces have been selected, end the procedure by selecting **Done** from the Menu Manager.

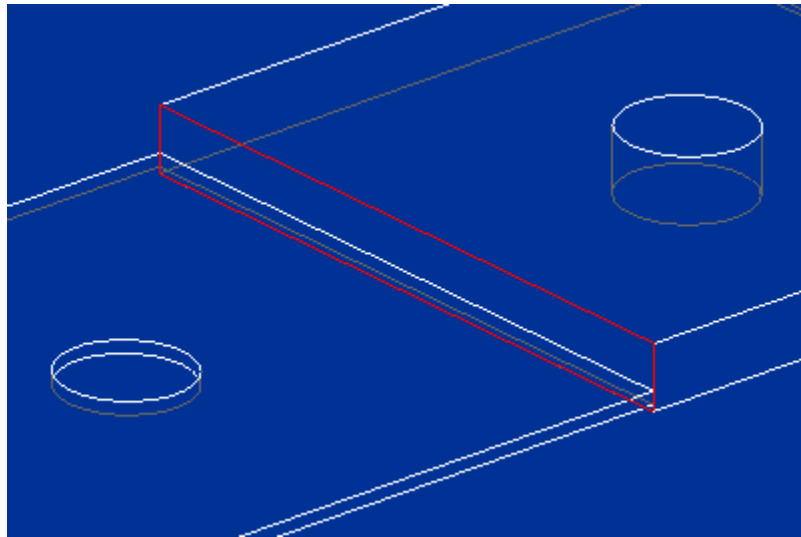


Figure 6

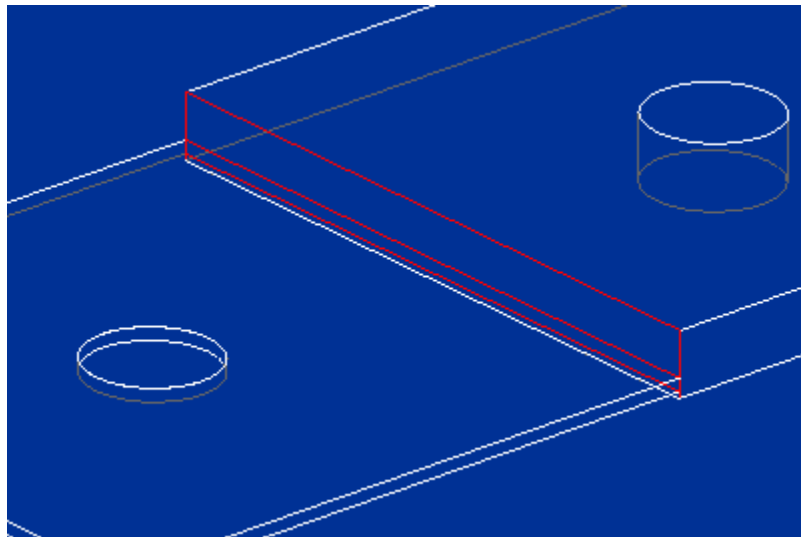


Figure 7

Repeat the procedure for the other end of the middle section. Once the Rigid Connections have been created successfully, the model should look like in Figure 8 with a 'box like' symbol for each entity in the Rigid Connection. The visibilities of the Rigid Connections can be turned on and off by modifying the Simulation Display Visibilities. The menu picks (from the main window) are **View, Simulation Display, Visibilities**.

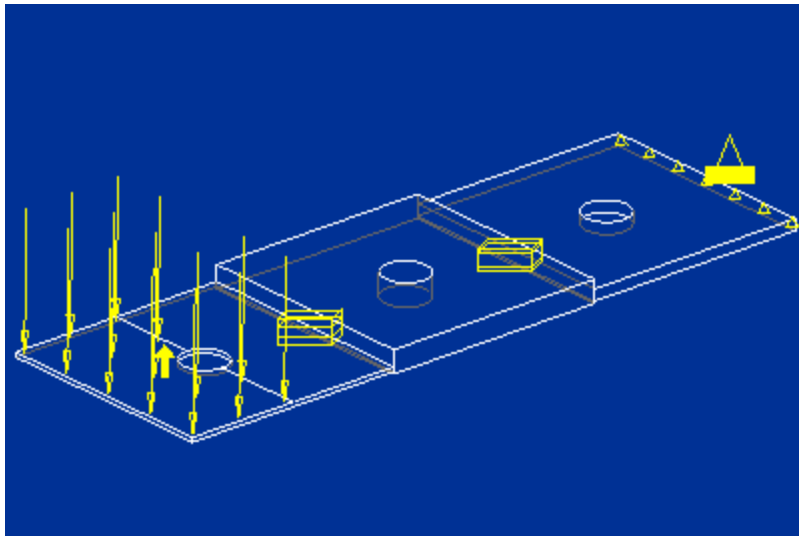


Figure 8

-
4. Now apply Loads, Constraints and assign a material for the part, in this case STEEL has been used. If not already done, define the shell idealization for the model with the menu picks **Model, Idealizations, Shells, Midsurfaces, Create**.

In this model, the top surface of the left plate was split in half using a Surface Region and a total load of the magnitude 100 N was applied normal to the surface region

The constraint is applied to the end surface. Starting with release 2000i2 it is no longer necessary to apply constraints, or loads, on an edge for a shell-compressed model. After the compression, the load and/or constraint applied to the surface will automatically be projected on the edge of the midplane model. See Figure 9 for the complete model.

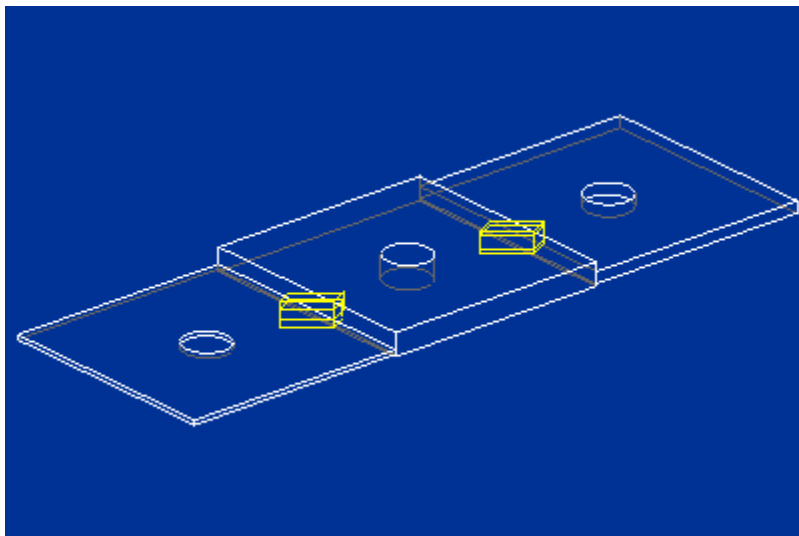


Figure 9

5. The fringe plots below shows a comparison between the maximum displacements for the shell-compressed model with Rigid Connections, Figure 10, and the corresponding solid model for a Single Pass Static analysis (SPA), Figure 11. The results are almost identical, as expected for this type of model where the Shell Idealization is a good approximation. (The thickness of a body modeled with one or more shells should be significantly smaller than the length of any of the body's other dimensions and radii of curvature.)

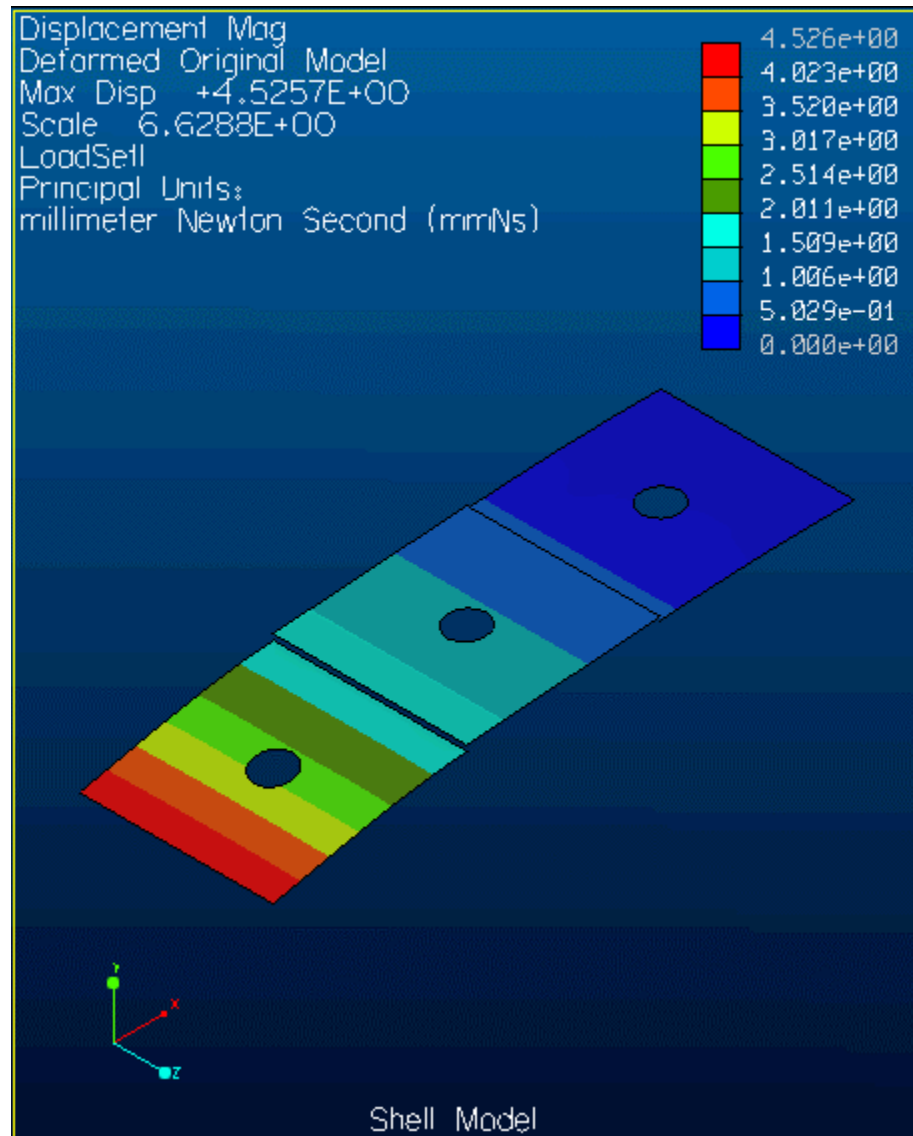


Figure 10

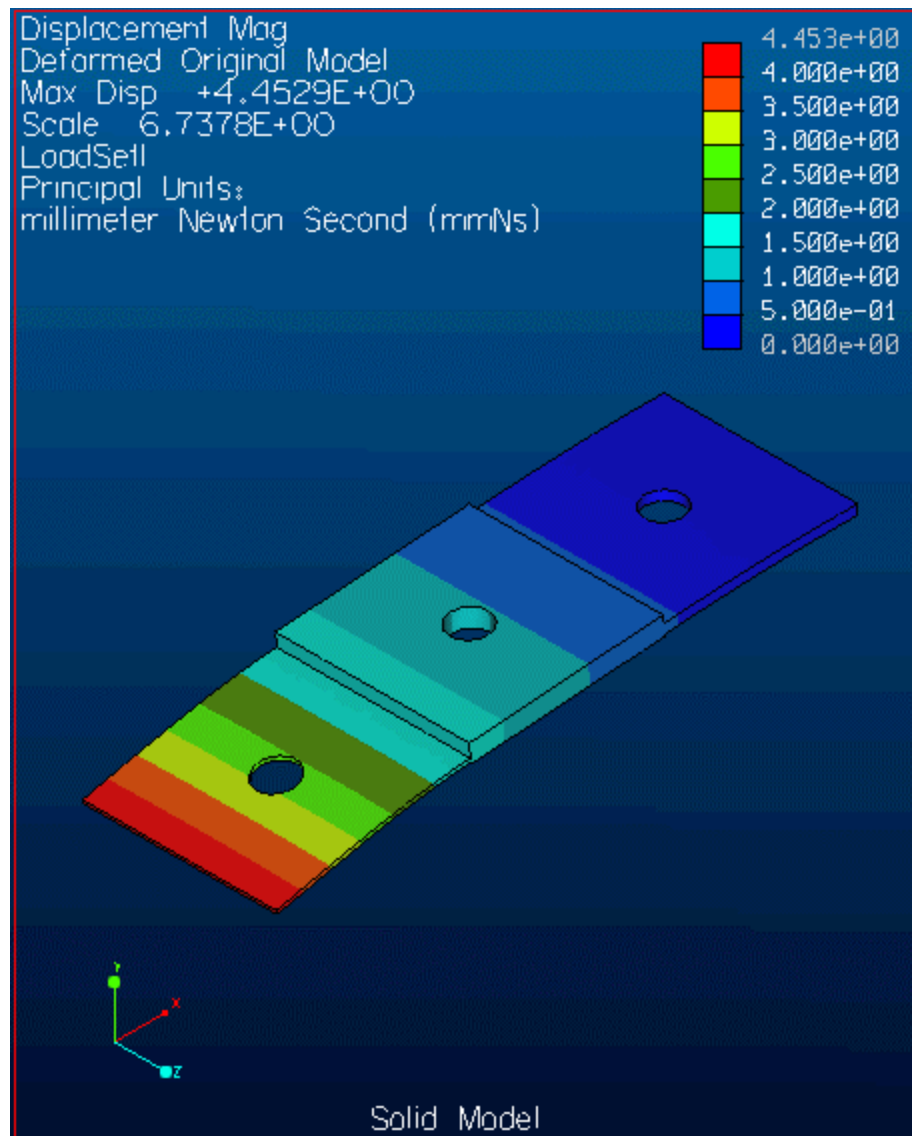


Figure 11

6. Note the following when using Rigid Connections:

Because Pro/MECHANICA uses linear constraint equations to enforce the rigid rotations, rather than equations with sines and cosines, Rigid Connections should only be used for small rotation angles of rigidly connected entities. Use Rigid Connections in this way, even if they are to be used in a large deformation analysis.

Pro/MECHANICA supports Rigid Connections for 3D models only.

In integrated mode, Rigid Connections can additionally be used to connect two or more parts at selected surfaces.

In integrated mode, Pro/MECHANICA assigns a name to each Rigid Connection - in independent mode, it does not.

Additional Information

[TITLE : Creating Volume Regions](#)

[TITLE : Creating Constant Thickness Shell Pairing in Integrated Mode](#)

[Schedule an Online Demonstration](#)

Upcoming Events and Training Class Schedules

Training Classes: Please visit the PTC website for the latest training class schedule and locations.

[Click here for Training Classes](#)
