MSC **Software**®

Case Study: **Framework Construction**

Static Simulation for Framework Construction in Power Plant Using MSC Apex

Overview

Many structures in plant engineering are characterized as thin-walled. The Finite Element Method (FEM) is a common method used to assess the performance of such thin structures. Creating a FEM model of a thin structure involves midsurfacing models and meshing with shell elements. However, the process for creating FEM models is time consuming often requiring hours and days. The use of MSC Apex can help produce midsurface models significantly faster than with other traditional CAE pre/post processors. In addition to FEM creation, MSC Apex can be used to perform strength analysis.

A deformation analysis is performed on a framework used at a power plant site.
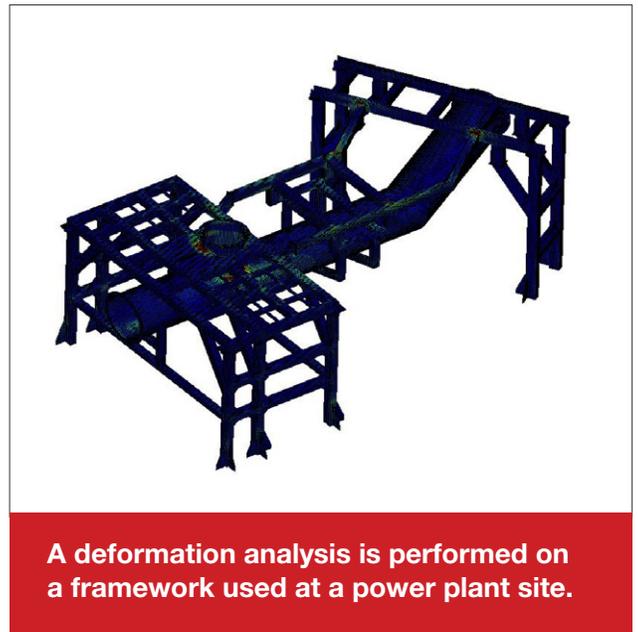
> **"The framework model has over 400 parts. In a traditional pre/post processor, 1-2 days of work could have been required to create the FEM. MSC Apex on the other hand required 4 hours of work before the FEM was completed."**

Challenge
Design and Simulation

MSC Apex accelerates the design process.

The original model in Figure 1 shows that there are many parts with thin-walled structures. By using the "midsurface" function of MSC Apex, midsurfaces of the whole model can be created in a few seconds. Furthermore with the help of "auto-extend surfaces", edges are stitched automatically. A few remaining edges that are not caught by "auto-extend" can be connected by manual edge drag. Then the midsurface model is meshed with shell elements.

For thin-walled structures, a shell mesh produces more exact results with much less elements, compared to a solid mesh. Figure 2 shows the shell mesh. In order to display the meshing details, one part of the top is enlarged and shown on the right side to the whole meshed model.

Then the material properties are defined; a linear static analysis requires only Young's Modulus and Poisson's ratio. The wall thickness can be assigned manually, or found automatically from the original solid model. For boundary conditions of this case, the bottom of ten support pillars are treated as fully fixed constraints. And considering the cooling water pressure of the power plant, 0.74 MPa is applied to the tube. All the boundary conditions are shown in Figure 3. Then the simulation is run.
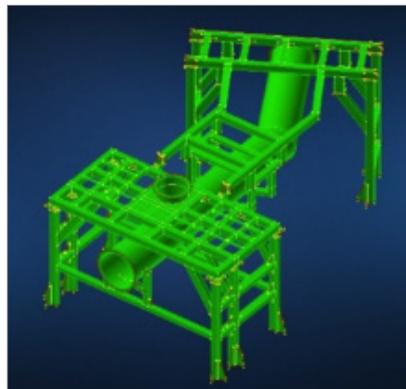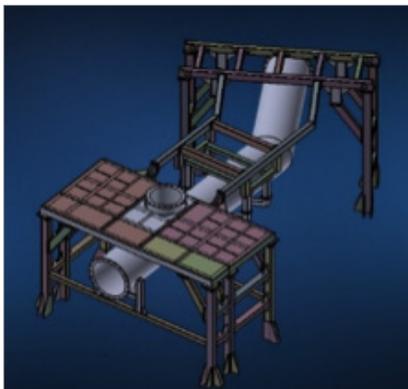
MSC Apex Structures uses and integrated solver based on MSC Nastran technology.

Results

Figure 4 shows the deformation results. On the left is the deformation in true scale. The undeformed geometry is shown in blue, while the deformed geometry is marked by red color. As the deformation is very small compared

Figure 1: Left: original solid model, right: midsurface model

Figure 2: Meshing
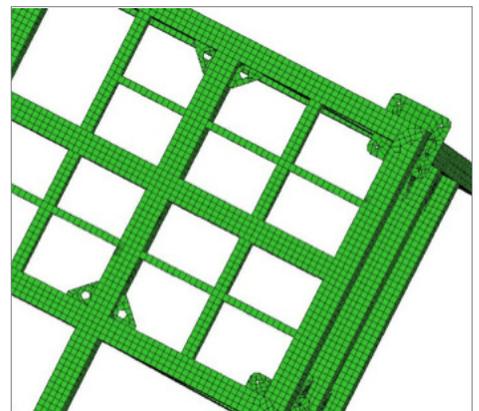
to the model dimensions, the deformation would not be visible in true scaling. So in the picture, the deformation is exaggerated as 5% of the largest model dimension. It is obvious that the largest displacement appears on the left end of the tube.

Figure 5 shows the von Mises stresses . The locations where the largest stress appears, are marked by dark red. It is obvious that these dangerous locations are usually in the joint parts. So if optimization design is carried out, these joints need close attention. More detailed modeling, e.g. by a locally finer mesh, could be required. MSC Nastran allows a subsequent fatigue analysis, or a wall thickness optimization.
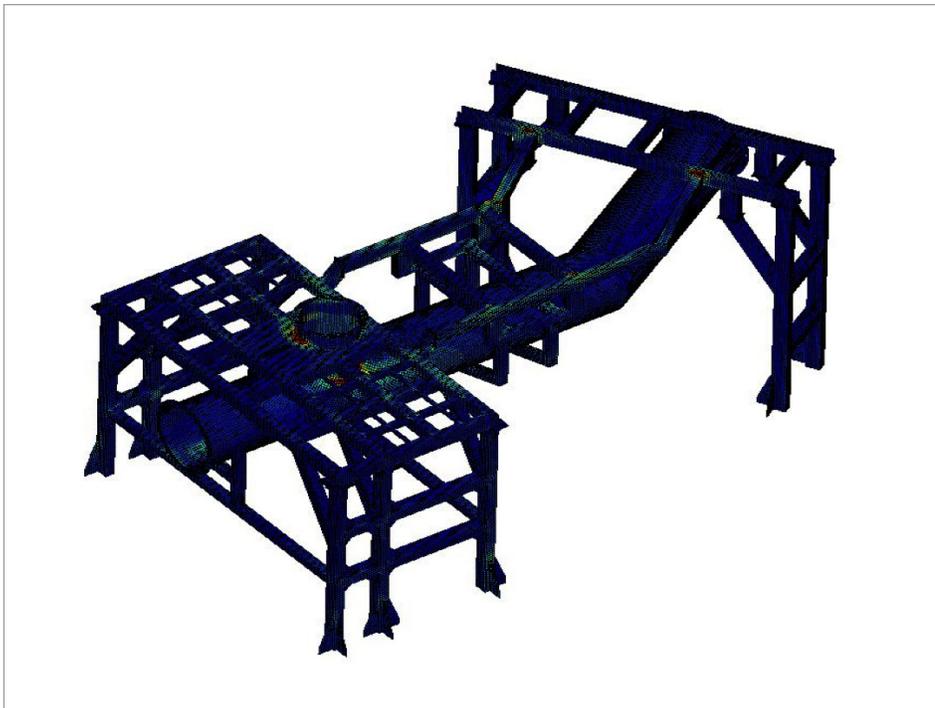

Figure 3: Constraints
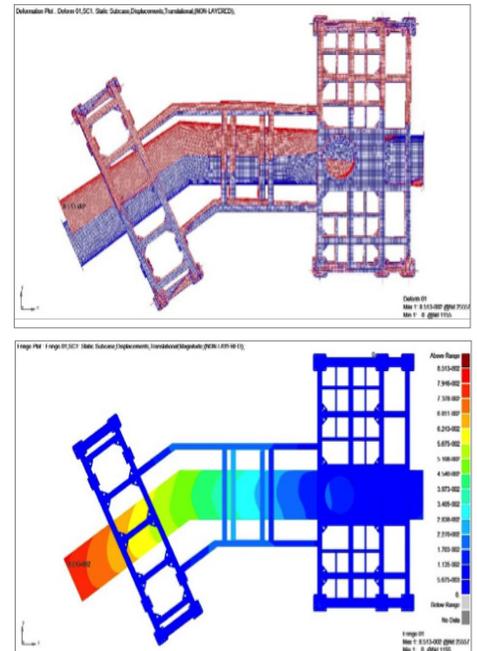
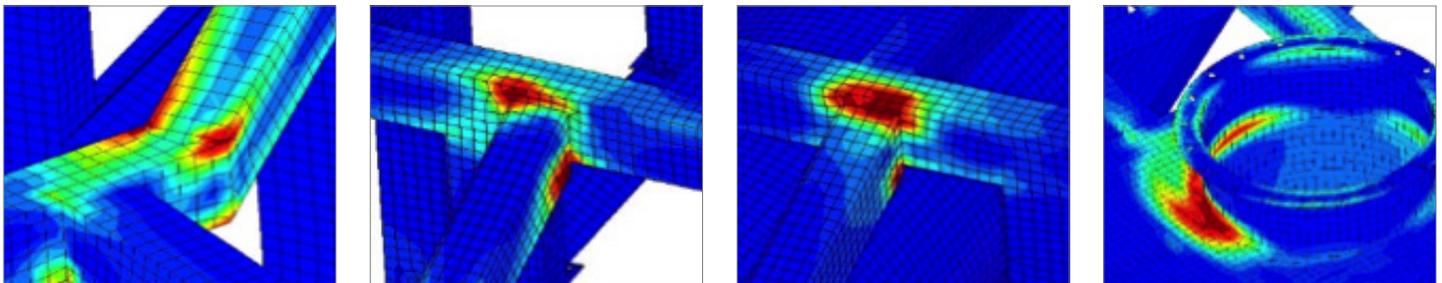Figure 4: shows the deformation as a color plot (fringe plot).

Figure 5. von Mises stresses

For more information on MSC Apex and for additional Case Studies, please visit www.mscapex.com

FC*2015JUN*CS