Not All Stresses Are Reported Equal

Different stress and strain results in ANSYS Mechanical versus Mechanical APDL may indicate need for a refined mesh.

By Ted Harris, CAE Support and Training Manager, Phoenix Analysis & Design Technologies, Inc., Tempe, U.S.A.

Among the features offered by ANSYS structural analysis tools are three different methods for calculating and reporting stress and strain results. However, viewing such results using ANSYS Mechanical software in the ANSYS Workbench platform compared to ANSYS Mechanical APDL (MAPDL) can lead to uncertainties about which displayed values are more accurate.

As a typical example, residents of warm climates — such as the southwestern United States — deploy portable expanding sunshades to keep cool outside. Unfortunately, high winds from weather phenomena, such as dust storms, can cause the shade's metal support members to bend and break. Therefore, understanding the structural limits is of interest to users in such locales.

A solid geometry representation of a typical portion of the frame consists of two rectangular hollow members, pinned to each other at the center, with pins at each end that, in the full structure, would be attached to additional components. For simplicity, the pins are fixed on the right side to the ground, while those on the left side have an upward-bearing load applied to the upper pin and a downward-bearing load applied to the lower pin. These loads tend to cause the members to bend at the central pin. The bearing loads represent the effect of a strong gust of wind hitting the fabric canopy above the frame, with the load eventually reacting through the frame to stakes that attach the frame to the ground at the bottom. The applied load is large enough to cause significant plastic deformation, similar to what might be experienced when one of these structures is subjected to high winds.

At issue here is the initial run with a very coarse mesh. When viewing the von Mises stress results in Mechanical within Workbench and then comparing them with the results obtained from the same results file in /POST1 in MAPDL, there is a difference of 4,700 psi between the peak values.
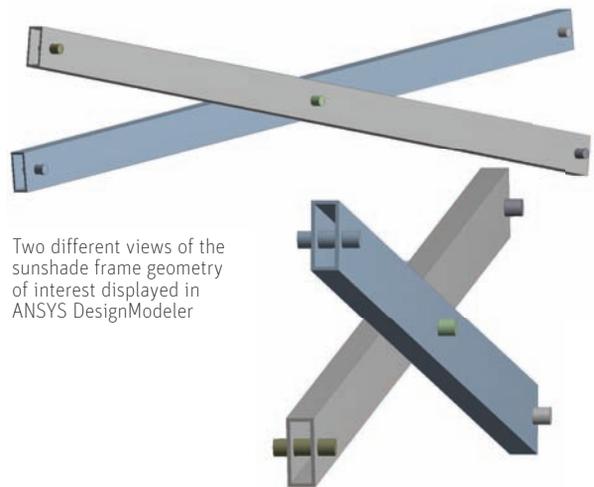
Two different views of the sunshade frame geometry of interest displayed in ANSYS DesignModeler

The reason for this difference has to do with how stresses are calculated.

In /POST1 in Mechanical APDL, the original graphics display system is known as Full Graphics. In 1994, ANSYS developed a newer graphics display system for MAPDL known as PowerGraphics. Both display systems are available to use, but there are several differences between them that affect how the results are presented.

By default, MAPDL uses PowerGraphics, which looks at results only on the exterior surfaces of the model. Full Graphics, on the other hand, includes interior elements in addition to the exterior surfaces when displaying results plots. Additionally, with PowerGraphics the number of element facets displayed per element edge can be varied with the /efacet command. The standard is one facet per edge, although for midside-noded elements it can be increased to two or four. Full Graphics, however, is limited to one facet per element edge. Mechanical in Workbench uses an algorithm for which the results tend to compare more favorably with Full Graphics, although it displays with

Values of peak von Mises stresses for a coarse mesh displayed with ANSYS Mechanical in Workbench (left) and with Mechanical APDL /POST1 (right). The upper support beam is shown to be bent by the applied load, simulating the effect of high winds on the sunshade.

two facets per element edge. A further test that can be done within MAPDL is to plot nodal (averaged) vs. element (unaveraged) stresses to check for large variations.

In evaluating the results from the three methods, they may all be considered correct but simply displayed differently. However, the difference in displayed results can be used as a guideline for mesh density and presence of singularities. If there is a significant difference between PowerGraphics and Full Graphics results in MAPDL, this usually indicates that the mesh is too coarse. As the mesh is refined, the difference between the two calculations should decrease. In the sunshade frame model, the differences in von Mises stresses and plastic strain results do decrease with a finer mesh.
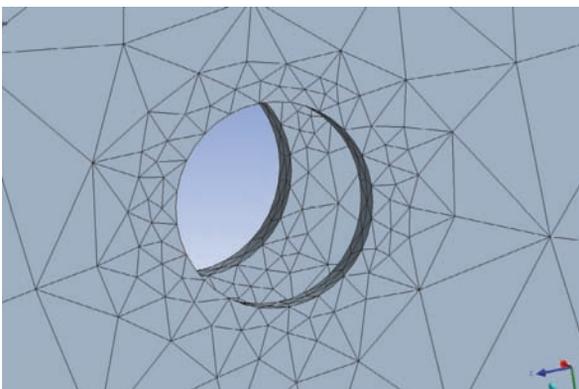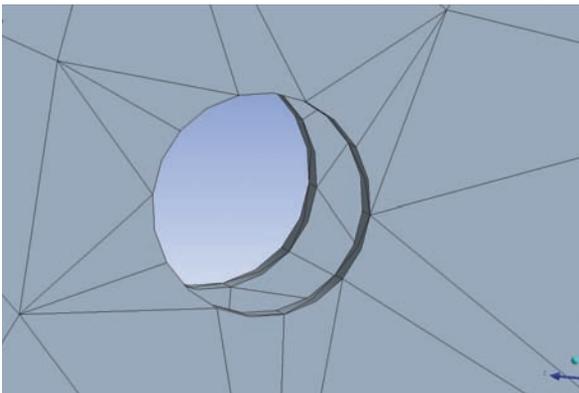
ANSYS 13.0 Equivalent Stress Result Comparison

Post-processing Algorithm	Coarse Mesh	%Diff	Fine Mesh	%Diff
Workbench Mechanical	79219	–	129560	–
MAPDL PowerGraphics	83873	6%	128474	-1%
MAPDL Full Graphics	79219	0%	129560	0%

ANSYS 13.0 Equivalent Plastic Strain Result Comparison

Post-processing Algorithm	Coarse Mesh	%Diff	Fine Mesh	%Diff
Workbench Mechanical	0.1998	–	0.444	–
MAPDL PowerGraphics	0.2231	12%	0.443	0%
MAPDL Full Graphics	0.2039	2%	0.448	1%

Results comparisons

In ANSYS Mechanical 13.0 in Workbench, averaged and unaveraged stress and strain results can be plotted as another check on the results. This choice is made in the details view for a given plot. A significant difference between averaged and unaveraged quantities indicates that mesh refinement is needed.

The conclusion is that if unacceptable differences in stress or strain results are observed using different results calculation methods, it likely means that the mesh, at least in the area of interest, is too coarse. It is, therefore, advisable to add mesh refinement and check the results again. In Mechanical in Workbench, adding a convergence item to a scoped result plot can at least partially automate this process. Take care, however, to make sure that singularities are not included in the desired convergence region.

Regarding the mesh densities used in this example, the coarse mesh had an element size of at least 0.05 inches on the member hole at the high stress–strain location, while the fine mesh had an element size of 0.025 inches on the same hole. Another way to look at the mesh refinement is that the coarse model had 20 elements on the hole of interest while the fine mesh had 104 elements on the same hole. Clearly, the coarse mesh in this example was too coarse for engineering purposes, but it was selected to ensure that the effect of different results calculation methods was significant and observable. ■

Phoenix Analysis & Design Technologies, Inc. is an ANSYS channel partner serving the U.S. Southwest. This article originally appeared as an item in *The Focus* newsletter.

Examples of coarse mesh (top) and fine mesh (bottom) used to predict the stress and strain quantities for sunshade support members