

Workbench Mechanical: Averaged vs. Unaveraged Contour Results



CFD Analysis of Convergent-Divergent and Contour Results



Have you wondered what other tools may be useful in determining the quality of your results? In addition to the convergence tools, you can also use contour results to help ensure quality results. Normally, contour results in the Mechanical application are displayed as averaged results. Each element calculates a unique, elemental nodal stress which is typically different than the other elements connected to that node. Averaged contours will average elemental nodal results across element and geometric discontinuities but will never average results across bodies. Some results can also display as unaveraged contours.

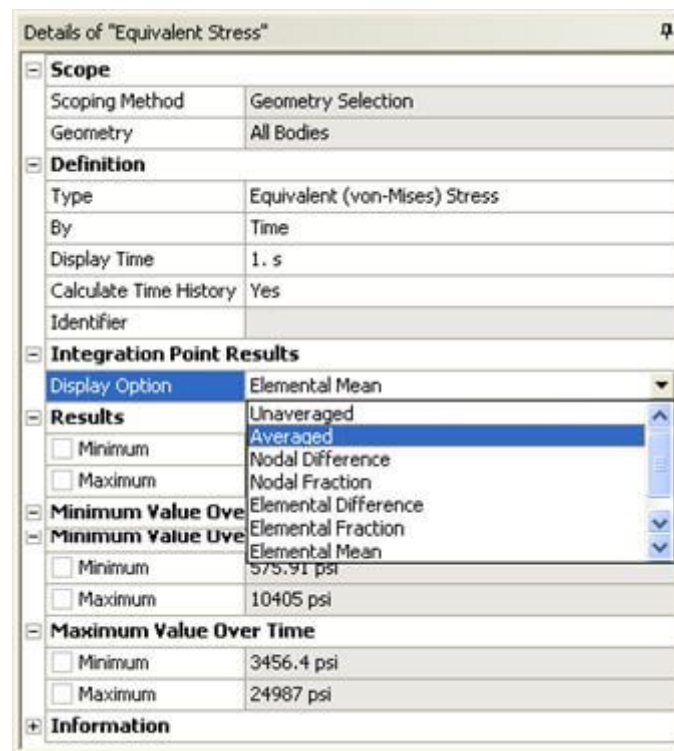
How to Change the Averaging on Contour Results (by Settling the Display Option Field)

You can change the averaging on contour results by setting the Display Option field, shown in the figure below, to one of the following:

- Unaveraged: Displays unaveraged results.
- Averaged: Displays averaged results.
- Nodal Difference: Computes the maximum difference between the unaveraged computed result (for example, total heat flux, equivalent stress) for all elements that share a particular node.

- Nodal Fraction: Computes the ratio of the nodal difference and the nodal average.
- Elemental Difference: Computes the maximum difference between the unaveraged computed result (for example, total heat flux, equivalent stress) for all nodes in an element, including midside nodes.
- Elemental Fraction: Computes the ratio of the elemental difference and the elemental average.
- Elemental Mean: Computes the elemental average from the averaged component results.

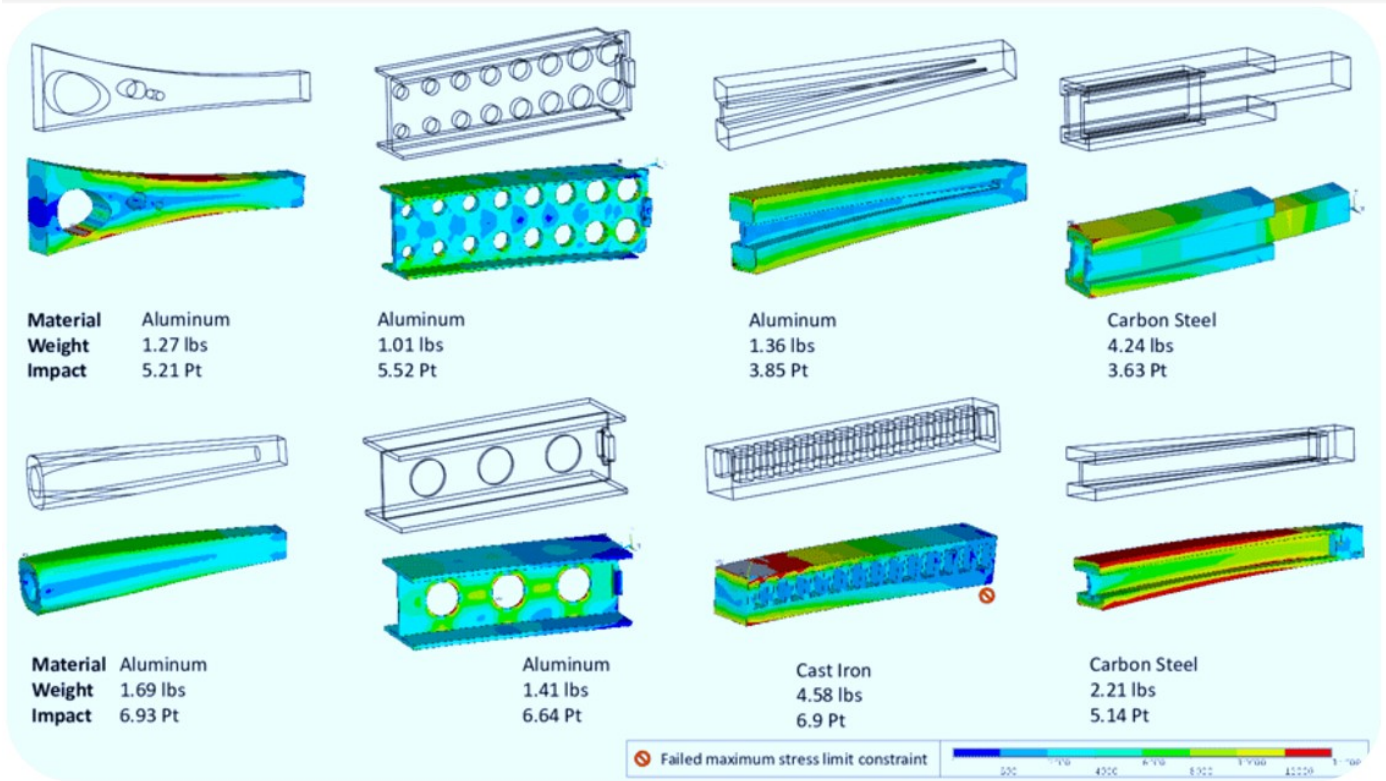
The Nodal Difference, Nodal Fraction, Elemental Difference, and Elemental Fraction aid in determining mesh quality. If there are large differences in results calculated by elements attached to shared nodes, then a refined mesh in that region may be necessary.



The Elemental Mean will create a checkerboard of results where each element will have one value. The behavior is similar to the ETABLE feature within Ansys Mechanical APDL. Results can be exported to Microsoft Excel and subsequent calculations, such as summations, can be performed.



Postprocessing | Contour Plots



Contour plots display the results of a single data set (time step, load step, or sub step) over the base model geometry. The range of values in the results set is divided into several subranges (nine by default), and each subrange is assigned a color. The colors are then mapped over the geometry to indicate the result values at each location in the model. Most results for continuum (e.g., PLANE and SOLID) elements can be displayed as contour plots. For beam and shell elements, it may be required to issue the **/ESHAPE** command before creating a contour plot or to plot the outcomes using element tables instead.

Contour plots of the nodal solution can be created using the GUI path:

- **Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu**

or

- Using the **PLNSOL**

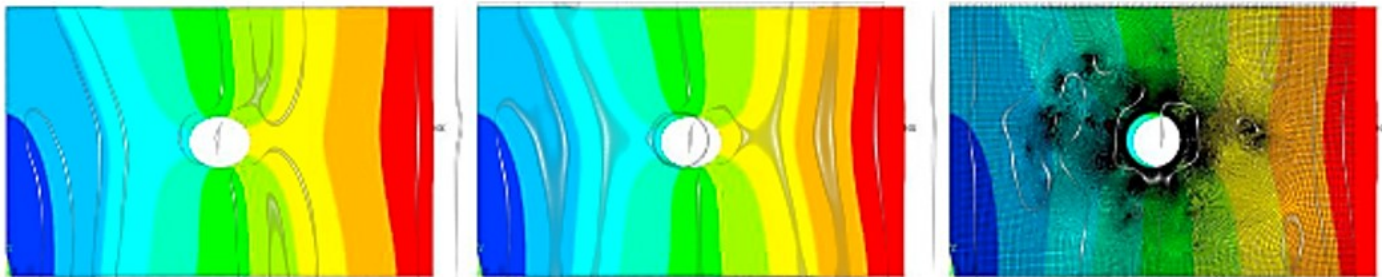
Contour plots of the element solution can be created using the GUI path:

- **Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu**

or

- Using the **PLESOL**

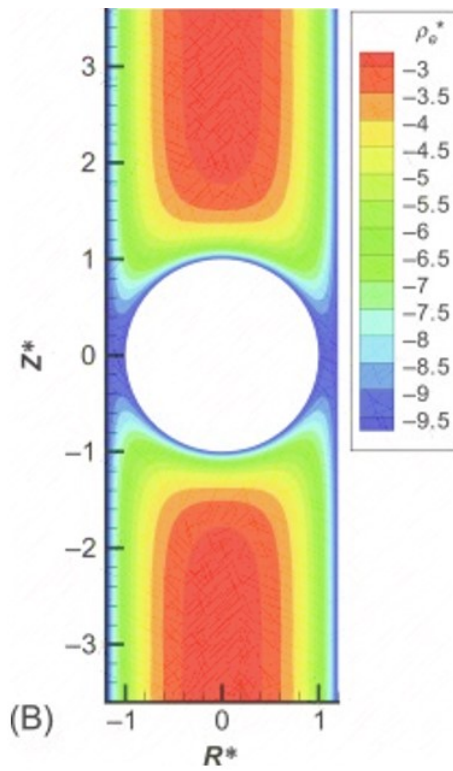
There are three important display options for contour plots: the non-displaced shape key, the scale factor, and the contour scale. The displaced shape option controls if and how the deformed model is shown in comparison to the undeformed (original) geometry. The “Deformed shape only” is the default and provides no basis for comparison (Figure 1, left). The “Deformed shape with undeformed edge” option overlays the contour plot on an outline of the original model (Figure 1, center). The “Deformed shape with undeformed model” option overlays the contour plot on the original finite element model (Figure 1, right).



Displacement Vector Sum from Contour

The scale factor determines if and how the deformed model is displayed in comparison to its as-calculated state. The deformations in most finite element models are relatively small and would be difficult or impossible to view if they were plotted as calculated (at their true scale). Therefore, ANSYS automatically scales the maximum displacement to 5% of the model dimension. The resulting scale factor is shown in grey to the right of the Scale Factor drop down menu in the Contour Solution Data dialog boxes.

Contour Plot Controls Showing the Auto Calculated Scale Factor



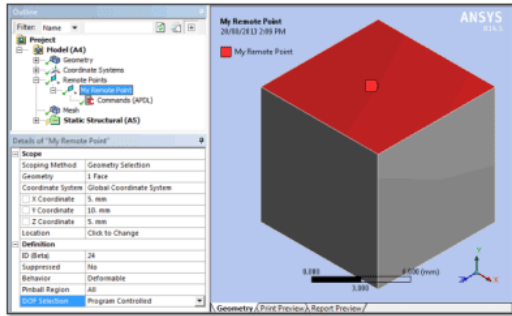
Finally, you can control the color contour display using the menus accessible via the GUI path:

- **Utility Menu>PlotCtrls>Style>Contours.**

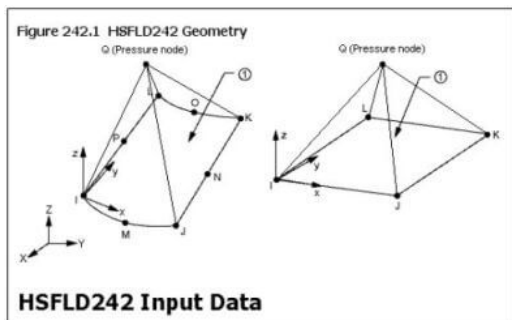
Here, it is possible to specify the number of contour intervals to use and to set the minimum and maximum contour values. Using the same number of contour intervals and the same minimum and maximum values on multiple plots allows meaningful side-by-side comparison of results from the same or similar models. This is a very powerful technique that will help ensure integrity and accuracy.

📊 Post Views: 3,101

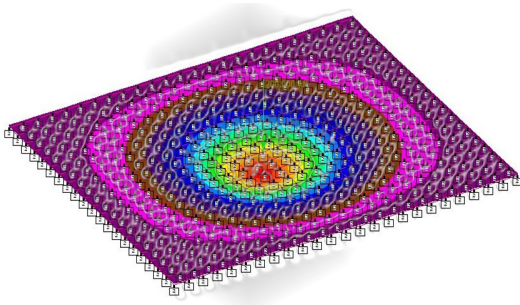
Most Recent Tips & Tricks



Measuring Face Translation and Rotation in Mechanical



Contained Fluid in Mechanical FEA Model | Working with HSFLD242 Elements



Normal and Tangential Elastic Foundations in Mechanical



Transient Thermal Analysis with Non-Physical Temperature Results

[View All Tips & Tricks](#)