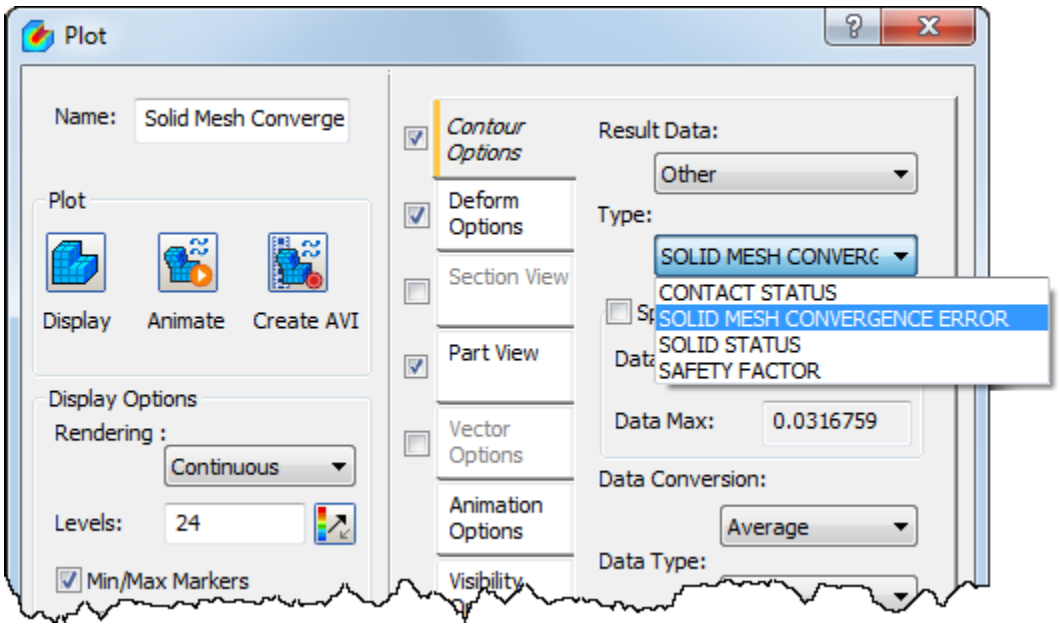
Autodesk Nastran In-CAD "Mesh Convergence Error"

Simulation (FEA) does not force the stress to be continuous from one element to the next. When a node is shared by multiple elements, the stress computed in each element at the node will be different. As the mesh is refined and the stresses converge toward a solution, the difference will diminish.

These differences in stress are used to estimate the error in the solution. Nastran In-CAD shows this calculation using the contour "Other > Solid Mesh Convergence" or "Other > Shell Max Mesh Convergence Error Bottom/Top" for solid and shell elements, respectively.



The result displayed for the "Mesh Convergence Error", e_i^*, is calculated as follow for each node:

$$e_i^* = \eta \frac{e_i}{|e_i|_{max}}$$

where

- e_i is the root mean square of the error at node i and is calculated as follows:

$$e_i = \sqrt{\frac{1}{N}\sum_{n=1}^{N}(\sigma_i^n - \bar{\sigma}_i)^2}$$

$$\bar{\sigma}_i = mean\ stress = \frac{1}{N}\sum_{n=1}^{N}\sigma_i^n$$

  - i is the index of individual nodes in the model, i=1 to I nodes
  - n is the index of volume elements attached to node i, n=1 to N elements
  - σ_i^n is the von Mises stress value at node i due to element n
- $|\ e_i\ |_{max}$ is the maximum value of e_i
- η is a relative stress error based on the entire model and the volume of the elements. Since $e_i = |\ e_i\ |_{max}$ at the point of maximum e_i^*, the value of η is equal to the largest value of the Mesh Convergence Error in the entire model. (The value of η is also listed in the .LOG file: search for the text "SOLID ELEMENT RELATIVE STRESS ERROR =".)

In other words, the Mesh Convergence Error is the root mean square error of the stress at a node, multiplied by the relative stress error η, and divided by the maximum root mean square error in the model.

Note that a low value for the Mesh Convergence Error does not guarantee that the stress results are accurate. It only indicates that the stress gradient is small. For example, the Mesh Convergence Error will be 0 in these circumstances regardless of the accuracy of the calculated stress:

1. At nodes that are shared by only one element.
2. At nodes where the results are symmetric.

Revision	Date	Change
0	Jan 18, 2017	Initial release