Webinar:
Recovering Stress Data with Simcenter Nastran

Andy Mouron, ATA Engineering

November 13th, 2019

13290 Evening Creek Drive S, Suite 250, San Diego CA 92128

(858) 480-2000

www.ata-e.com

ata-engineering

@ATAEngineering

ATA Provides High-Value Engineering Services With Expertise in Design, Analysis and Test

ATA Engineering helps to overcome product design challenges across a range of industries

Aerospace

Robotics & Controls

Themed Entertainment

Defense

Industrial & Mining Equipment

Consumer Products

ATA is a Value-Added Reseller for Siemens PLM Software

ATA offers training, free resources, and hotline support for a variety of Siemens products.

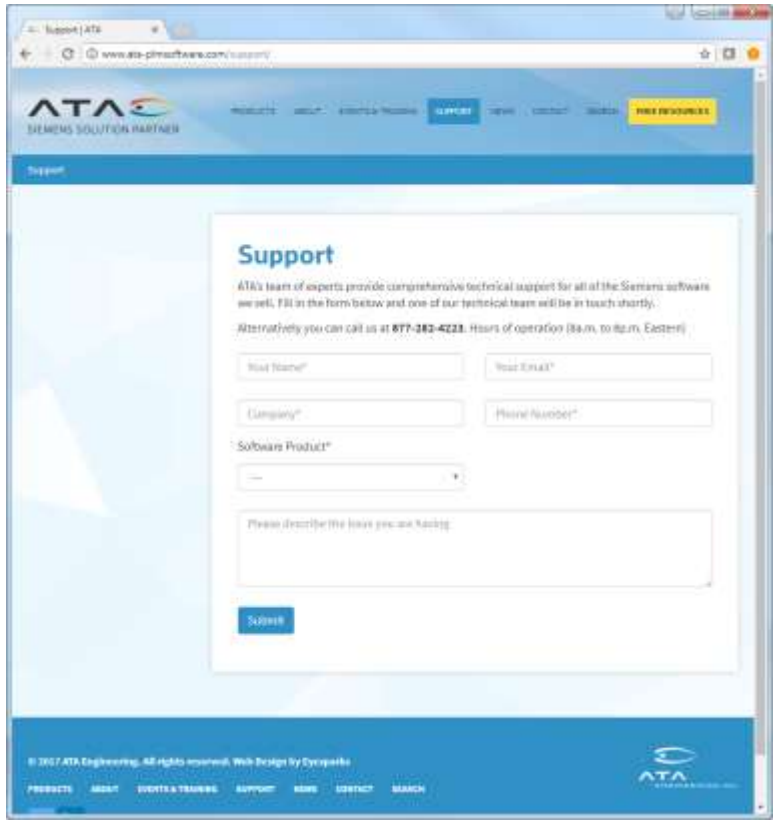
- Siemens product lines we support include:
  - STAR-CCM+
  - Femap
  - Simcenter Nastran (formerly NX Nastran)
  - Simcenter 3D
  - NX CAD & CAM
  - Teamcenter
  - Solid Edge
- Contact the hotline at 877-ATA-4CAE or http://ata-plmsoftware.com/support
- Developer of the official Simcenter Nastran training materials
- Preferred North American provider of Simcenter Nastran training
- Recognized as Smart Expert Partner with validated expertise in Femap and STAR-CCM+

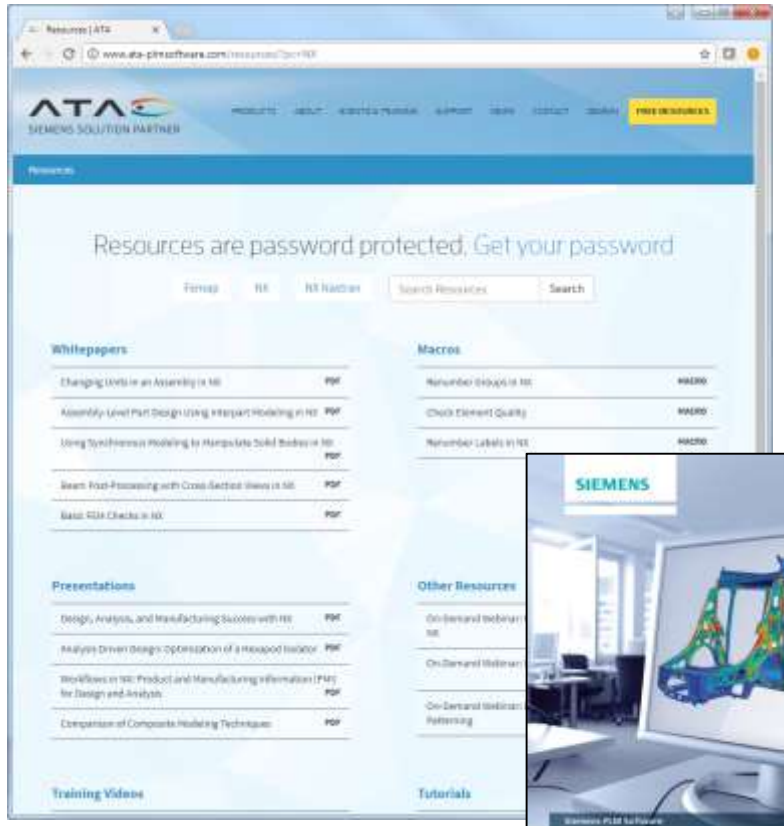
Solution Partner PLM | SIEMENS | Platinum Smart Expert Channel

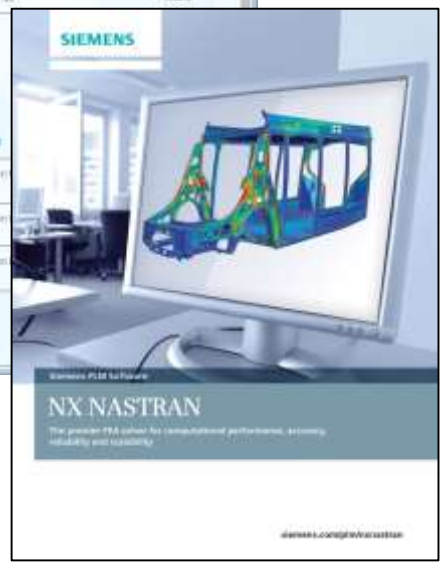
ATA ENGINEERING, INC.

Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com







ATA ENGINEERING, INC.

Webinar:
Recovering Stress Data with Simcenter Nastran

Andy Mouron, ATA Engineering

November 13th, 2019

13290 Evening Creek Drive S, Suite 250, San Diego CA 92128

(858) 480-2000

www.ata-e.com

ata-engineering

@ATAEngineering

Webinar Overview:
Recovering Stress Data with Simcenter Nastran

Purpose:

- Explain the difference between nodal and elemental stress results

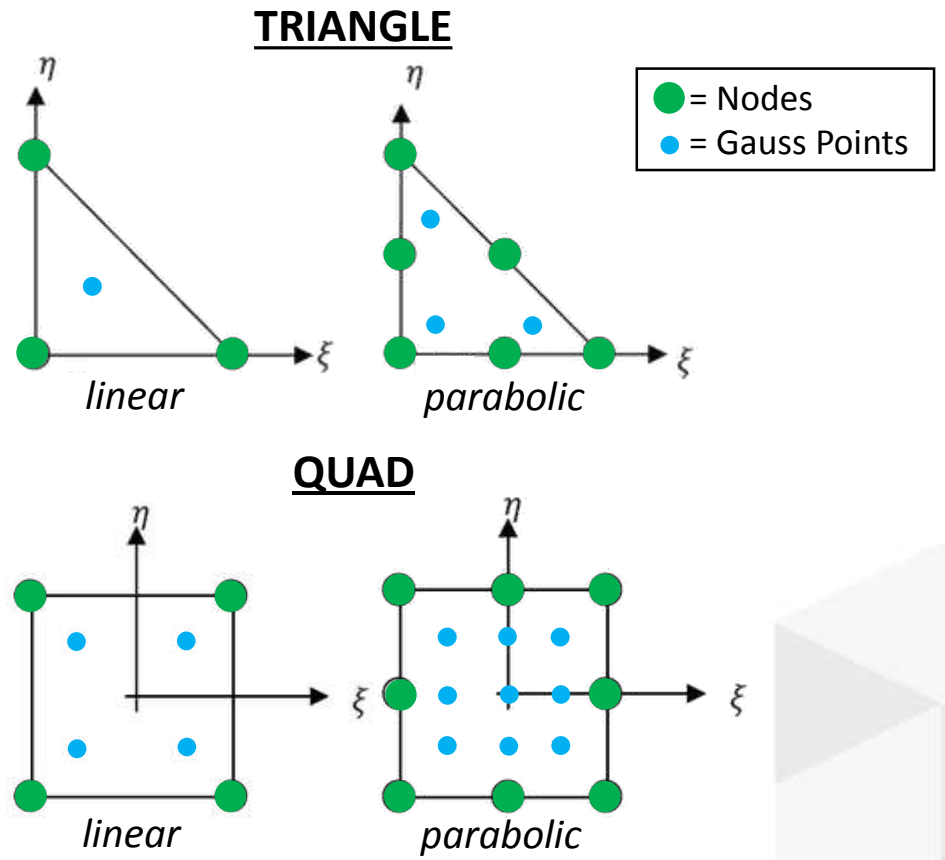
Motivation:

- A basic understanding of how stress is computed in FEA is important when reporting stress results
- Postprocessors allow users to flip between nodal and elemental stress results seamlessly
- Understanding how the solver reports stress output is critical to knowing how to interpret stress results in the postprocessor

How is Stress Computed for an Element or Node?

Gauss/Integration Points

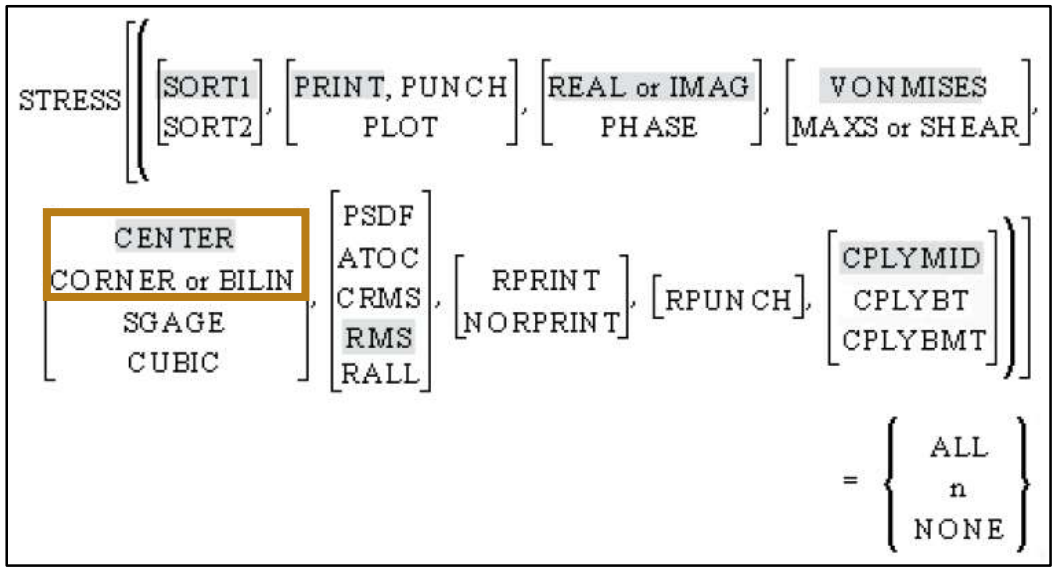
- In Finite Element Theory, stress results are computed at one or more points within the element (called Gauss or integration points).
- The number of Gauss points is determined by the type, shape, and order (linear vs. parabolic) of the element.
- The stresses obtained at these Gauss points inside the element are then extrapolated to the nodes when reporting nodal stress results.



Note: An 8-node brick element has 8 Gauss points, while a 20 node brick has 27 Gauss points.

Simcenter Nastran Stress Output Request

CENTER vs. CORNER



*From Nastran Quick Reference Guide (QRG); "greyed out" entries are the Nastran defaults

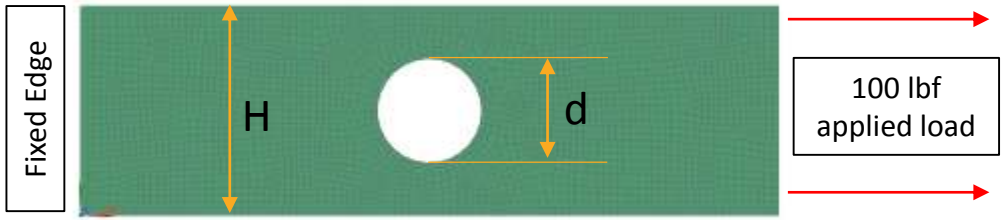
- **CENTER** = stress output at the center of the element only *(Simcenter Nastran default)*
- **CORNER** = stress output at the center of the element and at the nodes
- For parabolic elements, both center and corner stress is output by default

Users are encouraged to request CORNER as the stress output location to allow for proper viewing of both nodal and elemental stress results.

Viewing nodal stress results using the CENTER stress output location request can lead to inaccurate stress predictions, as is demonstrated in the following example.

Stress Output Location Example

Viewing Nodal Stresses With CENTER vs. CORNER Results



(Mesh Size = 0.2 in)

<u>Geometry of Test Sample</u>

d= 3”

H = 6”

d/H = 0.5

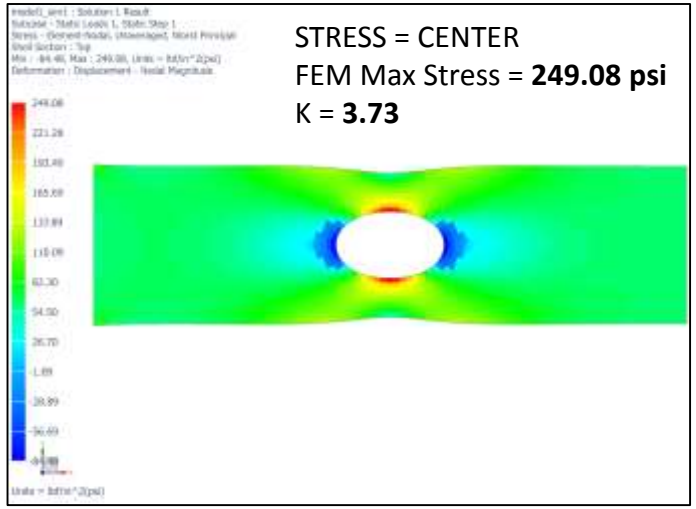
h = 0.25”

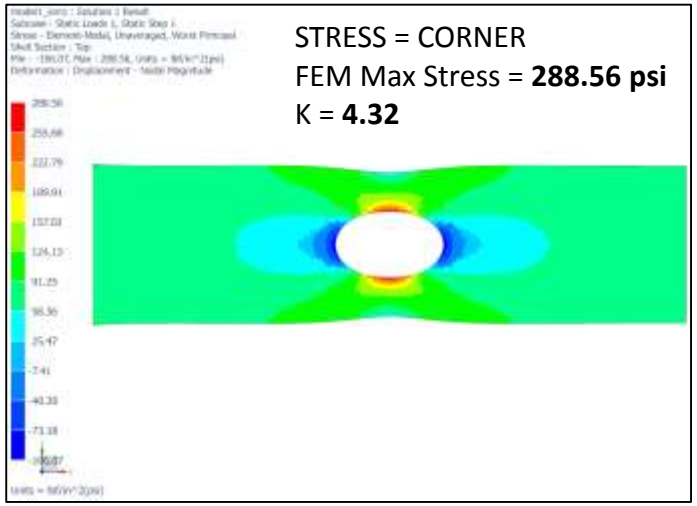
Nominal Stress = 100 lbf / (0.25 in*6 in) = 66.67 psi

Theoretical K = 4.32 (*from Peterson 3rd ed., Chart 4.1*)

Theoretical Max Stress = 66.667 psi * 4.32 = **288.8 psi**

The STRESS = CENTER output location request underpredicts the peak stress because the stress at the center of the elements is less than the extrapolated stresses at the edge of the hole.



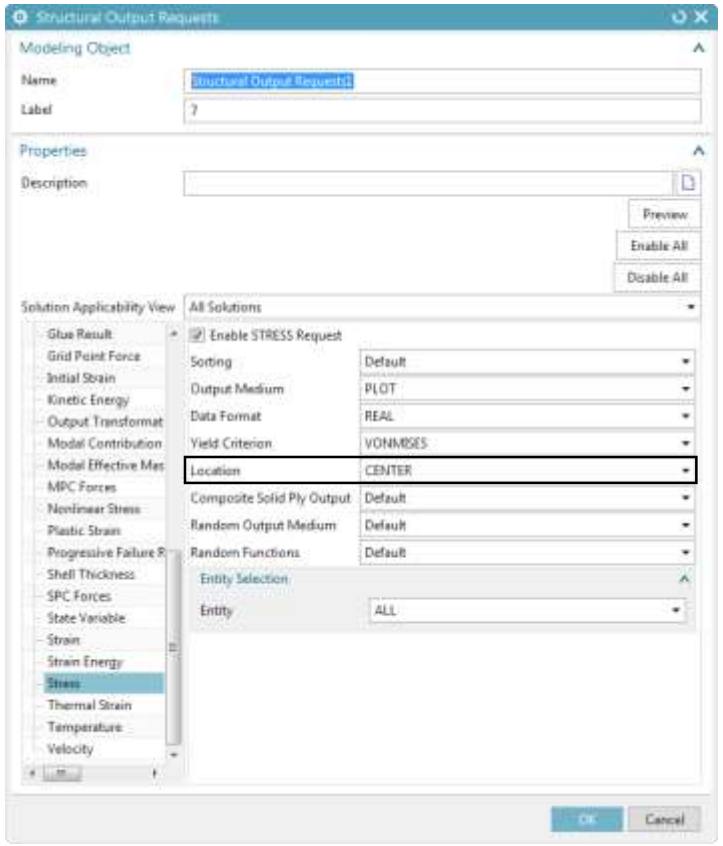


<u>Note</u>: Both plots use nodal stresses.

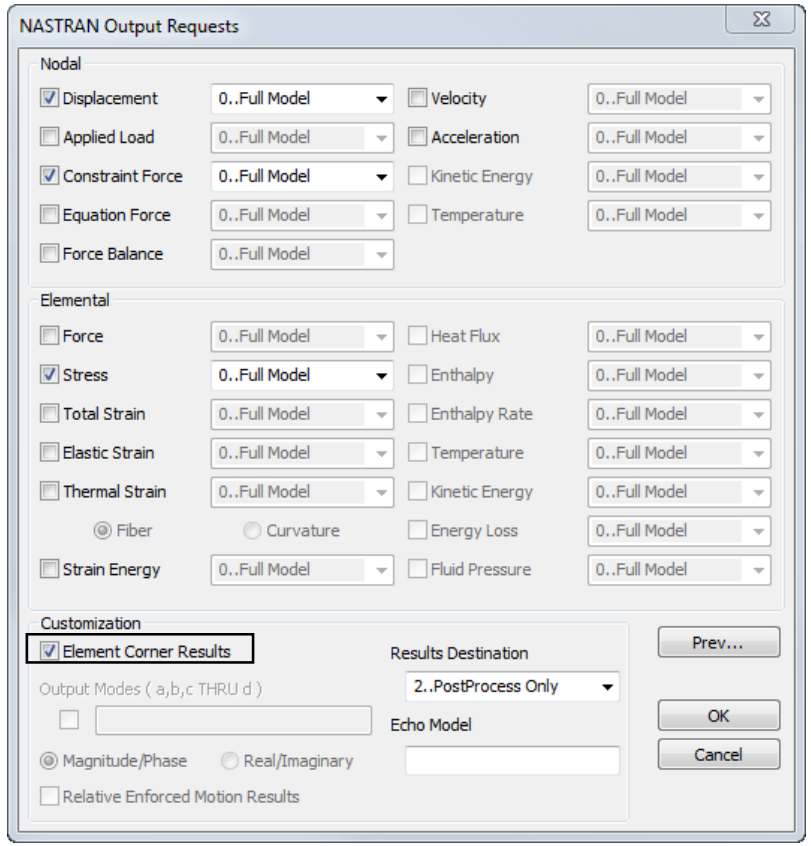
Stress Output Request

Simcenter 3D vs. FEMAP Defaults

- The default for **Simcenter 3D** is **CENTER**



- The default for **FEMAP** is **CORNER**



Changing the Default Stress Data Recovery Location in Simcenter 3D

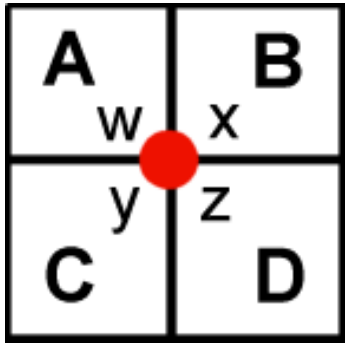
- The following steps will change the default setting for the STRESS data recovery location from CENTER to CORNER:
  1) Go to this folder, or the equivalent path on your installation and NX version: C:\Siemens\NX1851\SIMULATION\templates
  2) Open the file – "SimNxNastranEnglish.sim" or "SimNxNastranMetric.sim" – depending on the unit system of interest (you may not have write-access to either of these files in your installation directory so you might have to save a copy of the file to a temporary location such as your desktop)
  3) With the template file you will want to:
     a) Create a new solution
     b) Edit the Case Control Output Requests to your preferences
     c) Delete the solution (your structural output requests will be saved as a modeling object, and is not deleted)
  4) Save the templates file and, if necessary, move it back to the templates folder in your NX installation directory
- These presets will take effect for new .sim files you create, but not existing ones.

Different Ways to Report Stress

- **Nodal Peak**: reports the peak nodal stress from each contributing element
- **Nodal Average**: reports the average of all elemental input for each node
- **Element Centroidal**: reports the average of the Gauss point stresses

Stress Result Type	Calculation
Nodal Peak	max(w, x, y, z)
Nodal Average	(w + x + y + z)/4
Element Centroidal*	-

*No averaging between elements occurs with element centroidal output results; the average of each element's gauss point stresses is reported.

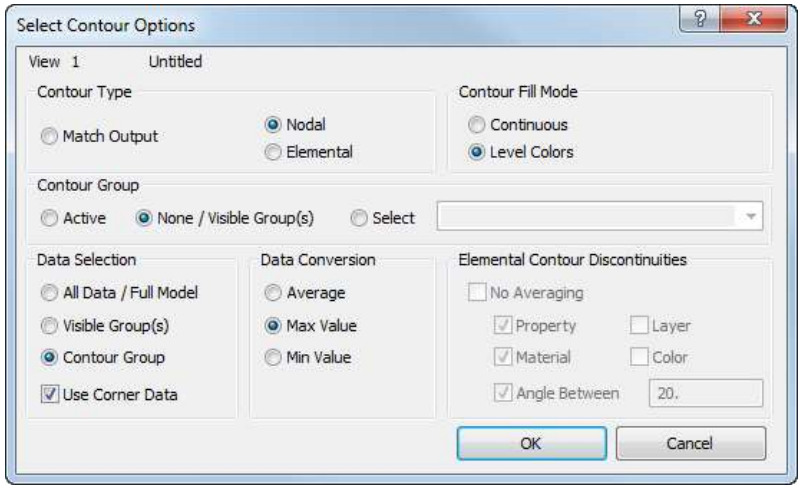


- **A, B, C, D** = average of each element's gauss point stresses (i.e., the element centroidal stress for each element)
- **w, x, y, z** = each element's extrapolated stress at the shared center node

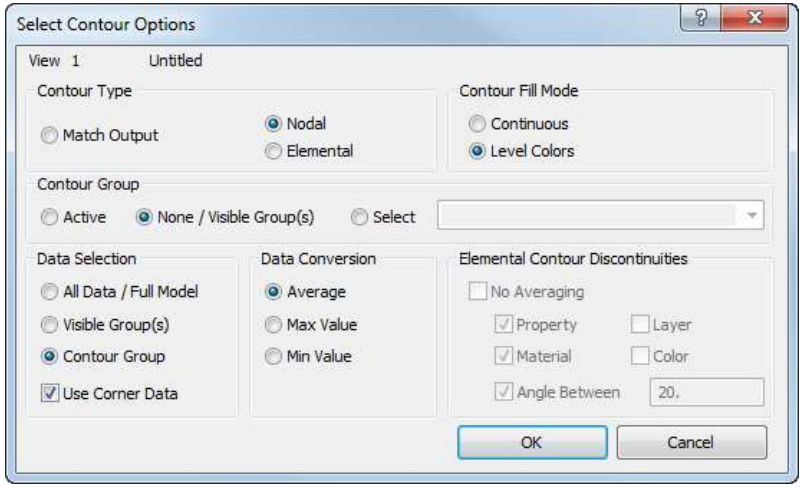
The stress at this interior node has four values associated with it for a given stress tensor.

Contour Settings to Use in FEMAP

Nodal Peak

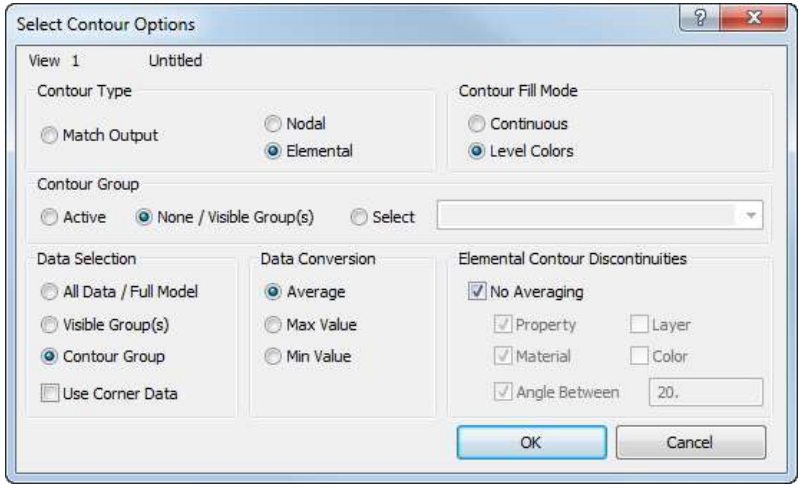


Nodal Average



Note: Nodal contouring simply averages all values at the nodes and cannot account for any discontinuities in material or geometry (hence the Element Contour Discontinuities options are not available).

Element Centroidal



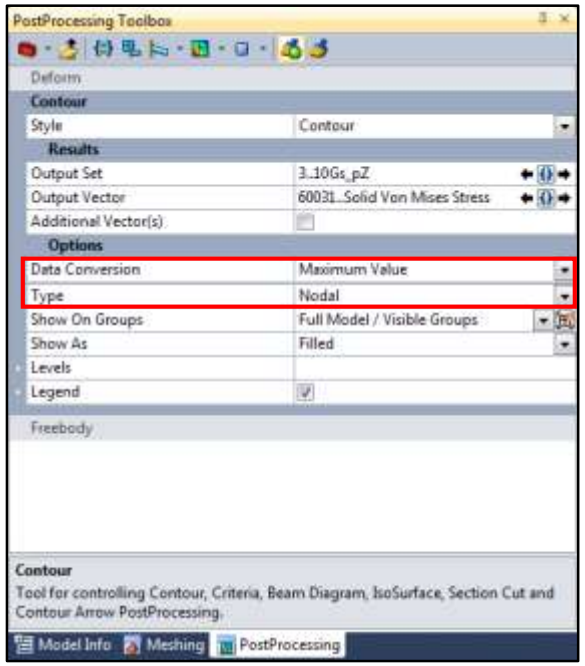
ATA ENGINEERING, INC.

Contour Settings to Use in FEMAP

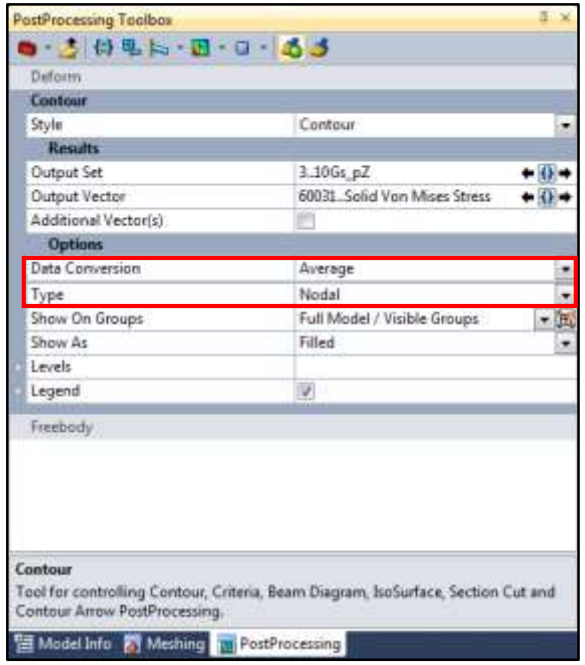
Post-Processing Toolbox

The same contour settings applied using the dialog boxes on the previous slide can also be specified in the Contour "tool" in FEMAP's PostProcessing Toolbox.

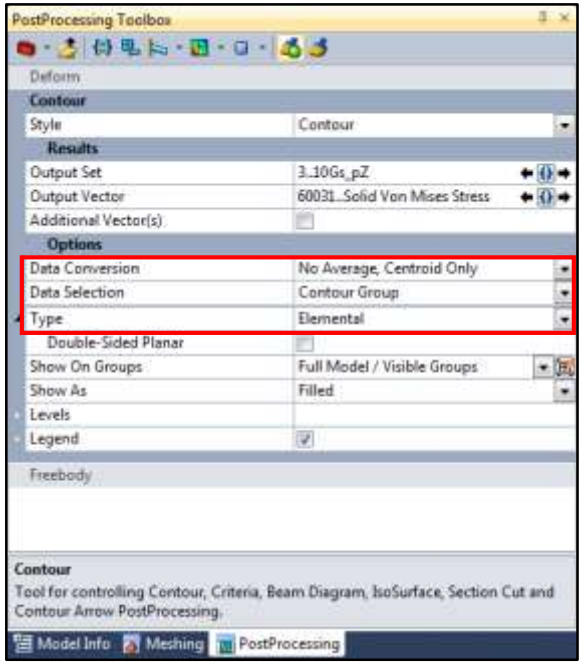
Nodal Peak



Nodal Average



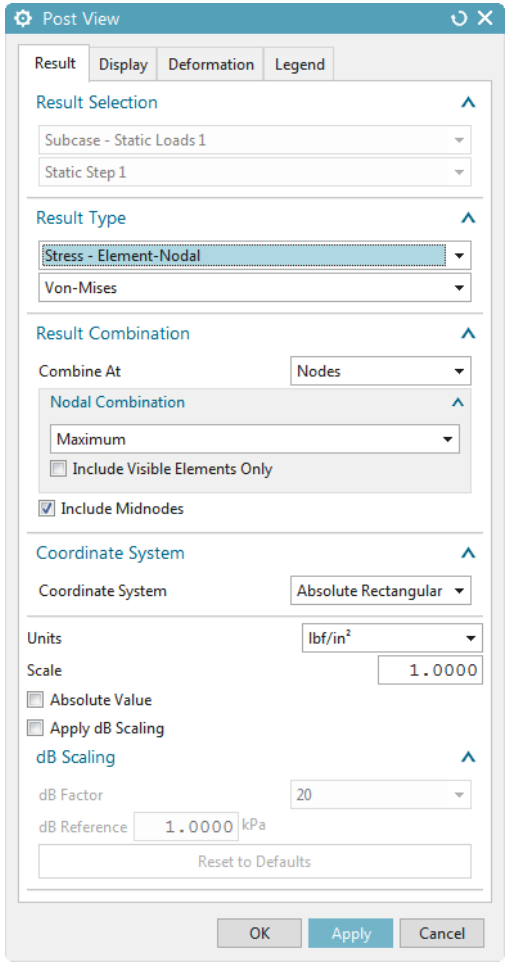
Element Centroidal



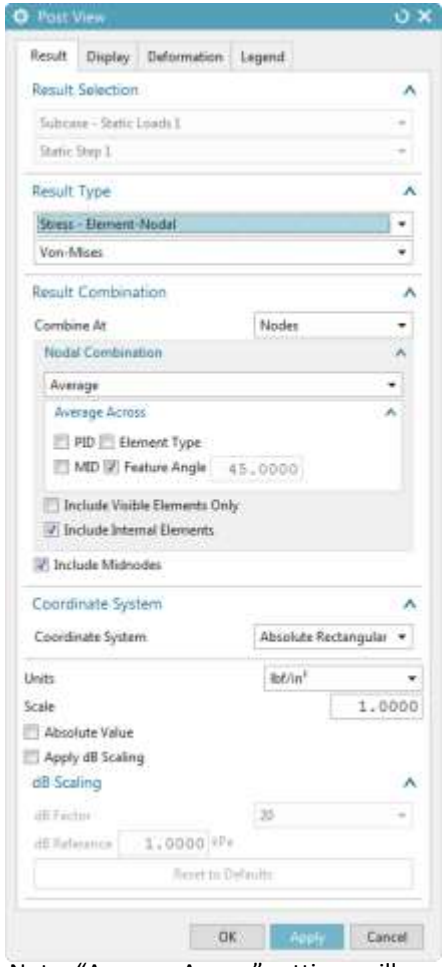
ATA ENGINEERING, INC.

Contour Settings to Use in Simcenter 3D

Nodal Peak

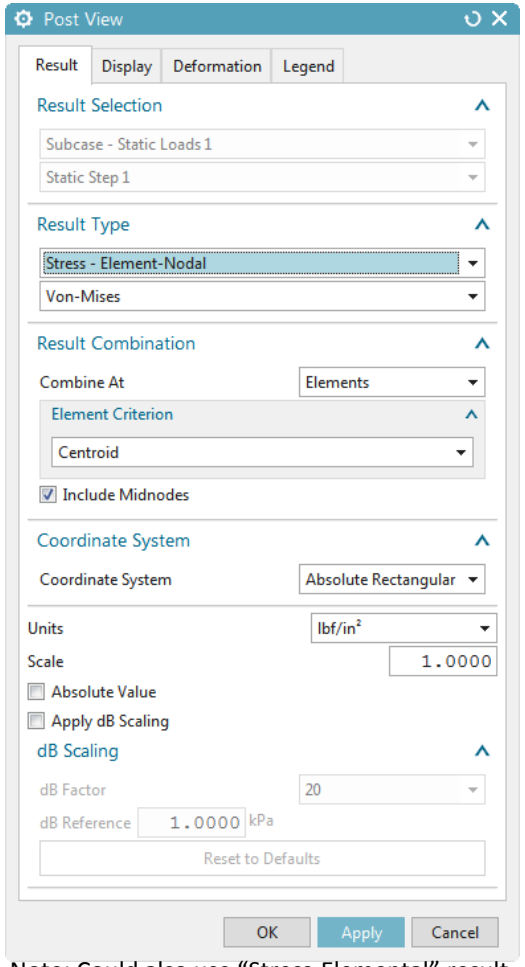


Nodal Average



Note: "Average Across" settings will be mesh dependent; uncheck the box to turn off averaging across feature.

Element Centroidal

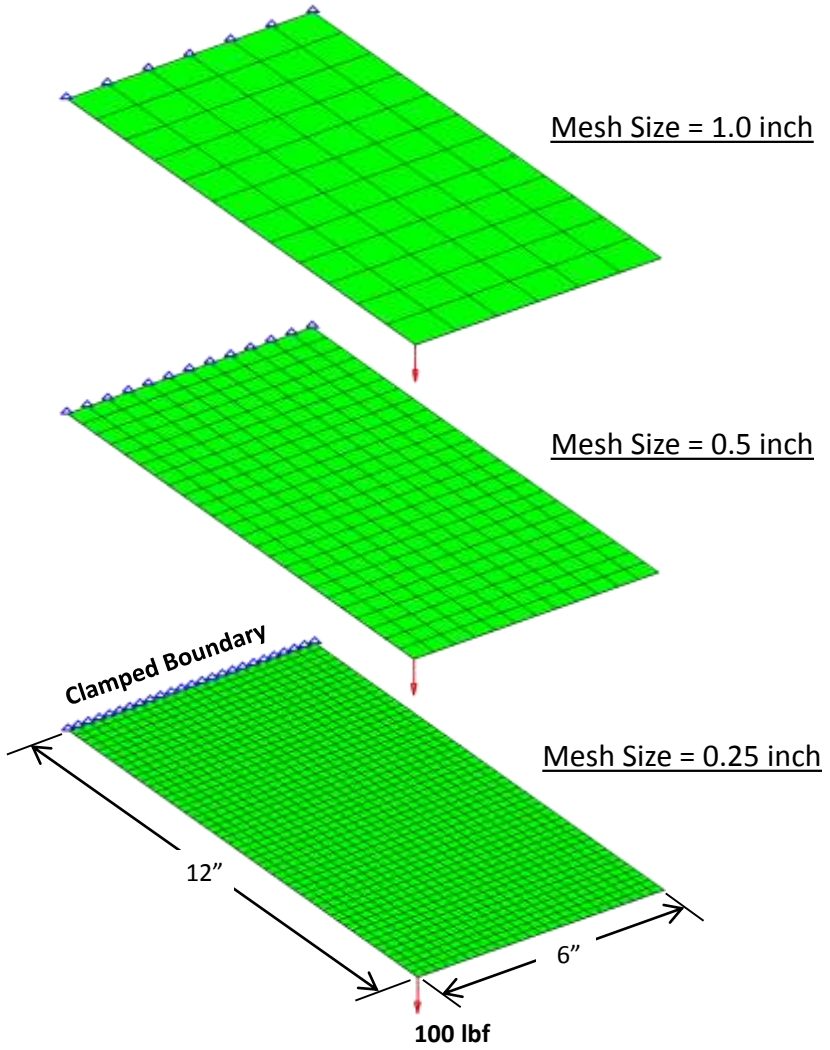


Note: Could also use "Stress-Elemental" result type with Result Combination set to "None" for element centroidal results.

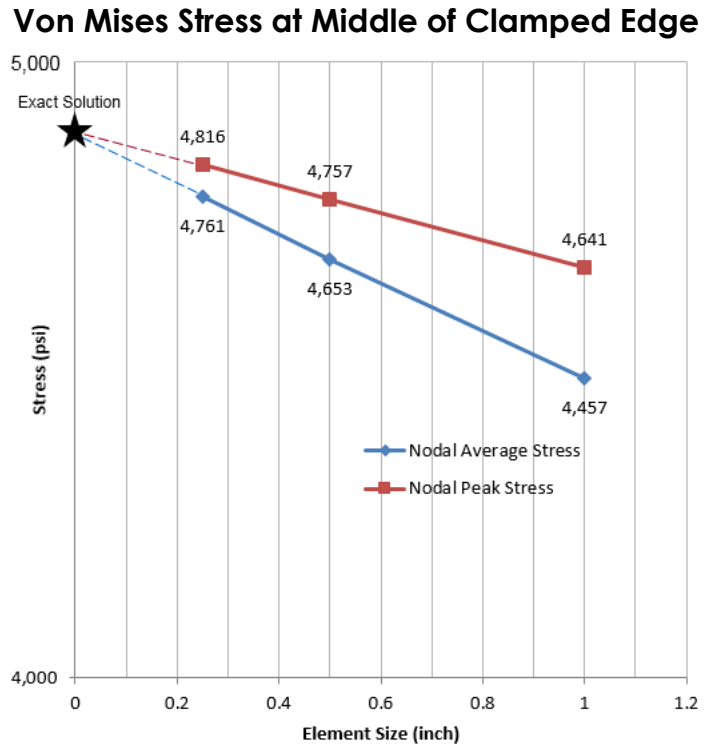
When Would I Use Nodal vs. Elemental Stress Results?

- General rules:
  - 1) Nodal peak is always more conservative than nodal average.
  - 2) Element centroidal is less conservative but may be required if you have a specific allowable correlated to this stress type (e.g., element-size correlated allowables).
- When assessing fatigue, nodal results are often used. For strength assessment, nodal or elemental results could be used depending on the required fidelity of the analysis.

Side Note 1: Comparing Different Stress Result Types is One Means of Evaluating Convergence



- The difference between nodal peak and nodal average stress decreases as the mesh size is reduced, the result of the elements' Gauss points moving closer to the nodes.
- As a result, comparing nodal peak with nodal averaged results can be used as a convergence criterion.



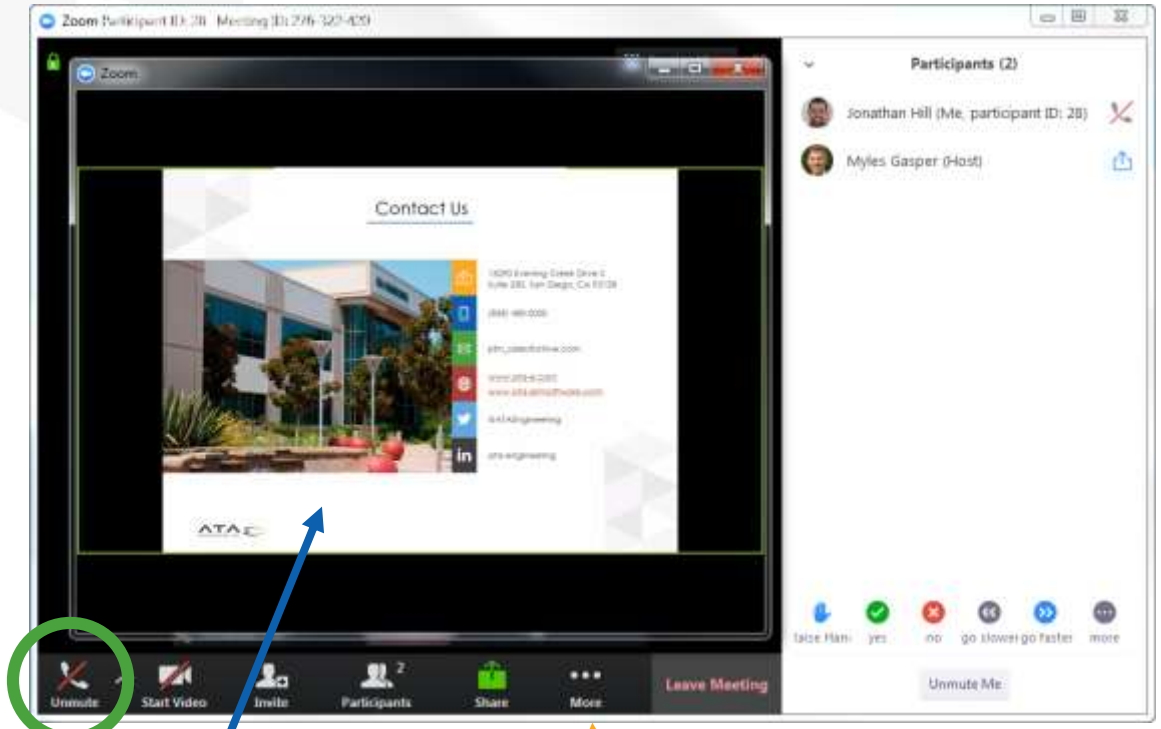
Side Note 2: Nodal Average Stress Calculation in Simcenter 3D vs. FEMAP

- The nodal average stress calculation is handled differently in Simcenter 3D and FEMAP:
  - **Simcenter 3D**: averages the component stresses at each node before computing the von Mises stress at the node
  - **FEMAP**: first computes von Mises stress from the component stresses then averages those von Mises stress values at each node
- The order of operations differs which is important to note if trying to compare Simcenter 3D nodal average stress results with FEMAP nodal average stress results

Questions?

Submit questions in the chat or unmute yourself now

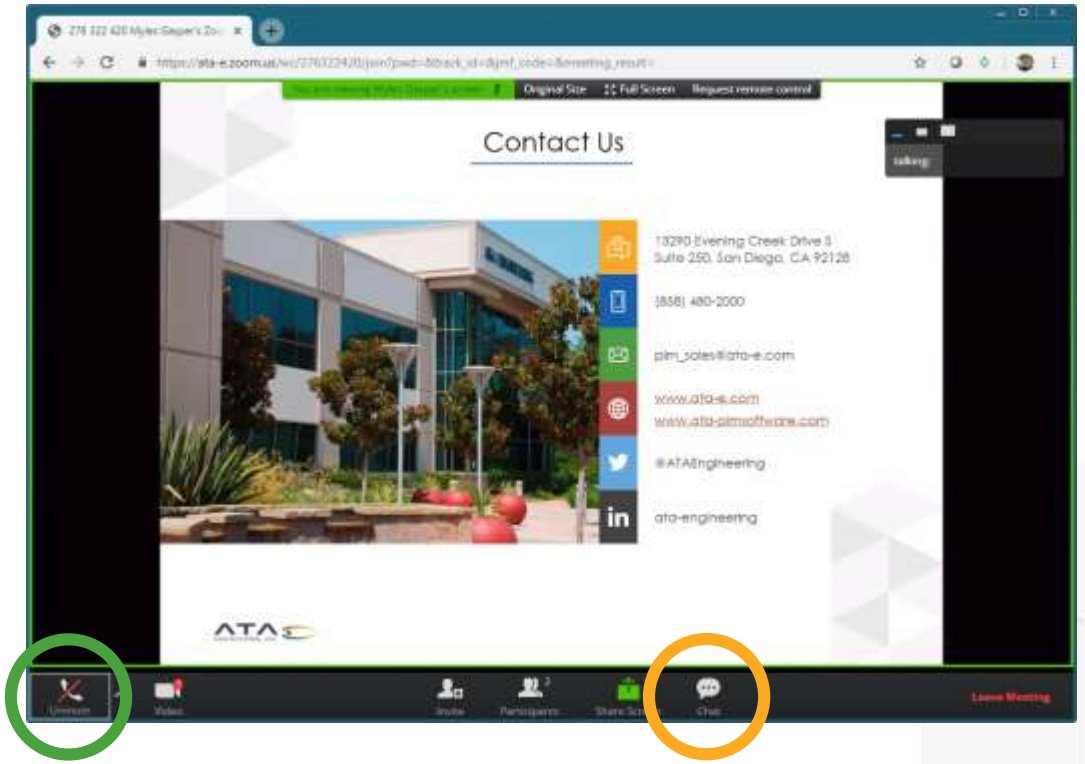
Zoom Application



Screenshare in separate window

Chat is available under More

Web Interface



Contact Us

13290 Evening Creek Drive S
Suite 250, San Diego, CA 92128

(858) 480-2000

plm_sales@ata-e.com

www.ata-e.com
www.ata-plmsoftware.com

@ATAEngineering

ata-engineering