



Webinar:
Checking Element Quality in Femap

Tommy Board, ATA Engineering
December 11th, 2019

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

 (858) 480-2000

 www.ata-e.com

 [ata-engineering](https://www.linkedin.com/company/ata-engineering)

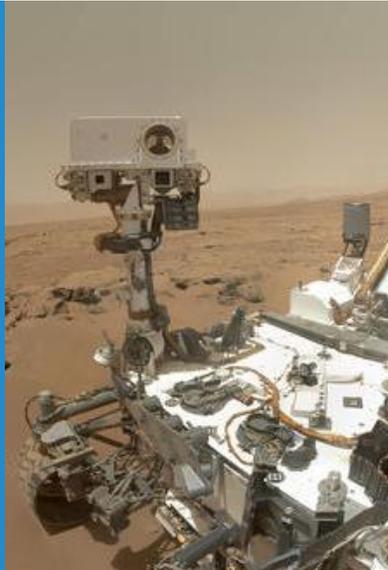
 [@ATAEngineering](https://twitter.com/ATAEngineering)

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

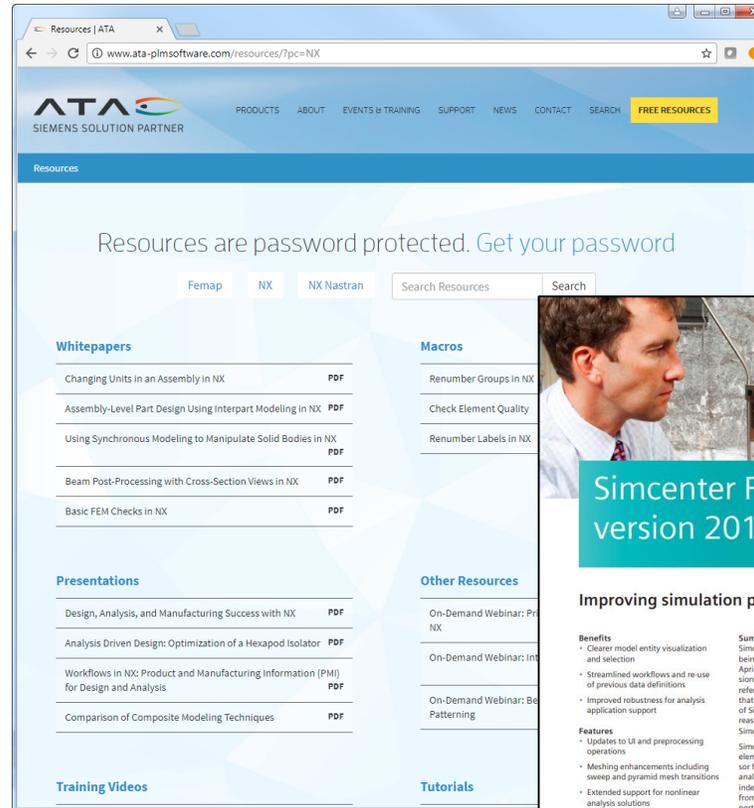
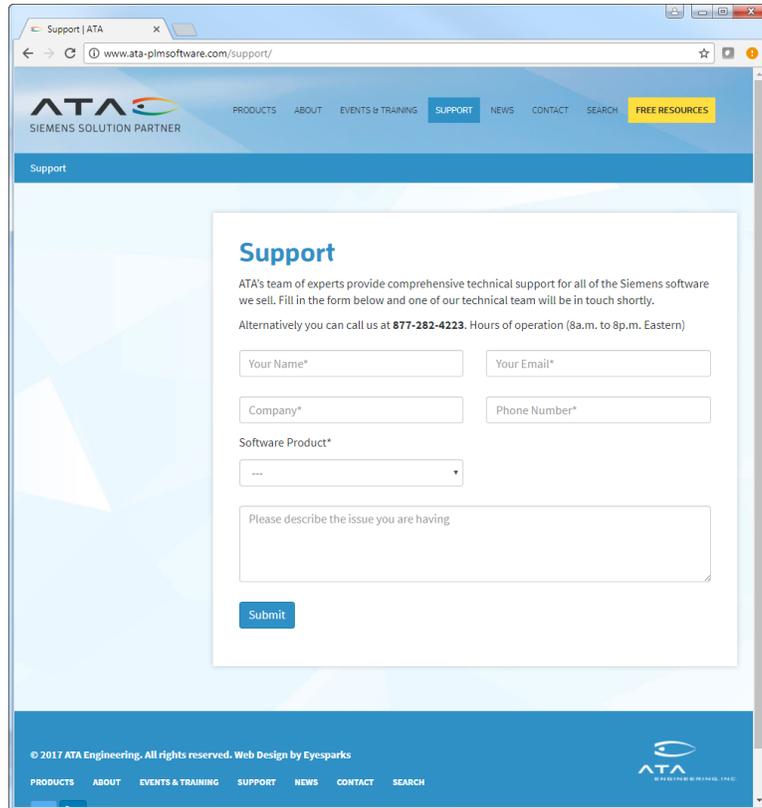
ATA offers training, free resources, and hotline support for a variety of Siemens products.



- 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+

Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com



Simcenter Femap version 2019.1

Improving simulation productivity

Benefits

- Clearer model entity visualization and selection
- Streamlined workflows and re-use of previous data definitions
- Improved robustness for analysis application support

Features

- Updates to IJL and preprocessing operations
- Meshing enhancements including sweeps and pyramidal mesh transitions
- Extended support for nonlinear analysis solutions
- Enhanced results review options for partial results vectors

Summary

Simcenter™ Femap™ software is now being released on a biannual schedule in April and October, beginning with version 2019.1. The software is now referred to as Simcenter Femap to reflect that it is a part of the Simcenter portfolio of Siemens CAE products. For the same reason, NX™ Nastran® software is now Simcenter™ Nastran™.

Simcenter Femap is a standalone finite element modeling pre- and postprocessor for engineering simulation and analysis. The software is CAD-independent and can import geometry from all major CAD platforms and supports most CAD data formats. Simcenter Femap also works in combination with a wide variety of finite element analysis solvers, including the industry-leading Simcenter Nastran software.

The latest release provides a variety of improvements that will improve your productivity across the simulation workflow. Model creation enhancements range from entity selection and geometry splitting operators to meshing, including new sweep and mesh transition options. There are also several updates to solver integration for Nastran®, Ansys® and Abaqus®, and new results viewing selections in postprocessing.

Visualization and user interface

Entity selection

Entity selection methods have been enhanced to allow you to select only from those entities that are visible on the screen at the time of selection, using the new "select visible" icon button on the entity selection dialog.

Also, elements can be selected based on a reference to an orientation mode. These additions enhance the current array of available tools, making entity selection more versatile and flexible. You'll also notice that the ID fields in the entity selection and other dialog boxes throughout Simcenter Femap have been widened.

Previous vector

Simcenter Femap 2019.1 includes the ability to recall a previously defined vector in any operation that requires a vector direction definition to be created. Accessing previously defined data accelerates the model creation process, enhancing productivity.

Locate in model info

A new "locate in model info" toggle is added as a new selector mode to the selector toolbar that highlights the selected entity (solid, CYS, property, material or layout) in the model info tree. This makes it easier to locate the selected item in the model info tree when there are many items of that type in the model.

siemens.com/femap



Webinar:
**Checking Element Quality in
Femap**

Tommy Board, ATA Engineering
December 11th, 2019

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

 (858) 480-2000

 www.ata-e.com

 [ata-engineering](https://www.linkedin.com/company/ata-engineering)

 [@ATAEngineering](https://twitter.com/ATAEngineering)

What's So Bad About "Bad Elements" Anyway?

- Element quality is key for producing high-quality, accurate results and deliverables
- Finite element models inherently have error because they use discrete elements to represent continuum bodies
- Bad elements deviate considerably from equilateral shapes and further reduce model accuracy
- This webinar will include:
 - What do bad elements look like?
 - How to identify bad elements in Femap
 - Methods to improve those bad elements
 - Live demo in Femap v2019.1, but commands are similar across different versions

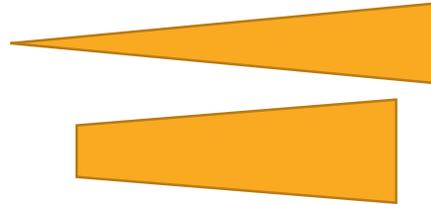
Poor Element Quality Comes in Different Forms

- Measures for quantifying element quality include:

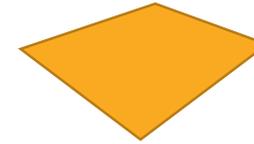
Aspect Ratio



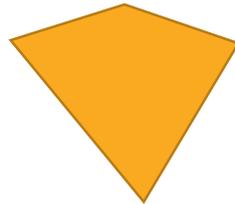
Taper



Element Warping
(out-of-plane deviation)



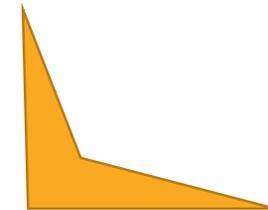
Interior Angles



Skew



Jacobian

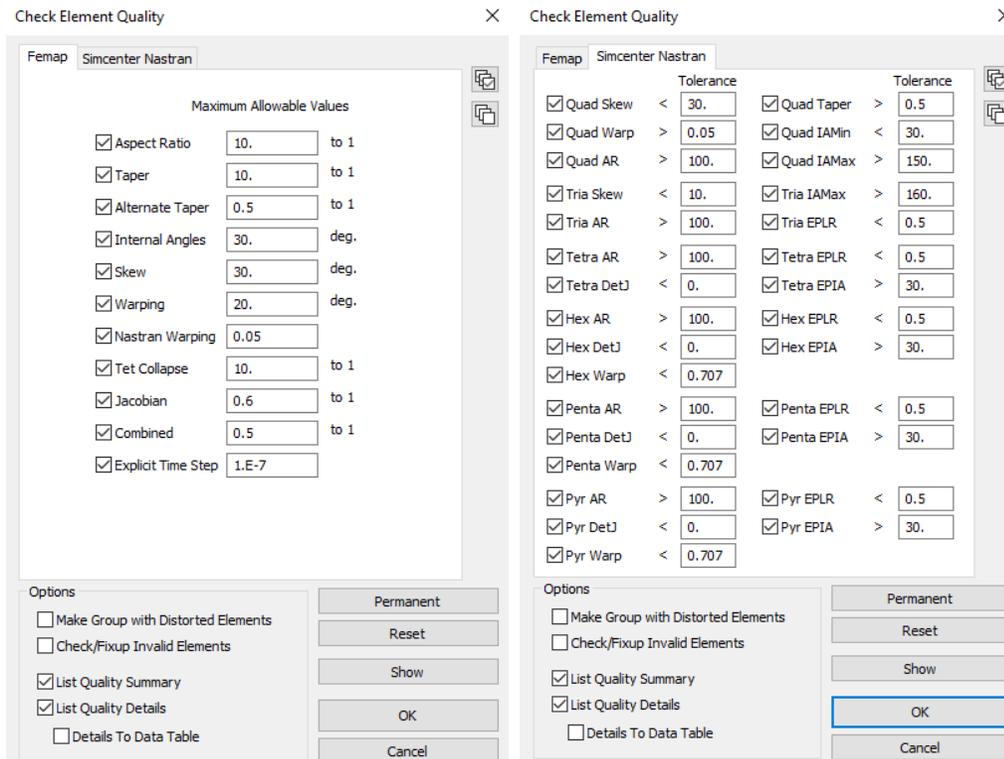


- Jacobian determinant:

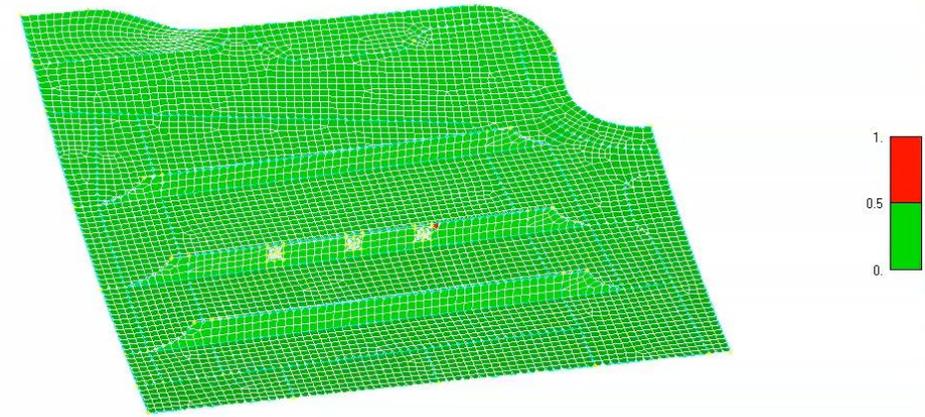
- Negative values occur at node if corresponding interior angle is greater than 180°
- Solution will fail with negative determinant
- Examples:
 - When a triangle starts to flatten into a straight line
 - When a quad starts to look like a triangle

How to Check Element Quality in Femap

- In the Menu bar, Tools -> Check -> Element Quality
- Select the elements of interest
- The next dialog box has two tabs, Femap and NX Nastran
 - If NX Nastran is your solver, use this tab with the default checks
 - Femap checks may be helpful if you are using a different solver. They have different default values and some elements may fail under those that would not fail with Nastran checks.



You can also create a contour of good/bad elements in Meshing Toolbox under Surface Mesh Quality:



Output Set Surface Mesh Quality
Elemental Contour: Quad Taper

How to Check Element Quality in Nastran

You should do this **in addition** to checking quality in Femap

➤ F06 file will list out poor quality elements

TOLERANCE LIMITS ARE: SA = 30.00, IA(MIN) = 30.00, IA(MAX) = 150.00, WF = 0.05, TR = 0.50, AR = 100.00
(xxxx = LIMIT VIOLATED)

ELEMENT TYPE	ID	SKEW ANGLE	MIN INT. ANGLE	MAX INT. ANGLE	WARPING FACTOR	TAPER RATIO	ASPECT RATIO
QUAD4	302104	76.84	54.56	118.99	0.00	0.53 xxxx	1.98
QUAD4	302252	69.59	47.02	115.35	0.00	0.55 xxxx	2.29
QUAD4	302253	78.66	55.82	122.06	0.00	0.55 xxxx	1.92
QUAD4	302255	83.86	60.56	124.48	0.00	0.53 xxxx	1.92
QUAD4	302256	71.23	47.73	120.93	0.00	0.59 xxxx	2.30
QUAD4	302258	83.25	60.08	124.90	0.00	0.54 xxxx	1.81
QUAD4	302260	70.83	47.88	120.48	0.00	0.59 xxxx	2.23
QUAD4	302277	75.82	51.29	124.96	0.00	0.59 xxxx	2.20
QUAD4	302279	84.12	60.21	127.33	0.00	0.55 xxxx	1.86

'xxxx' indicates an element warning; 'FAIL' is printed for elements that fail the NASTRAN element quality check

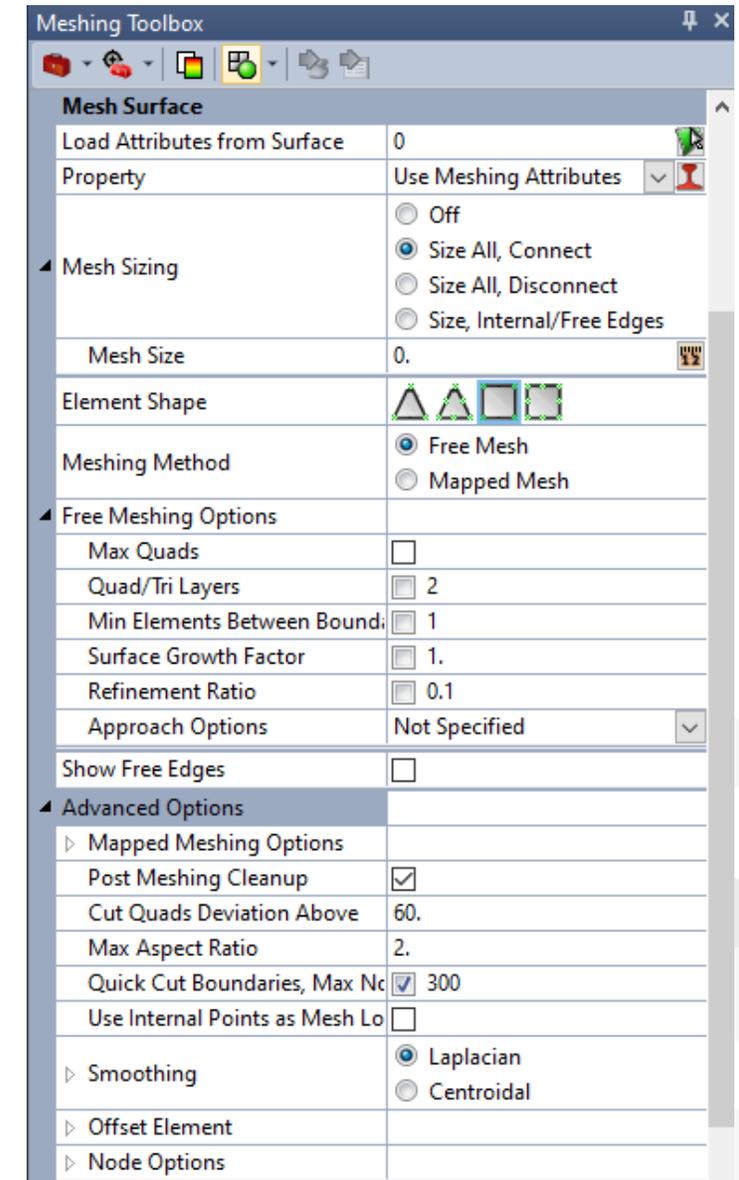
- Nastran will sometimes flag elements that didn't get flagged by pre-processor element quality checks, and vice versa
- What if the .f06 file didn't print out all the bad elements? In case control, add this line:
 - GEOMCHECK MSGLIMIT = 5000

CheckElems API Tool

- Available as a Femap API program
- Reads an .f06 file and adds any elements that exceeded the solver's tolerances to a new group in Femap
- Located on our Siemens PLM Website:
 - <https://www.ata-plmsoftware.com/resources/check-element-quality/>
 - Password: plmsoftware
 - Click "Download the Femap API"
 - Save the file "CheckElems.exe" into your local Femap API directory

Fixing Bad Elements in Femap

- Make a group of all the bad elements to visually understand how to improve them
- Bad geometry leads to bad elements
- Things to Try:
 - Change the element size
 - Free vs. mapped meshes
 - Add geometric partitions
 - Meshing Toolbox has a variety of options
 - Manual meshing (details on next slide)
 - For solid meshes, seed faces with 2D Plot-Only Planar elements
 - For a larger model, check quality of separate meshes before connecting them together



Femap Manual Meshing Commands

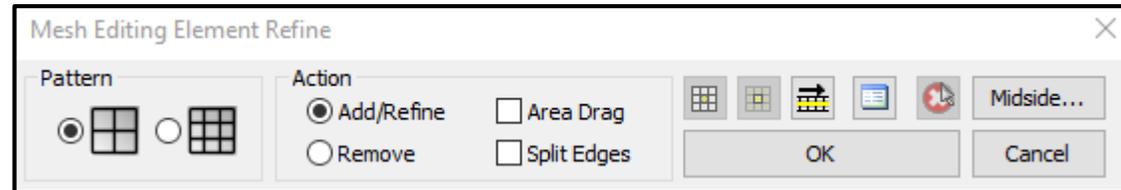
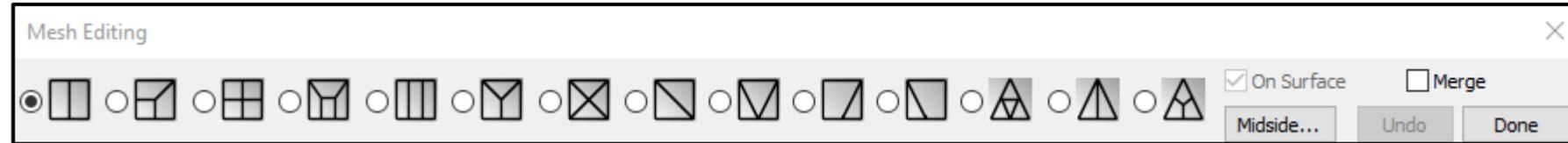
Useful for mesh improvements when geometry is not available

➤ Mesh->Editing

➤ Interactive

➤ Split

➤ Element Refine

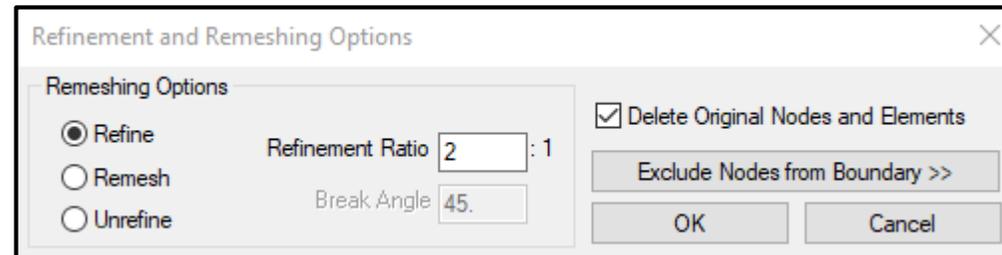


➤ Mesh->Remesh

➤ Refine

➤ Update

➤ Un-refine

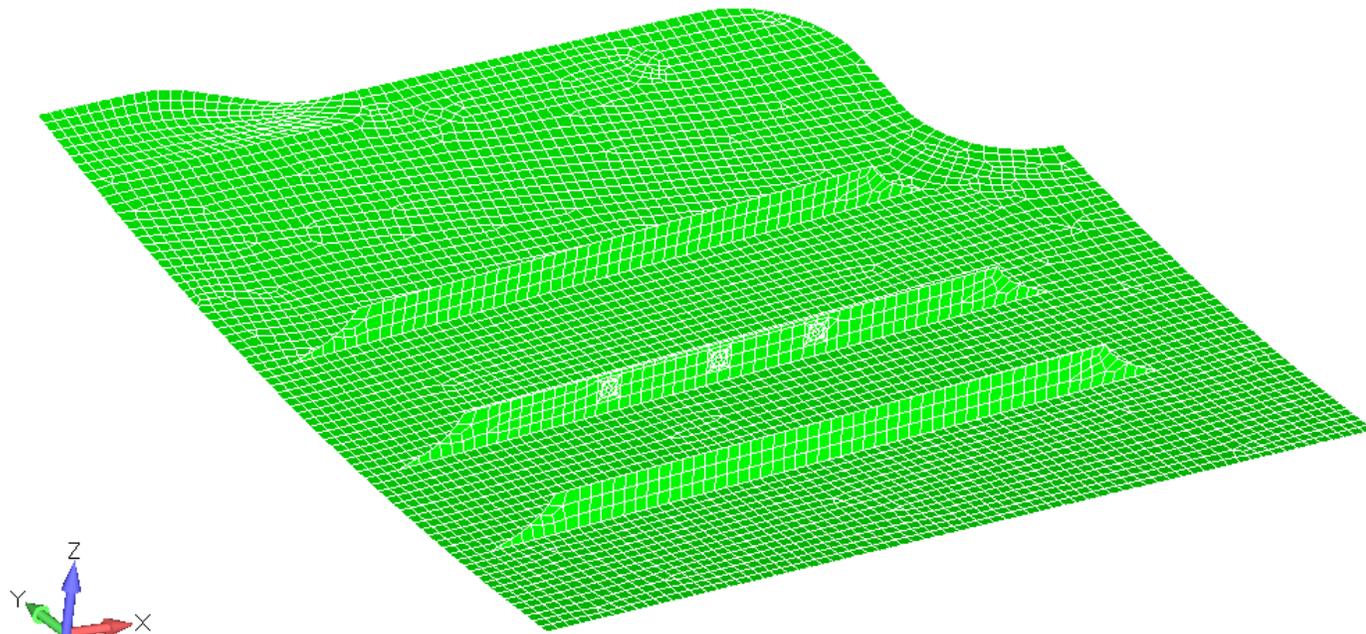


Be careful to ensure the mesh stays connected after manual meshing

Femap v2019.1 Demo

Untitled x

V: 1



Meshing Toolbox

Feature Suppression
Feature Removal
Feature Editing
Geometry Editing
Combined / Composite Curves
Combined / Boundary Surfaces
Mesh Sizing
Mesh Surface
Mesh Locate

Surface Mesh Quality

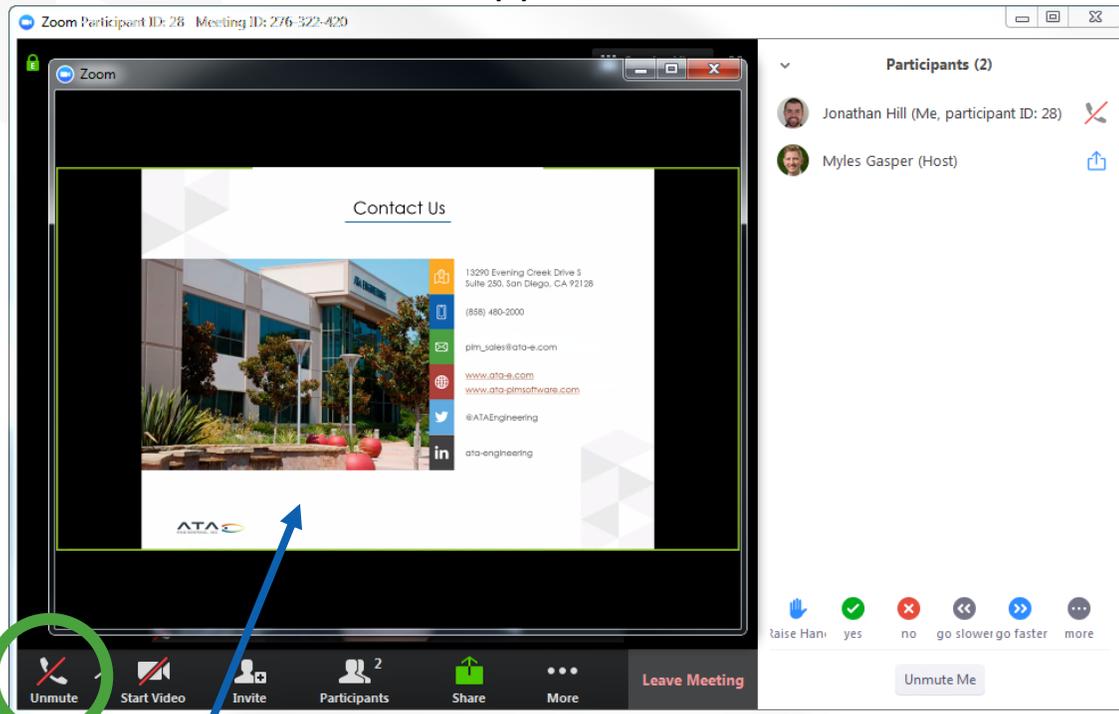
Quality Source	<input type="radio"/> Femap
	<input checked="" type="radio"/> Nastran
Quality Type	Quad Taper
Number of Quality Levels	2
Smooth Contours	<input type="checkbox"/>
Quad Taper	<input checked="" type="checkbox"/>
Max Allowable Value	0.5
Min Quality	0.
Max Quality	0.493364

Output Set: Surface Mesh Quality
Elemental Contour: Quad Taper

Questions?

Submit questions in the **chat** or **unmute yourself** now

Zoom Application

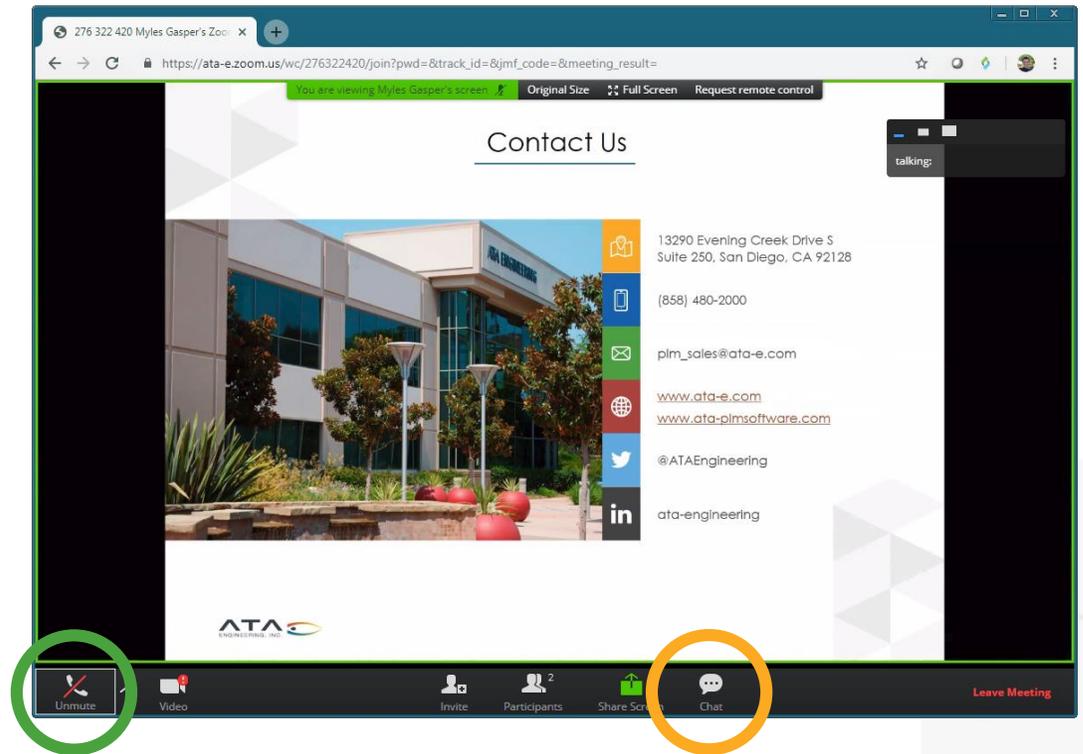


Screenshare in separate window

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