Webinar:
What's New in Femap 2020.1

Jonathan Hill, ATA Engineering

January 8th, 2020

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

📞 (858) 480-2000   🌐 www.ata-e.com

in 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 Digital Industries Software

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

➢ Siemens product lines we support include:
  - ➢ Simcenter STAR-CCM+
  - ➢ Simcenter 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
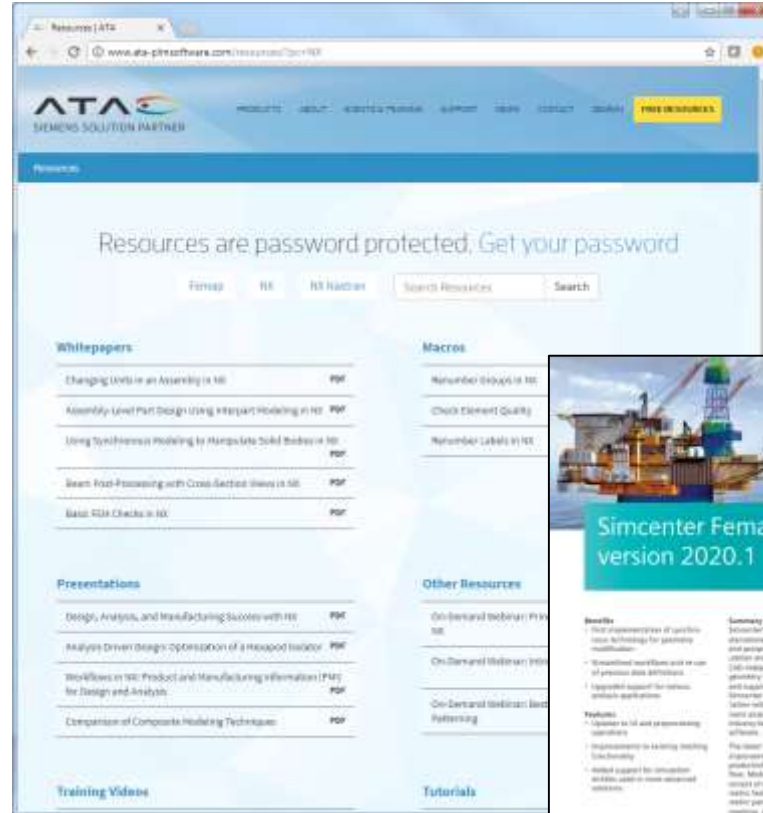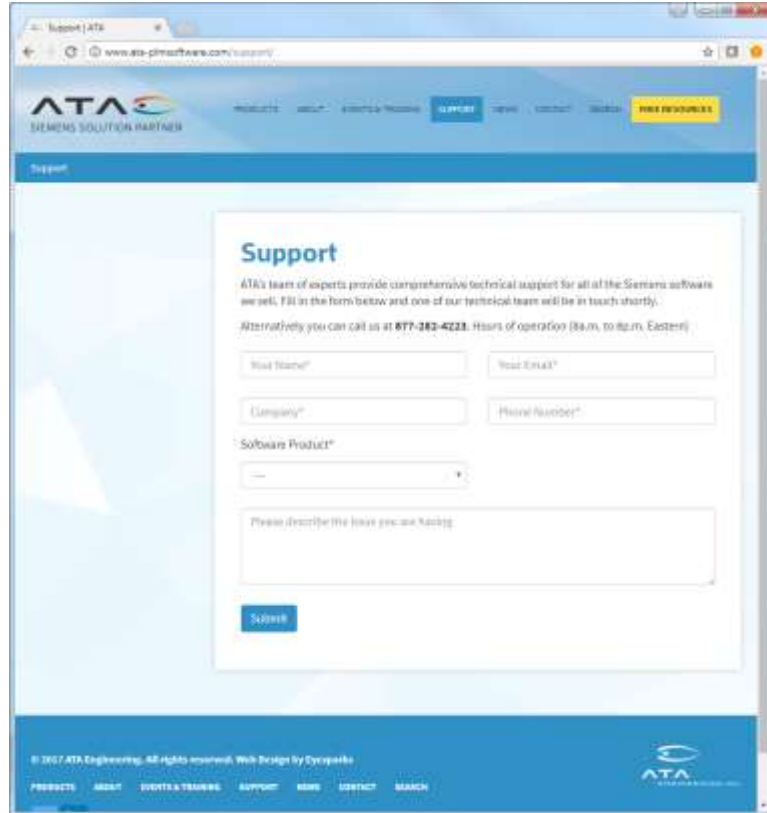
Webinar:
What's New in Femap 2020.1

Jonathan Hill, ATA Engineering

January 8th, 2020

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

📞 (858) 480-2000   🌐 www.ata-e.com

in ata-engineering   🐦 @ATAEngineering

Agenda

1. Introduction
2. Visualization and User Interface
3. Geometry Editing
4. Pre-Processing
5. Post-Processing
6. Solver Support
7. Q&A's

Femap 2020.1 is Available Now

➢ Femap 2020.1 MP1 has been released and is available for download through GTAC:
- ➢ MPs are available as full installations or as patches to existing 2020.1 installations
- ➢ Simcenter Nastran updates will be available as needed
- ➢ https://download.industrysoftware.automation.siemens.com/

➢ Don't forget to download a new license file, also available from GTAC:
- ➢ https://www2.industrysoftware.automation.siemens.com/LicenseManagement/Application

➢ This release enables users to maximize their efficiency of FEA tasks by building upon Femap's strong capabilities
- ➢ Geometry idealization and processing for FE models
- ➢ Powerful meshing, model creation and interactive editing
- ➢ In-depth support for industry standard solvers
- ➢ Flexible customization tools to streamline analysis processes

Femap 2020.1 Continues Biannual Release Cadence

➢ Femap targets biannual feature releases every April and October
  ➢ Add new features and expand functionality
  ➢ Accompanied by database changes and will require licensing updates
➢ Maintenance patches and Simcenter Nastran updates will be available as needed
  ➢ Critical updates only – database, UI, and licensing is unchanged
  ➢ MPs are available as full installations or as patches to existing 2020.1 installations
➢ Simcenter Femap 2020.1 (Nov. 2019) supersedes Femap 2019.1 (May 2019)
➢ Simcenter Femap 2020.1 (Nov. 2019) ships with Simcenter Nastran 2019.2 1884 (Sept. 2019)
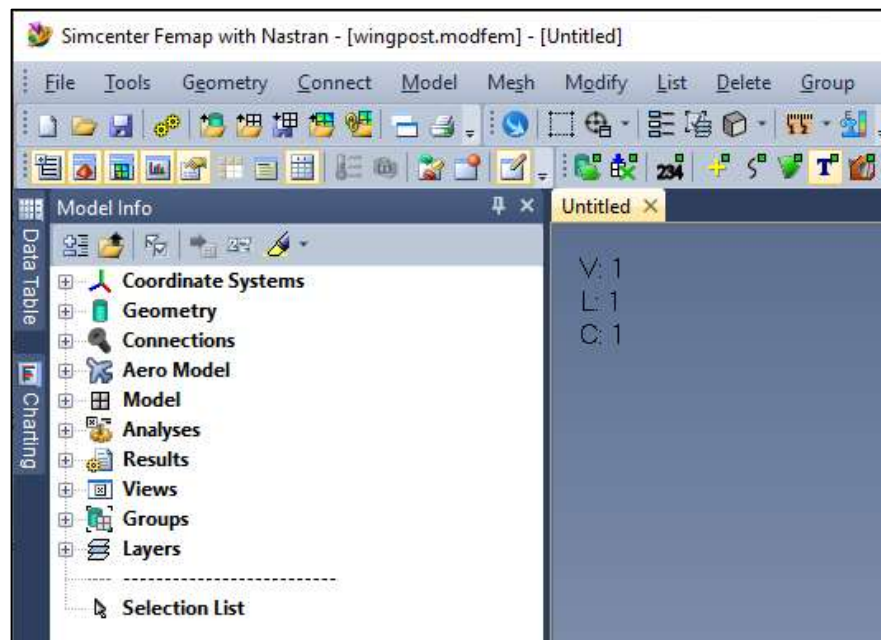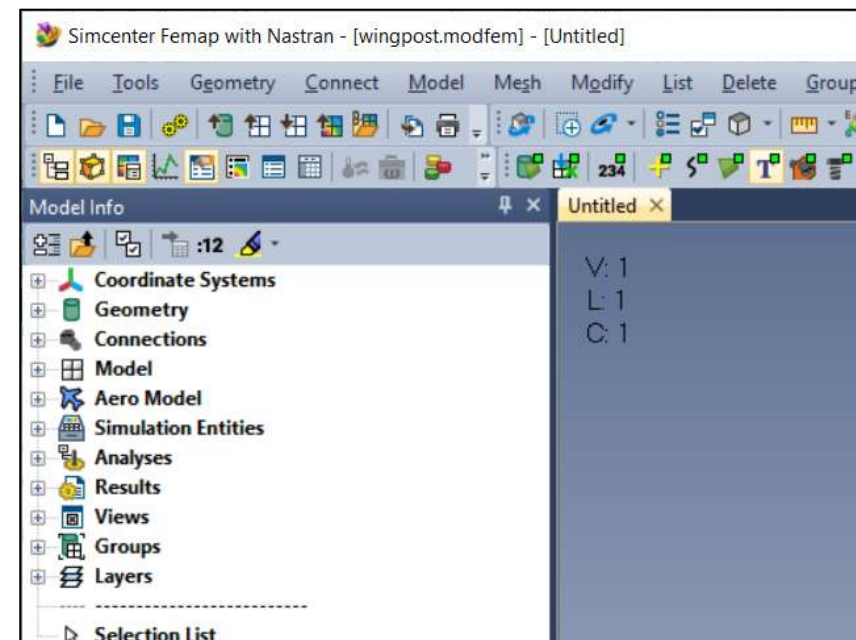
New Icons Are Introduced

Visualization and User Interface

➢ All 1600+ icons have been redrawn

  ➢ Designed to maximize familiarity while enhancing color contrast and providing a consistent theme

➢ Vector displays will allow for full support of high resolution displays in the future

2019.1   2020.1

Users Can View CBUSH CSYS and selected Color Palette

Visualization and User Interface

➢CBUSH element coordinate systems can be displayed
  - ➢ Available for coincident and non-coincident elements
  - ➢ Size is controlled with symbol size

➢*Color Palette* button is updated to show the current color and non-default pattern and line style

➢*Select from Standard Selection Dialog* icon allows for standard selection box functionality such as *Paste* or *^Pick* for check box entities
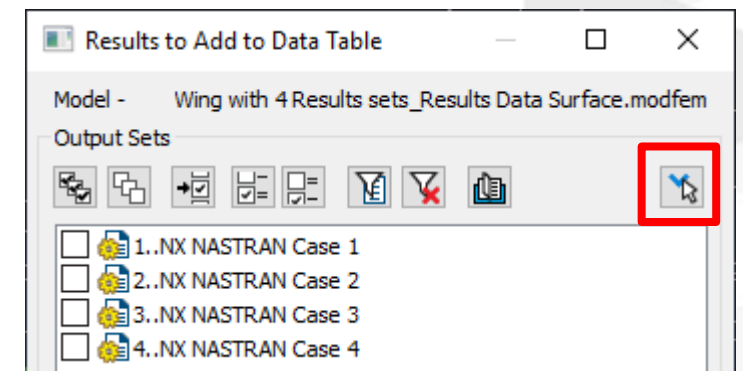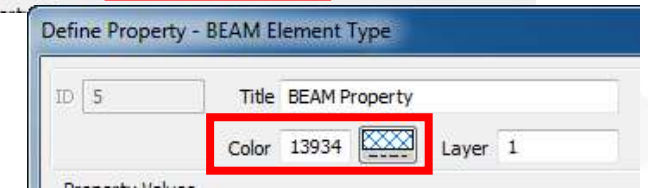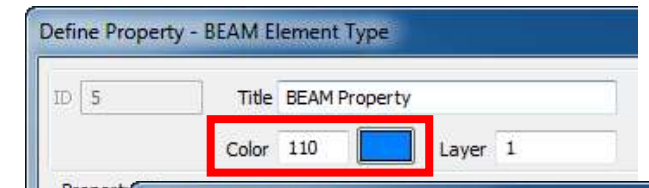
Visualization and User Interface

➤ Global options for digit control are set in File>Preferences, Graphics

    ➤ Locale: Specify global options for *Style, Decimal Zeros*, and *Sign*

    ➤ Options: Specify a *Method* (Exponent, Significant Figures, etc.) and Round to Zero value

➤ Control individual digits in *View Options* (F6)

    ➤ Available for *Load Vectors, Constraint, Deformed Model, Contour/Criteria Style, Contour/Criteria Legend, Contour Arrow Options*, and *Freebody Vectors*.
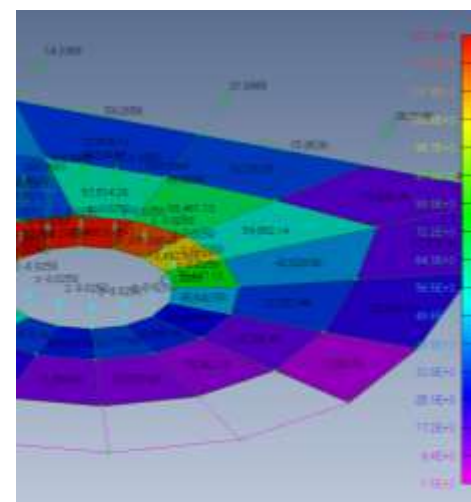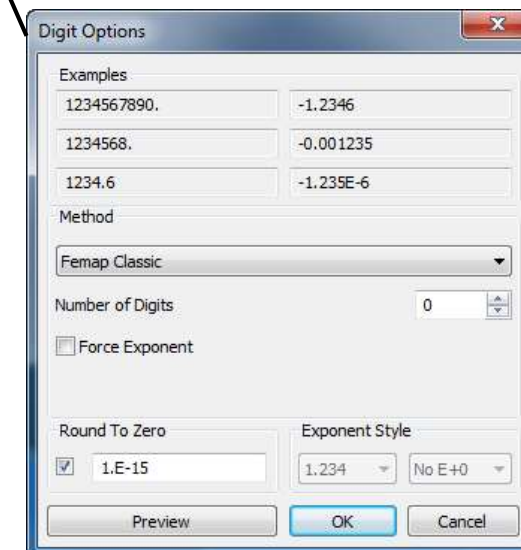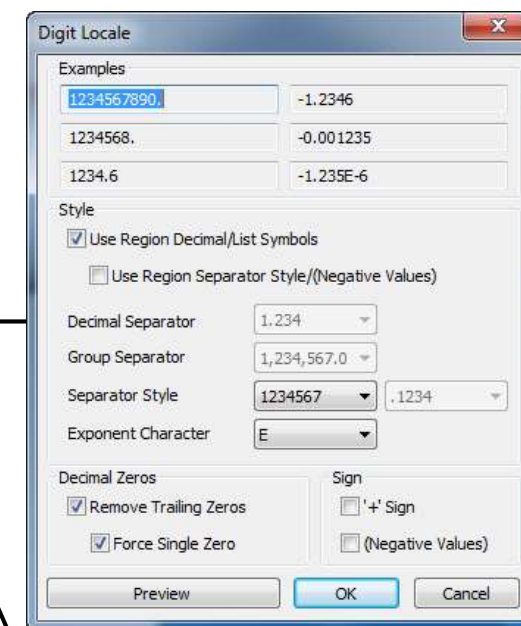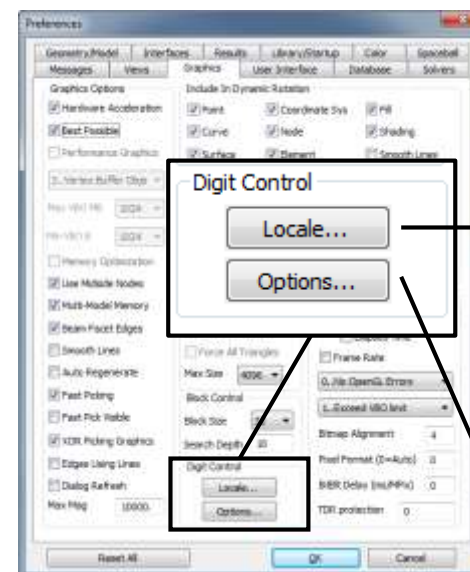
Smart Select comes to Feature Editing in the Meshing Toolbox

Geometry Editing

➢ Initial release of Smart Select for Feature Editing

➢ Leverages Siemens synchronous technology for translating and rotating surfaces

➢ Automatically selects additional surfaces that are offset, identical, concentric, colinear, and/or have tangent edges

➢ New Translation Options:
  ➢ Default (Workflow in 2019.1)
  ➢ Auto Normal (Enter Distance)
  ➢ Interactive (Prompts for Vector)

With Offset Detection
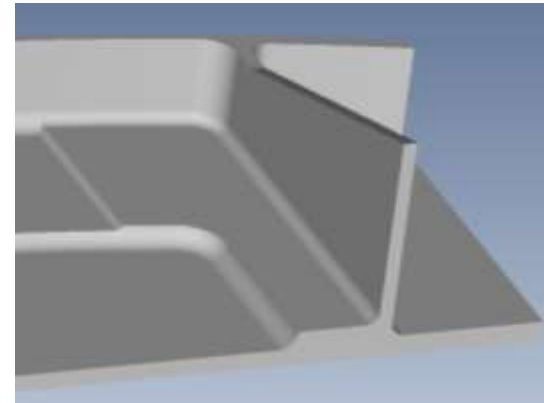
Without Offset Detection
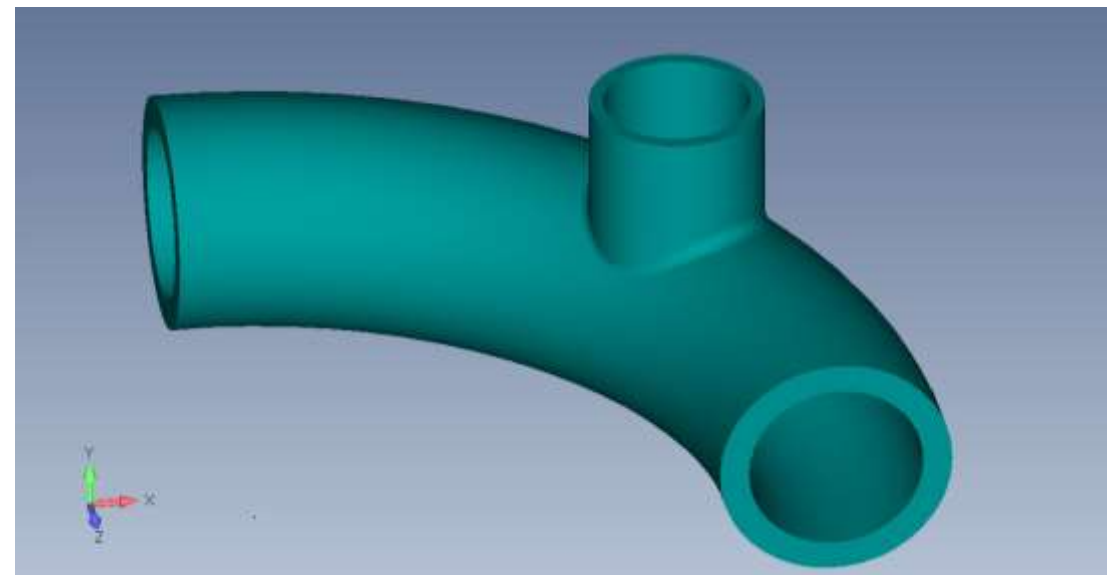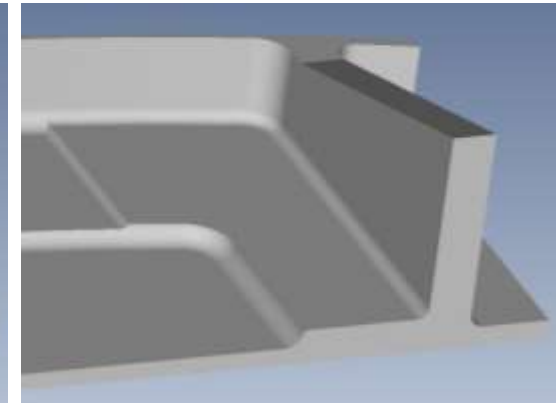
Geometry Editing

➢ Blend Removal is more robust
  ➢ Attemps previous "all at one time"
  ➢ If that fails, removes blend "chains" one at a time
  ➢ Makes a 2nd pass as required
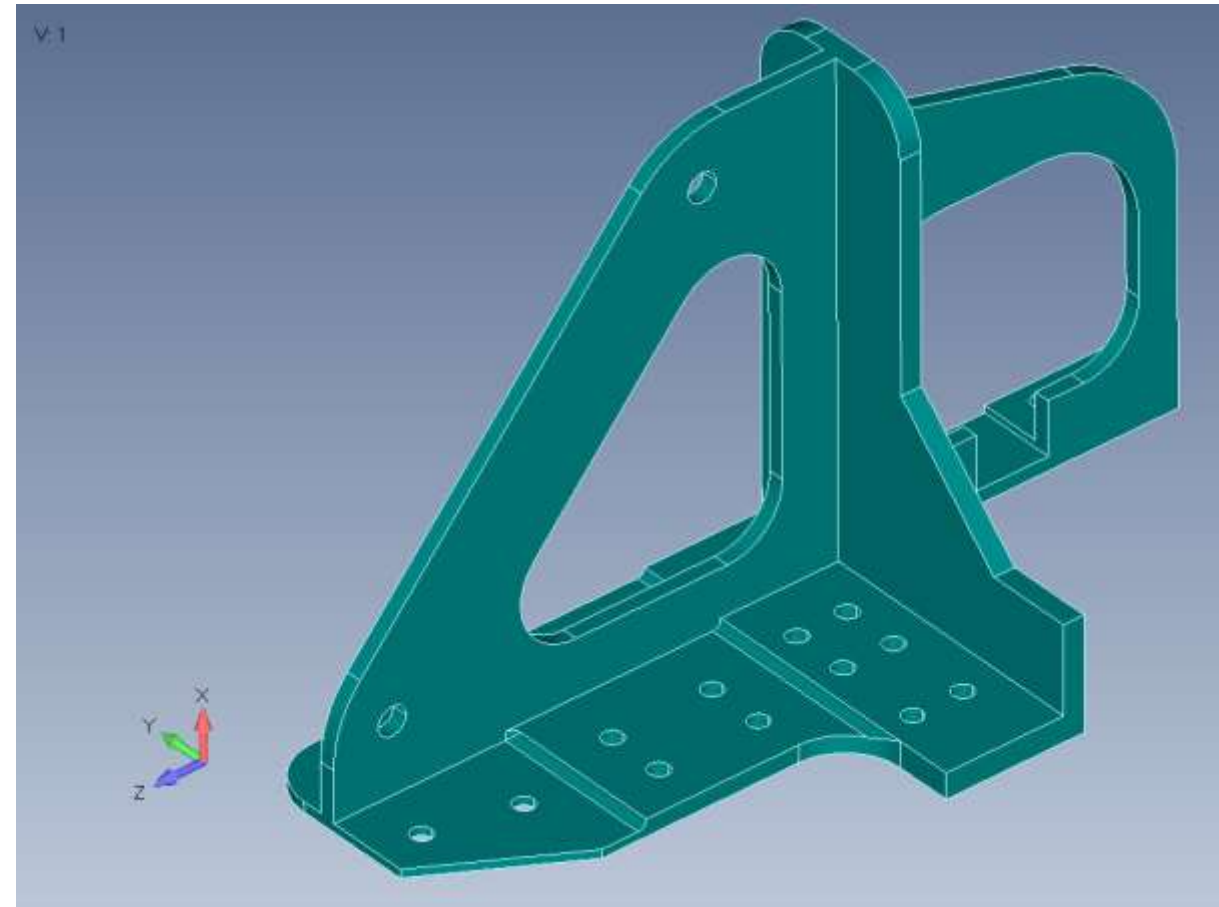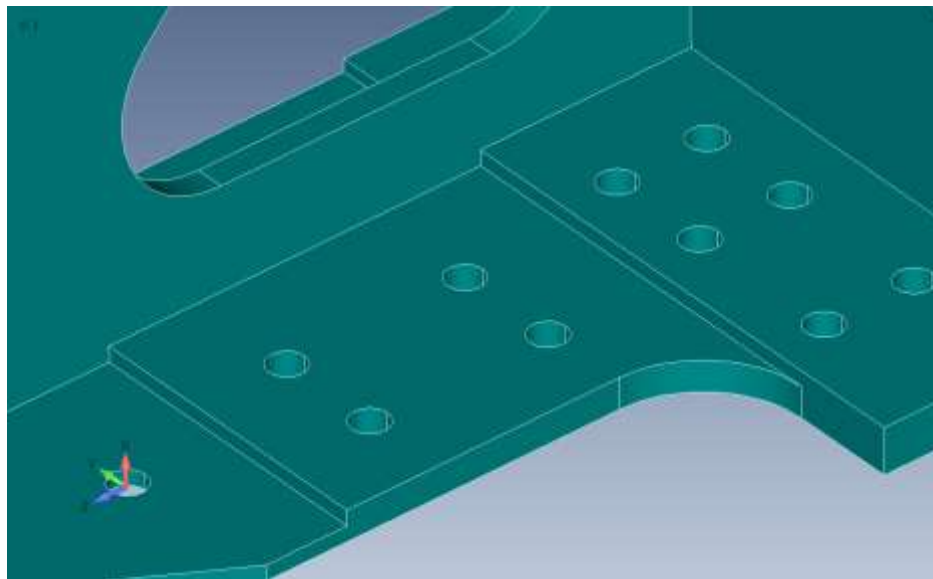
Align Washers Created in the Meshing Toolbox

Geometry Editing

➢ New *Alignment* option when adding washers in the Meshing Toolbox:

- ➢ Automatic
- ➢ Vector
- ➢ Tangent to Curve
- ➢ Perpendicular to Curve
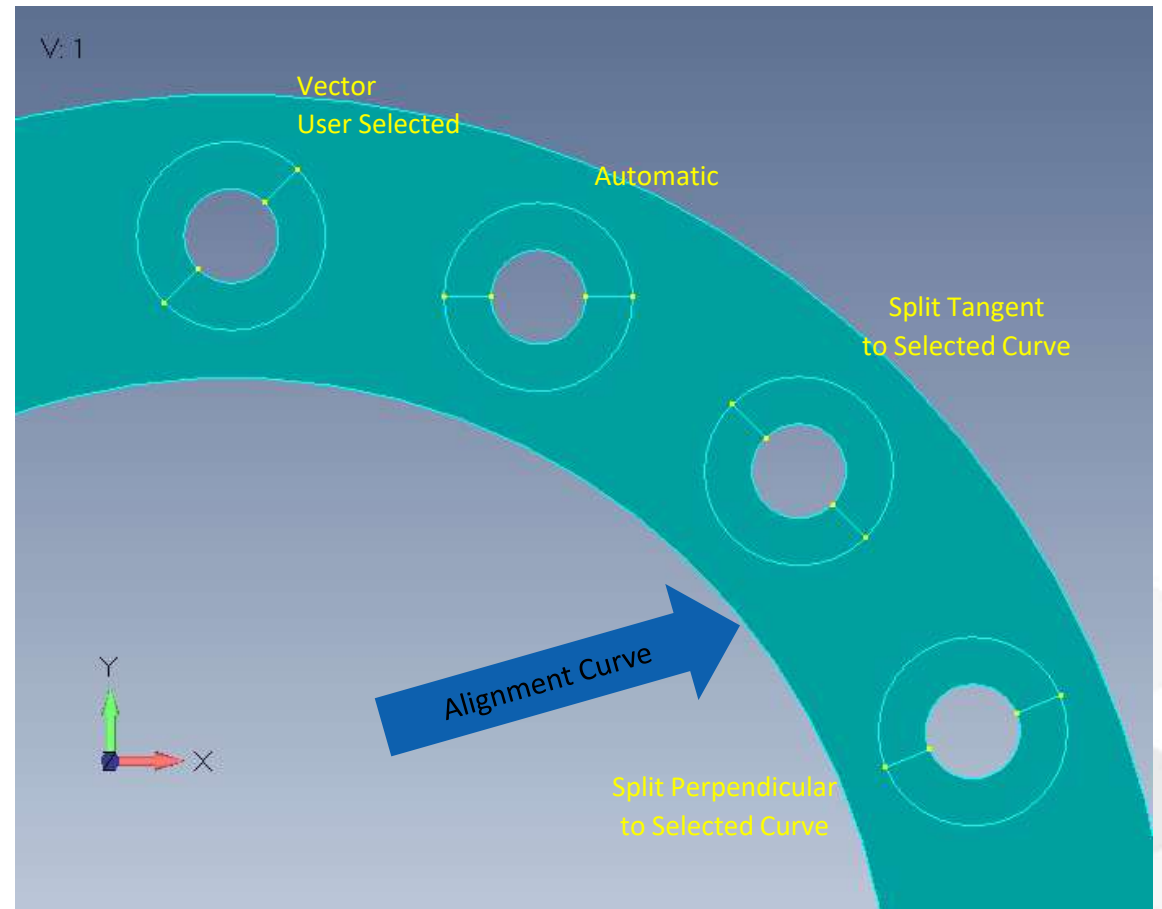
Edge/Skin Elements is Streamlined and Solid Meshing Provides Greater Control over Merged Nodes

Pre-Processing

➢Mesh > Edge/Skin Elements workflow is streamlined
  ➢ Directly pick element edges or faces, then select property to use
  ➢ New *Model Free Faces/Edges* option
  ➢ Prompt to apply line element offsets

➢New *Merge Nodes* options during Mesh > Solid
  0. Off
  1. New Nodes
  2. Into Model
  3. All Nodes

3 = All Nodes – All nodes in model
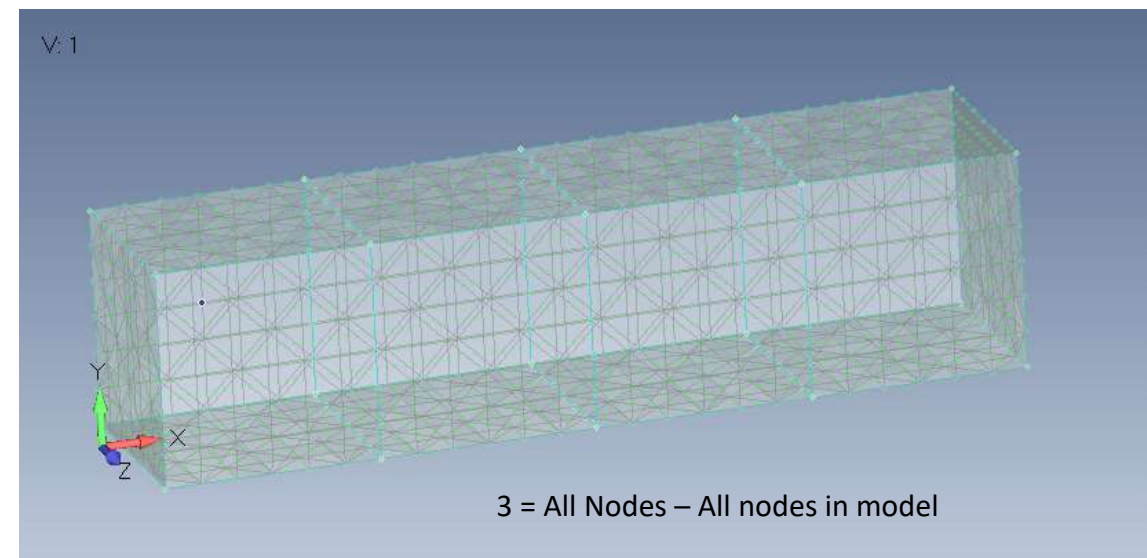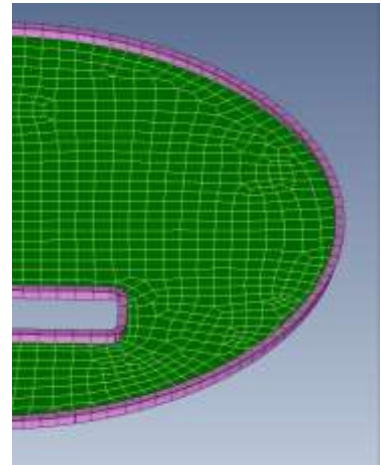
New Options are Available for Rigid Connections and Swept Meshes

Pre-Processing

➢ Mesh > Connect > Rigid Enhancements

  ➢ Specify a minimum and maximum target nodes per source node

  ➢ Choose if target nodes can be mapped to multiple source modes

  ➢ Performance is enhanced (over 30x!)
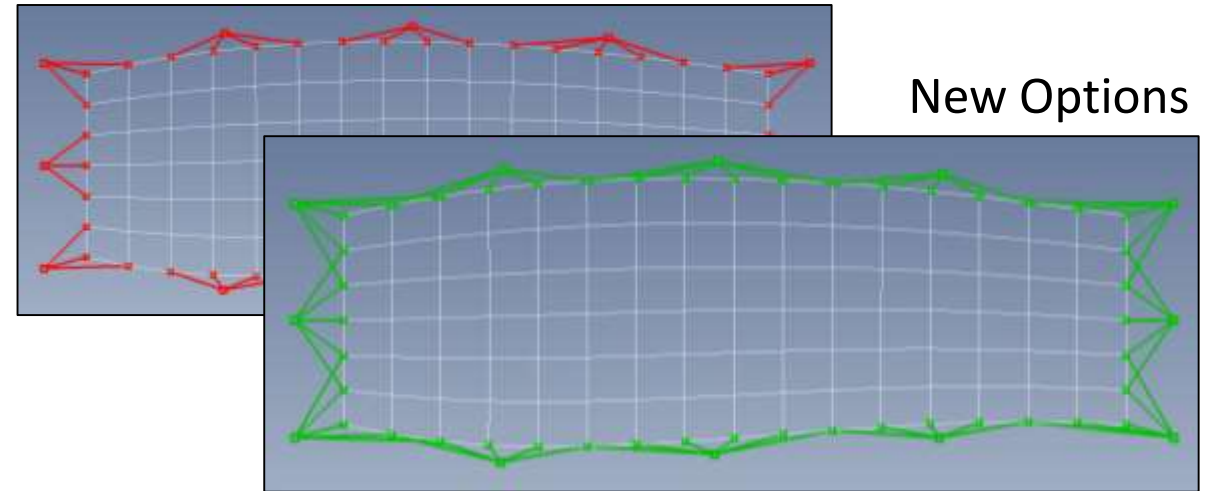
➢ Mesh > Sweep commands have new options for sweeping along a mesh

  ➢ Merge Nodes to Adjacent Elements

  ➢ Limit Sweep to Visible Mesh – Control sweeping over dissimilar meshes

Original

New Options

Check Mesh Interference and Sum Forces for Multiple Load Sets

Pre-Processing

➢ Mesh Interference Command is introduced
  - ➢ Includes cross-sections of 1D elements and thicknesses and offsets of 2D elements
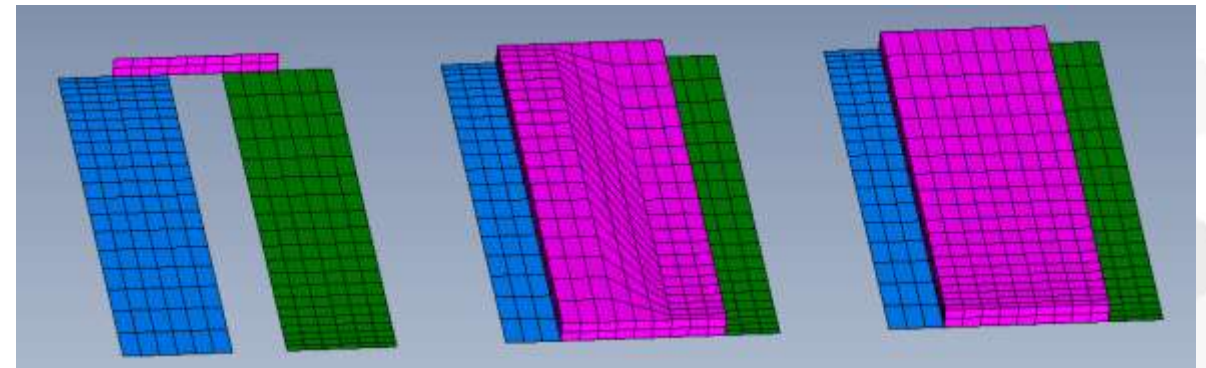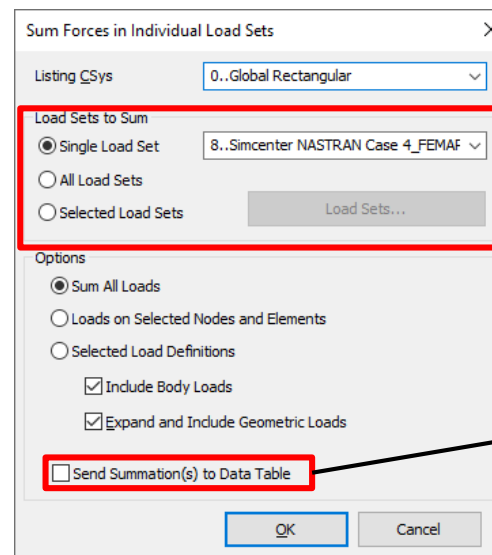 - ➢ Elements are highlighted and optionally written to a group

➢ Sum Forces enhancements:
  - ➢ List individual load summations for multiple Load Sets to the Message window simultaneously
  - ➢ Send load summations summaries to the Data Table

Sum Forces in Individual Load Sets

Listing CSys: 0..Global Rectangular

Load Sets to Sum
- ● Single Load Set: 8..Simcenter NASTRAN Case 4_FEMAP
- ○ All Load Sets
- ○ Selected Load Sets — Load Sets...

Options
- ● Sum All Loads
- ○ Loads on Selected Nodes and Elements
- ○ Selected Load Definitions
  - ☑ Include Body Loads
  - ☑ Expand and Include Geometric Loads

☐ Send Summation(s) to Data Table

OK   Cancel
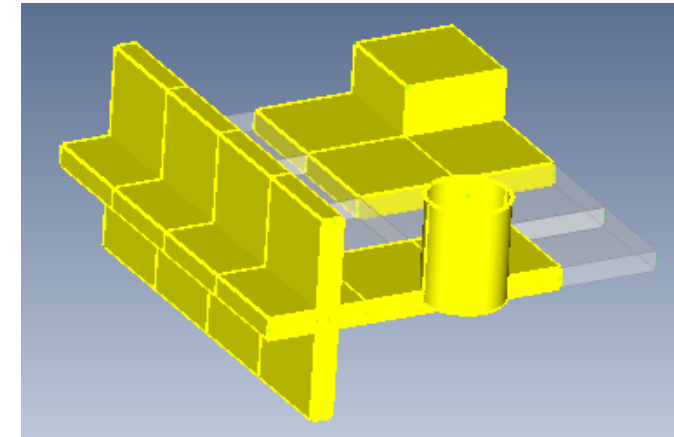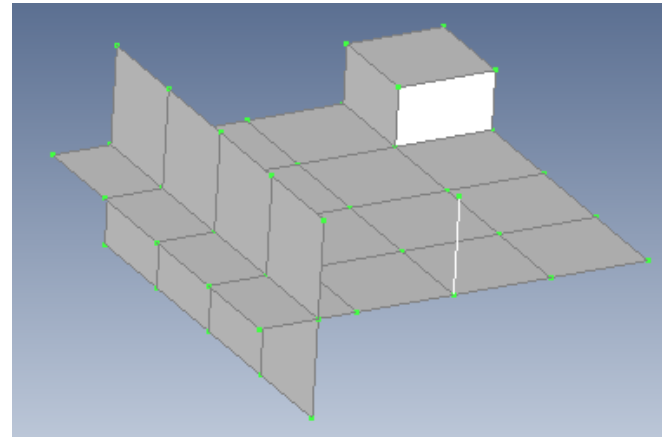
Data Table

Load Set	Total Force, FX	Total Force, FY	Total Force, FZ	Total Moment, MX	Total Moment, MY	Total Moment, MZ
1	0.	1500.	0.	-135000.	0.	23253.75
2	-0.0525418	2943.246	-89.67481	-124125.8	1482.071	50868.56

ATA ENGINEERING, INC.

Pre-Processing

➢Create multiple new Load Sets from multiple Output Sets automatically

  ➢ Optionally, create corresponding Constraint Sets when creating displacement loads

  ➢ Additional options allow for improved face selection for pressure loads

➢Copy Load and Constraint definitions to multiple sets at once with a new RMB command

➢Select which layers and related entities are listed with the List Layers command
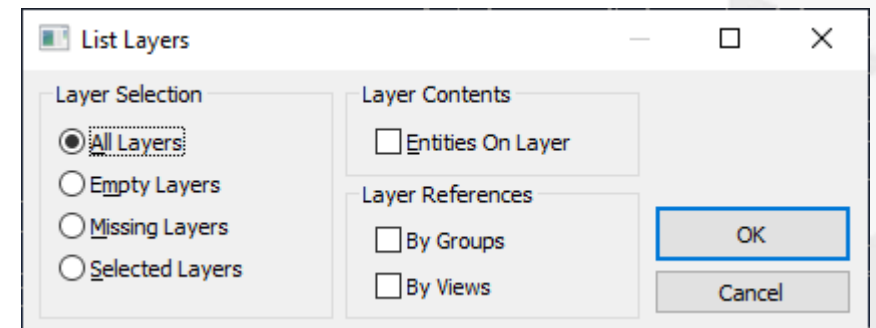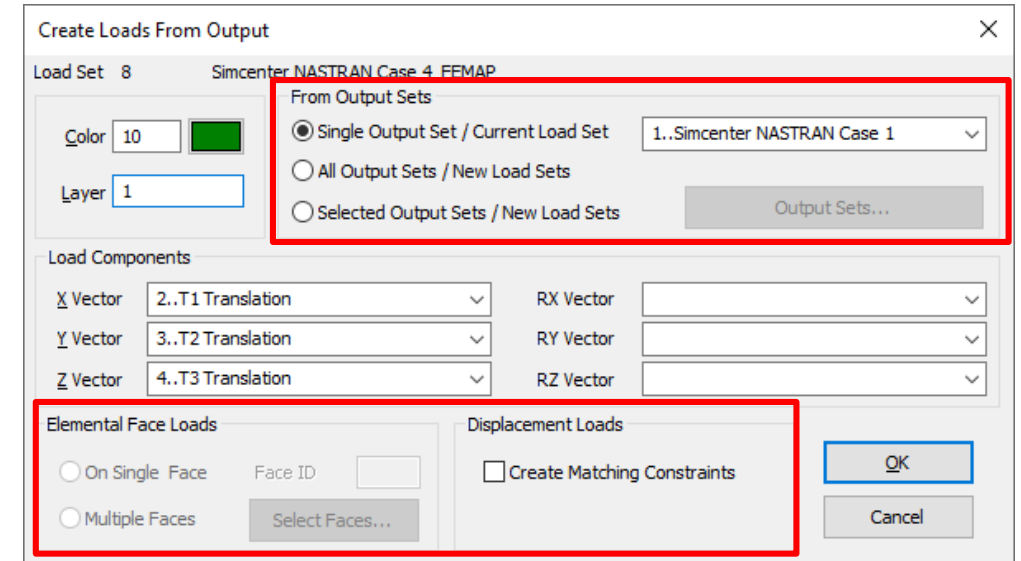
Post-Processing

- ➢ New vectors leave room to support new types of output for many different types of elements
- ➢ Output Vector IDs from pre-2020.1 models will be automatically handled during neutral file conversion
- ➢ API Impacts:
  - ➢ Existing scripts will continue to run with pre-2020.1 IDs, **but should be updated**
  - ➢ The Output Object has been fully deprecated; new APIs will use the Results Browsing Object
  - ➢ New "V2" API methods and properties have been added
  - ➢ API calls have been added to convert IDs to 2020.1 and vice versa
 - ➢ New API calls can specify output with pre-defined quantities like "Top von-Mises Stress"

Output Vector Range Comparison		
Output Type	**Vector Ranges from 9.3 through 2019.1**	**Vector Range 2020.1 and Above**
Nodal output	1 to 2,999	1 to 2,999
Line element output	3,000 to 5,999	3,000 to 5,999
Plate element output	6,000 to 7,599	6,000 to 9,999
Plate corner output	100,000 to 299,999	10,000 to 25,999
Not currently used	7,600 to 59,999	26,000 to 59,999
Solid element output	60,000 to 69,999	60,000 to 69,999
Solid corner output	70,000 to 73,999	70,000 to 79,999
Output on any element type	80,000 to 89,999	80,000 to 89,999
PATRAN elemental output	90,000 to 99,999	90,000 to 99,999
Not currently used	300,000 to 999,999	100,000 to 999,999
Plate Laminate Ply output	1,000,000 to 1,999,999	1,000,000 to 3,999,999
Plate Laminate Ply corner	2,000,000 to 5,999,999	4,000,000 to 15,999,999
Solid Laminate Ply output	6,000,000 to 6,499,999	16,000,000 to 17,499,999
Solid Laminate Ply corner	6,500,000 to 8,499,999	17,500,000 to 23,499,999
User Defined output	9,000,000 to 9,999,999	24,000,000 to 24,999,999
Complex Phase output	Add 10,000,000 to above ranges	Add 25,000,000 to above ranges
Complex Real output	Add 20,000,000 to above ranges	Add 50,000,000 to above ranges
Complex Imaginary output	Add 30,000,000 to above ranges	Add 75,000,000 to above ranges

ΛTΛ
ENGINEERING, INC.

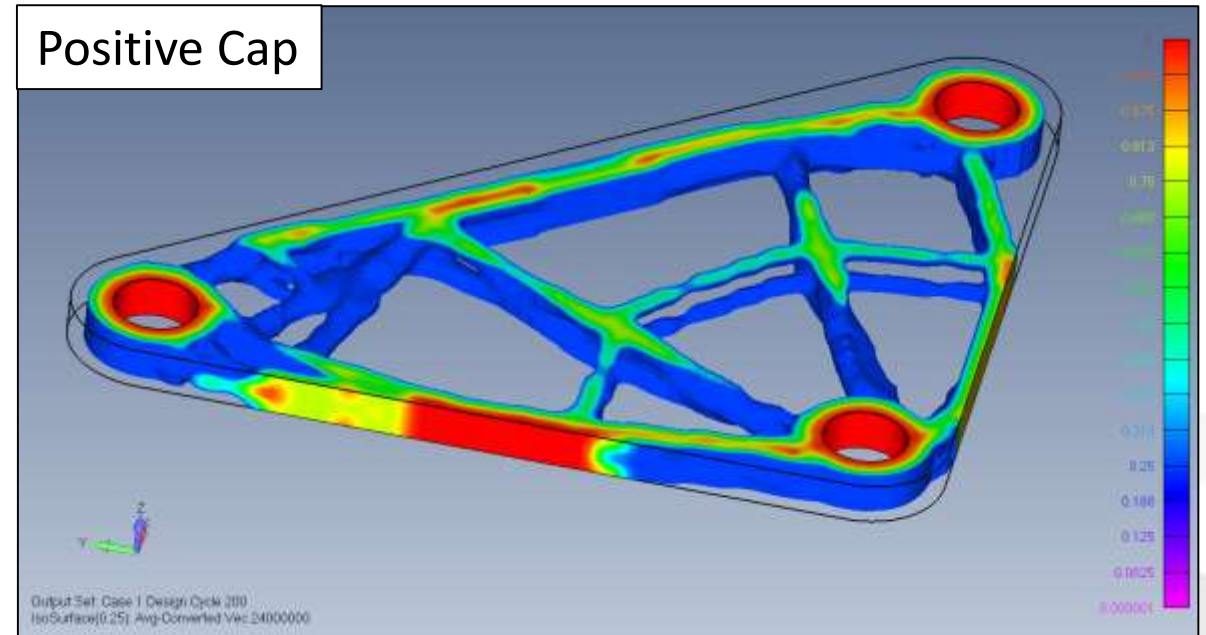# Set Positive and Negative Caps for Iso-Surface Plots

Post-Processing

➢ Positive/Negative iso-surface plots show a value and all above/below values
  ➢ Could be useful for things like Topology Optimization
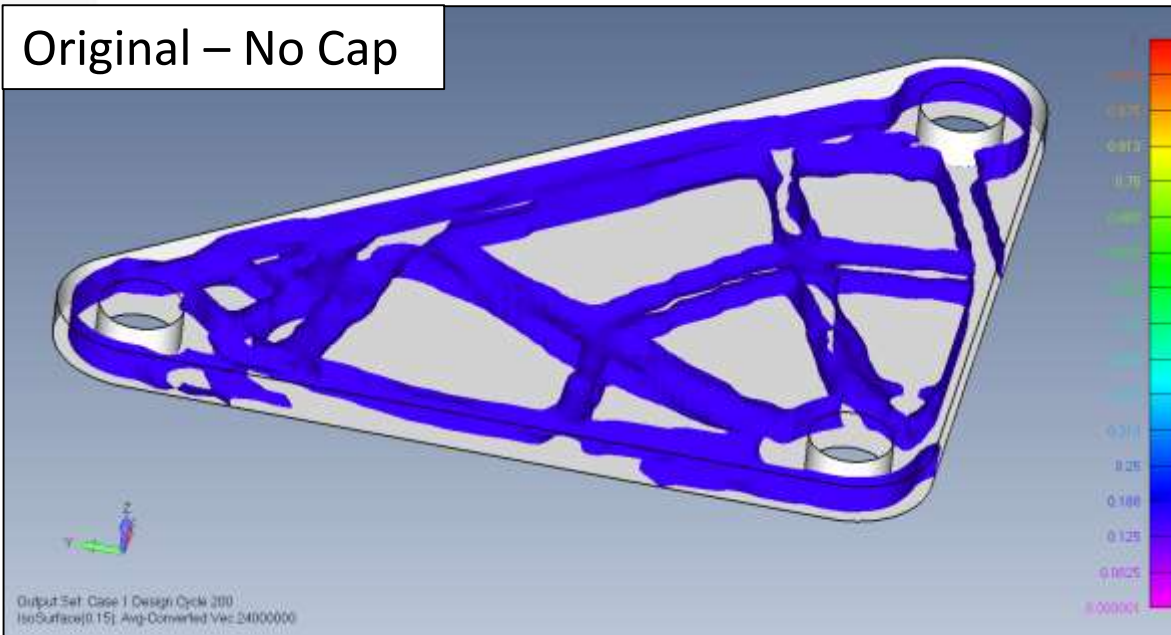
Simcenter Nastran

Solver Support

➢ Ability to handle Large IDs (above 99,999,999)

➢ Creation and read/write of:
  - ➢ Monitor Points (MONPNT1 and MONPNT3)
  - ➢ DMIG entities - Internal as FEMAP Matrix Input Entities or external attached PCH
  - ➢ BOLTFRC outside of SOL401/402

➢ New nonlinear control options for SOL 401/402:
  - ➢ Restart from end of subcase or last converged time step
  - ➢ Disable specific categories of mechanical loads (SOL 401)
  - ➢ Specifcy version to use for parameter defaults (SOL 402)

# Updated Support for Entities in ANSYS, ABAQUS, and LS-DYNA

Solver Support

- ➢ ANSYS Translator Support:
    - ➢ Multiple excitations during random analysis
    - ➢ Variable Shell Thickness
    - ➢ Element and node selections are more completely supported when reading input files and become Femap groups
    - ➢ Writing a portion of a model is supported similar to Simcenter Nastran

- ➢ LS-DYNA Translator Support:
    - ➢ Variable Shell Thickness
    - ➢ Writing a portion of a model is supported similar to Simcenter Nastran
    - ➢ Timoshenko Beams (ELFORM=13)

- ➢ ABAQUS Translator Support
    - ➢ No longer support fixed field format files (incompatible with solver)
    - ➢ Add import of *CONNECTOR SECTION types BUSHING, BEAM elements
    - ➢ Better Import/Export of entity titles and names
    - ➢ Import/Export support of DLOAD, TRVEC

Questions?

Submit questions in the chat or unmute yourself now

Zoom Application
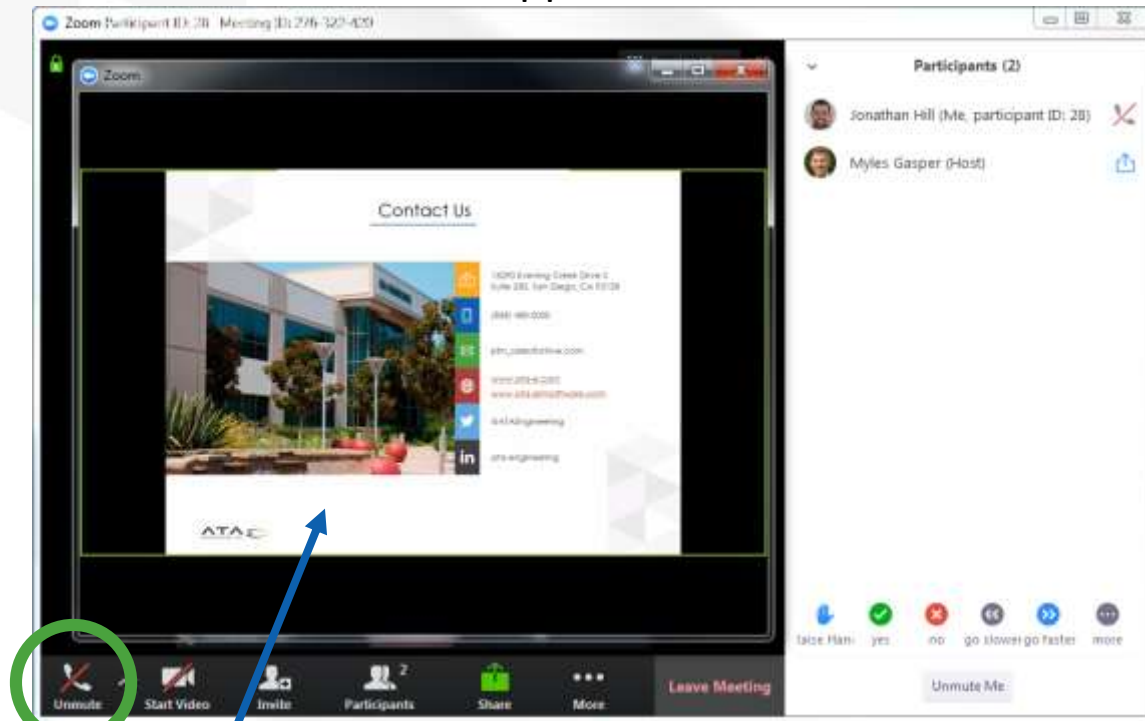
Web Interface
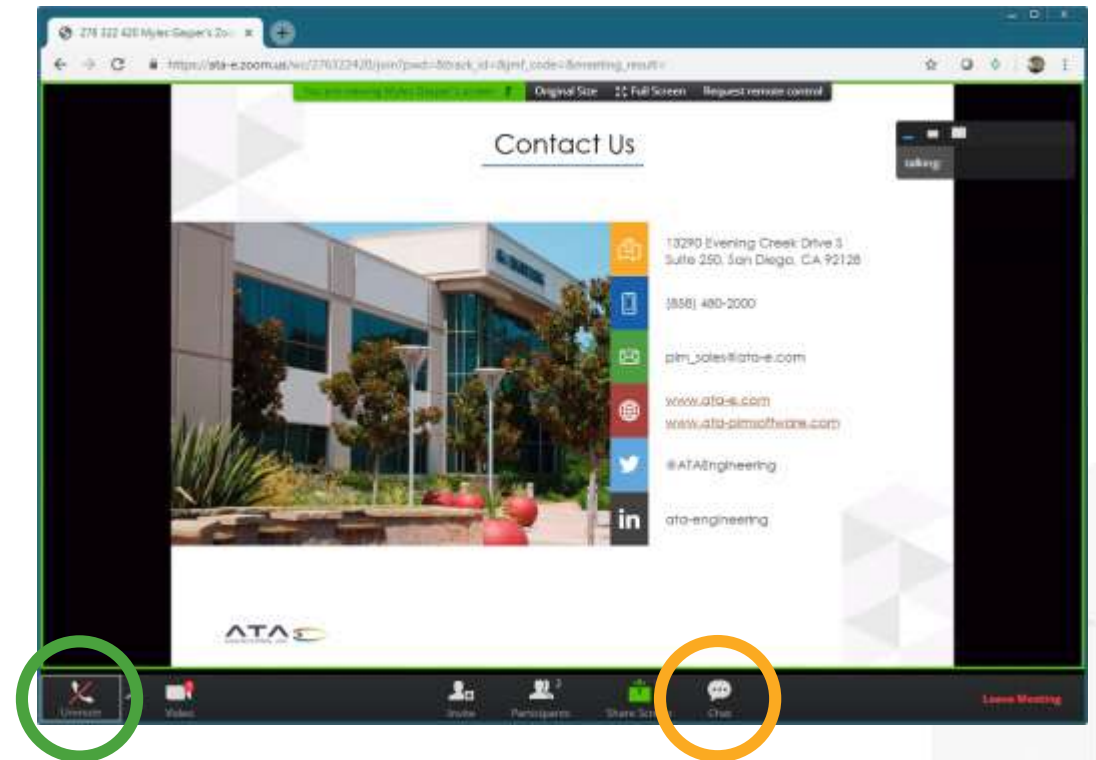

Screenshare in
separate window

Chat is available
under More

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