Webinar:
What's New in Femap 2019.1

Jonathan Hill, ATA Engineering

July 10th, 2019

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

📞 (858) 480-2000

💼 ata-engineering

🌐 www.ata-e.com

🐦 @ATAEngineering

ATA Provides High-Value Engineering Services

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

Aerospace

Robotics & Controls

Themed Entertainment

Defense

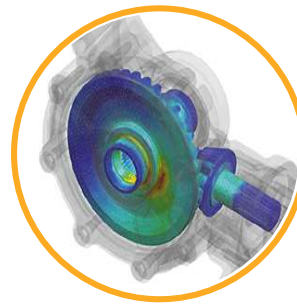Industrial & Mining Equipment

Consumer Products

We Offer Complete, Integrated Solutions

With expertise in design, analysis, and test, ATA engineers regularly work across disciplines to find the optimal solution

Design

From initial concept development to detailed structural design

Analysis

Comprehensive structural, fluid, acoustic, and thermal analysis services

Test

Industry-leading structural test services for extreme loading environments

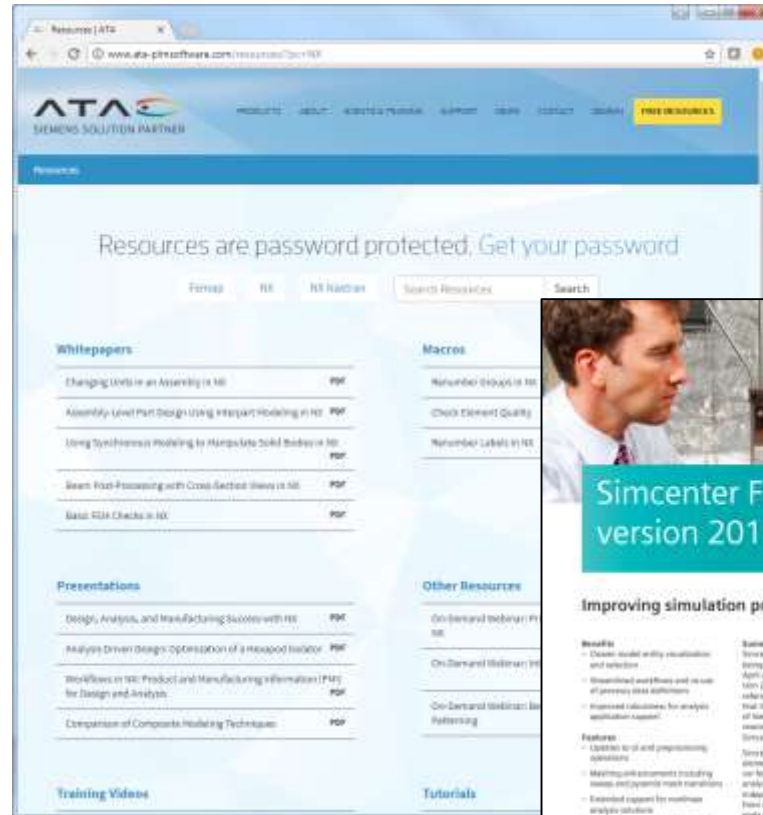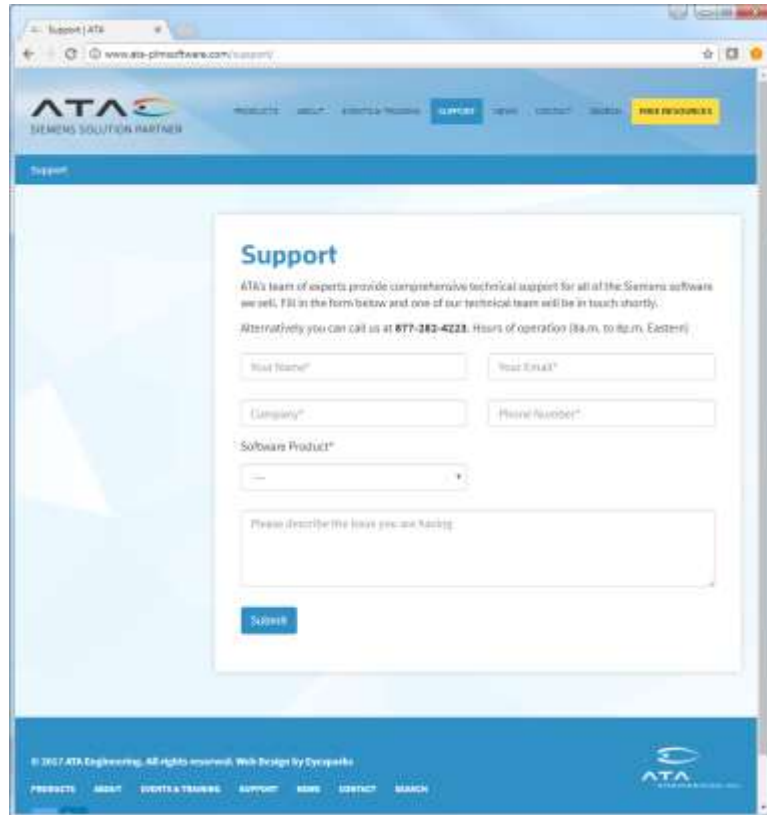# ATA is a Value-Added Reseller for Siemens PLM Software

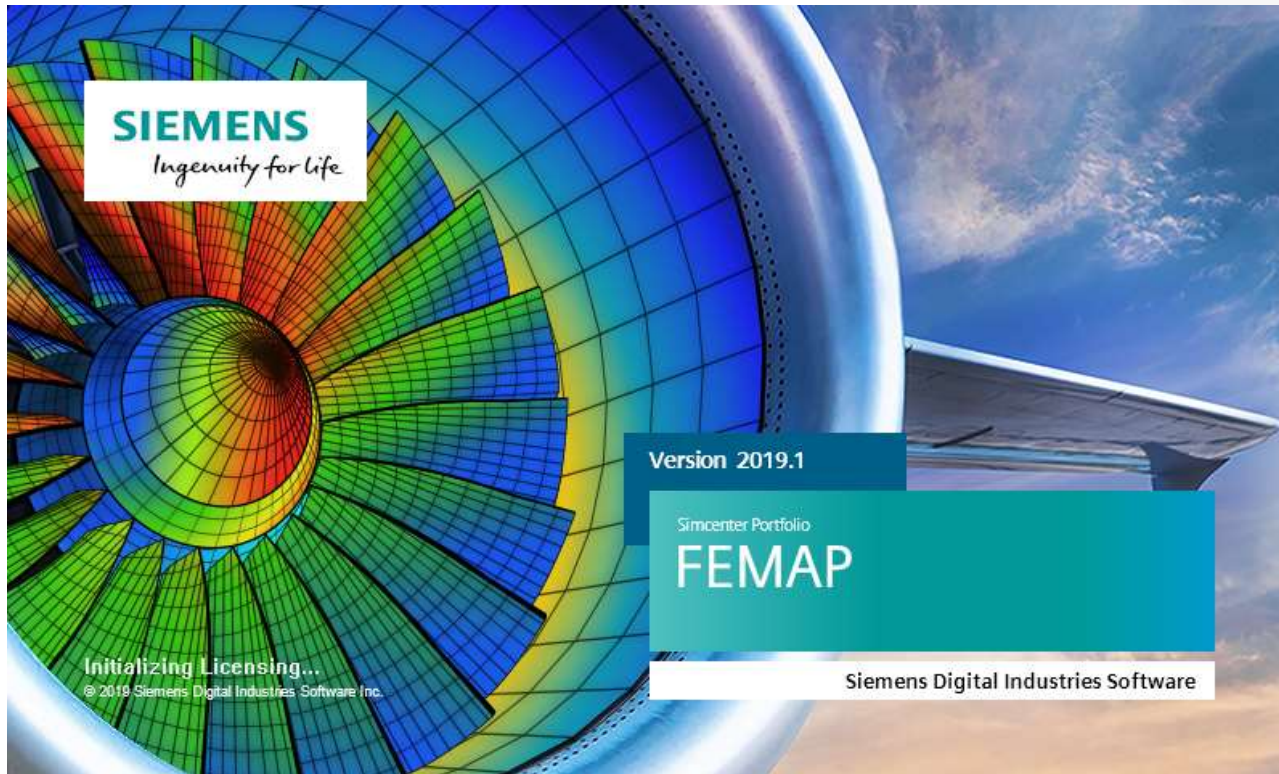ATA offers training, free resources, and hotline support for a variety of Siemens products.

➢ 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

Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com

Webinar:
What's New in Femap 2019.1

Jonathan Hill, ATA Engineering

July 10th, 2019

13290 Evening Creek Drive S, Suite 250, San Diego CA 92128
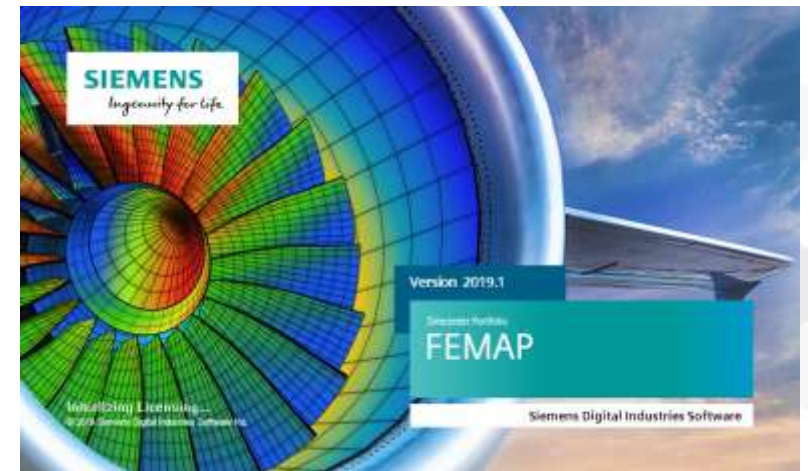
📞 (858) 480-2000   🌐 www.ata-e.com

💼 ata-engineering   🐦 @ATAEngineering

Agenda

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

Femap 2019.1 is Available Now

➢ Femap 2019.1 has been released and is available for download through GTAC:

  ➢ 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
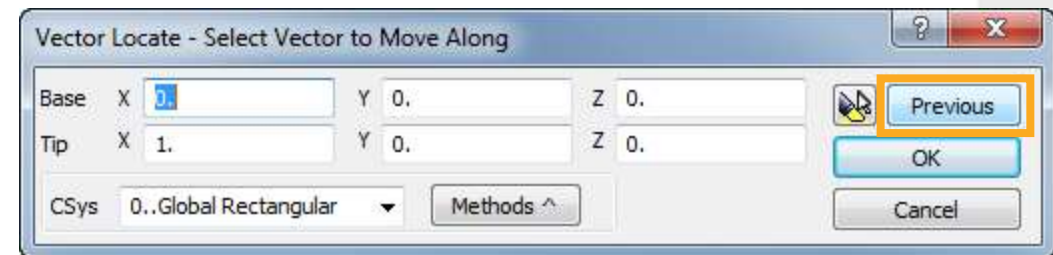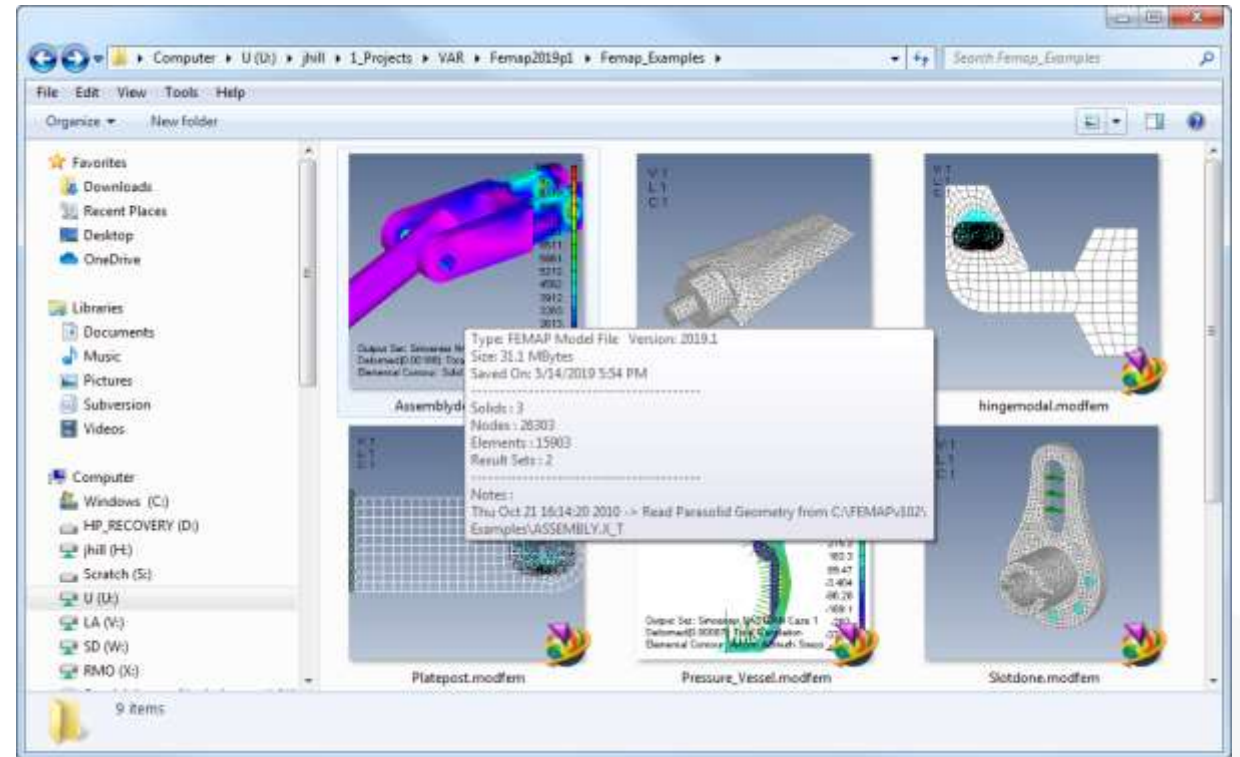 ➢ Flexible customization tools to streamline analysis processes

Femap 2019.1 Introduces a New Naming Convention and Release Cadence

➢ Femap's rebranding as **Simcenter Femap** reflects its inclusion in the Simcenter portfolio of Siemens CAE products

    ➢ NX Nastran is similarly rebranded as **Simcenter Nastran**

    ➢ Rebranding does not inherently alter Femap or Nastran outside of new release updates and enhancements

➢ **Simcenter Femap 2019.1** supersedes Femap 12.0

➢ **Simcenter Nastran 2019.1** supersedes NX Nastran 12

➢ Femap will be released on a biannual cadence every April and October

    ➢ Feature releases will add new features and expand functionality

    ➢ Feature releases will be accompanied by database changes and will require licensing updates

    ➢ Maintenance fixes will be released as needed

File Preview and Previous Vectors

➢Models and basic entity statistics are available for preview in File Explorer:
  ➢Femap Version
  ➢Size
  ➢Save date
  ➢Number of entities
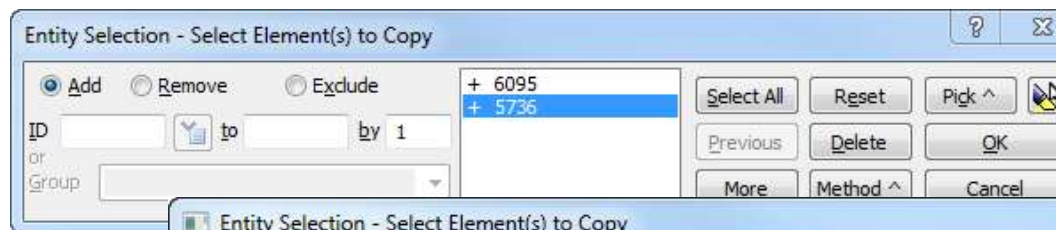 ➢Model notes from File > Notes

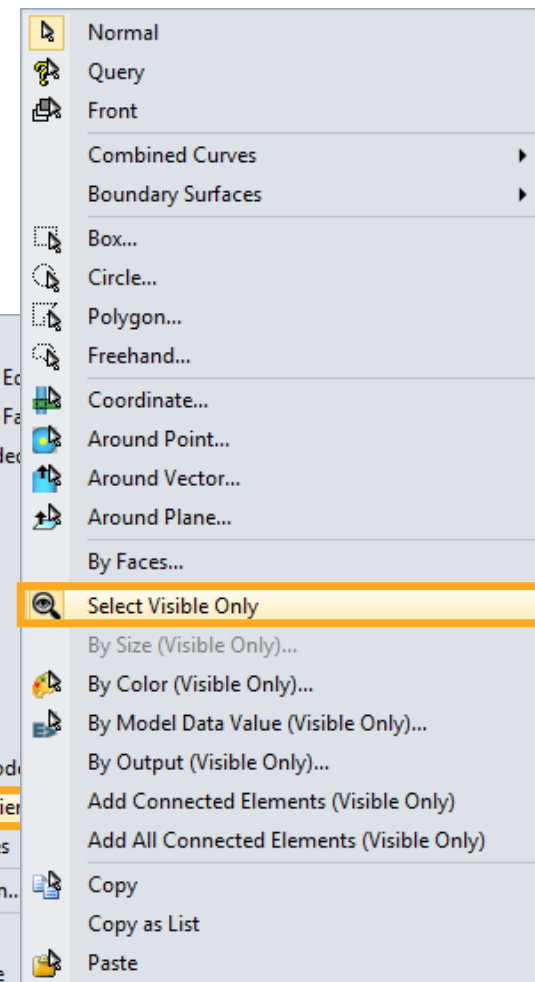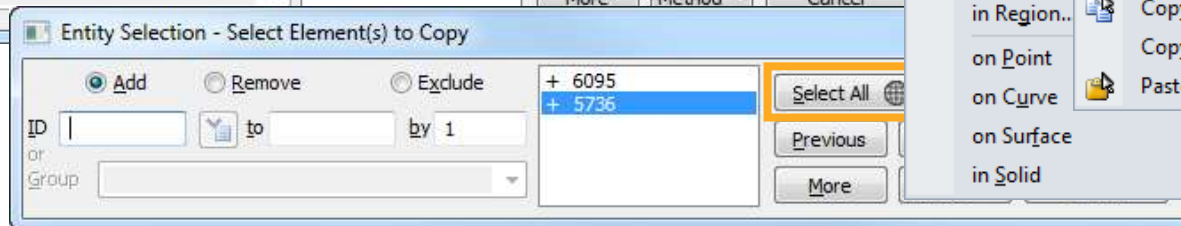➢Previous vectors can be recalled

Visualization and User Interface

Locate in Model Info and Entity Selection

➢Locate in Model Info option highlights selected entities in the model info pane

➢Entity selection enhancements include:
  ➢New Icons and more ID space
  ➢Select Visible button
  ➢Select Visible Only pick option
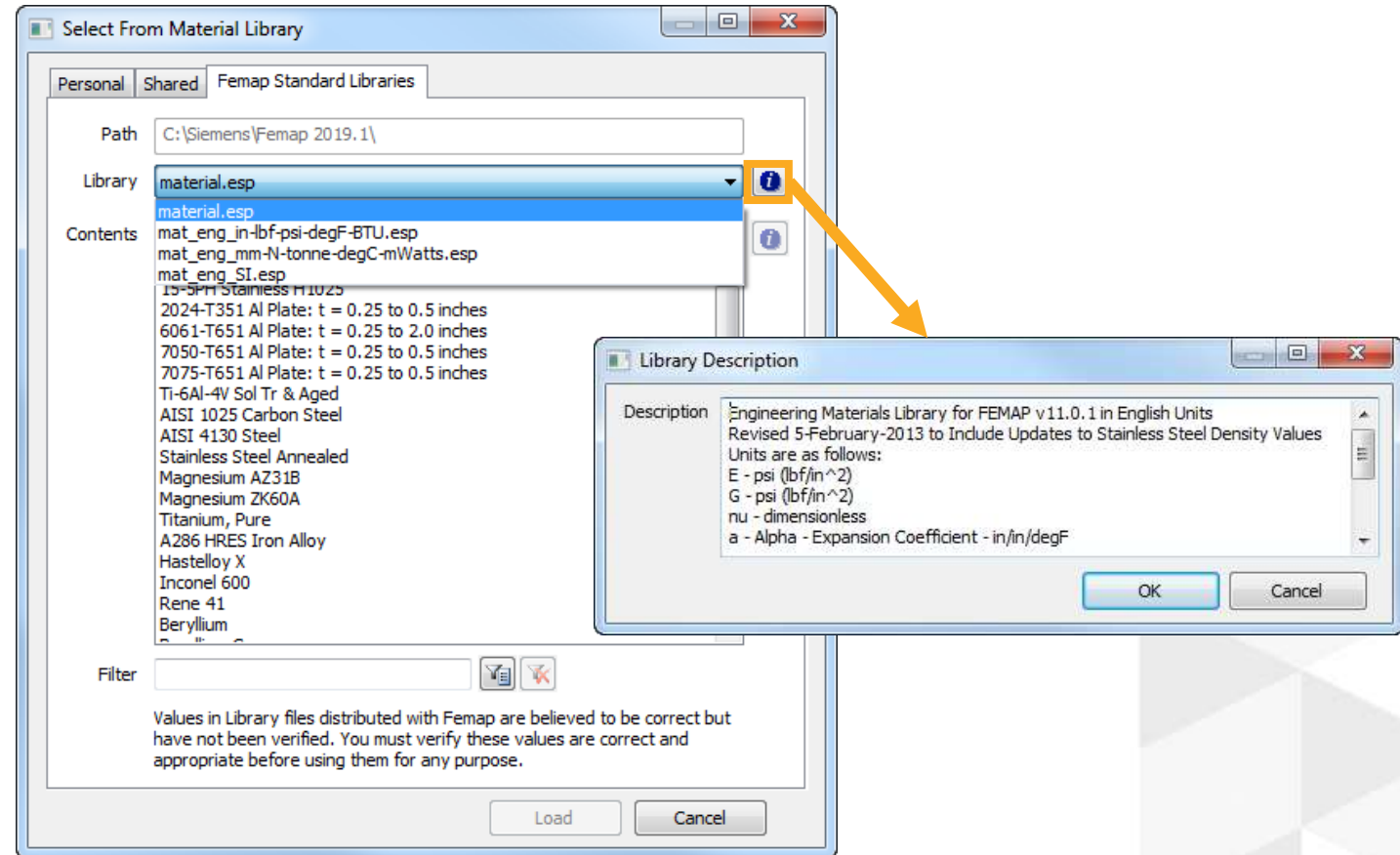 ➢Using Orientation Node method

Femap 12

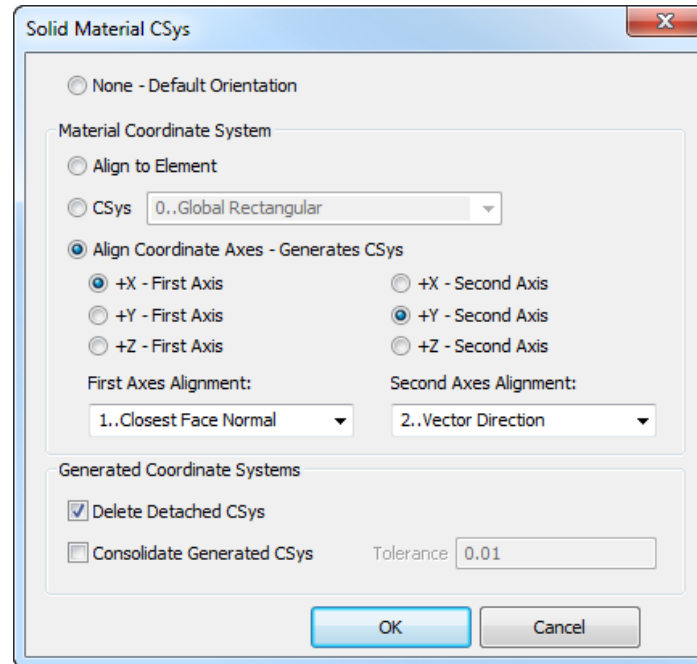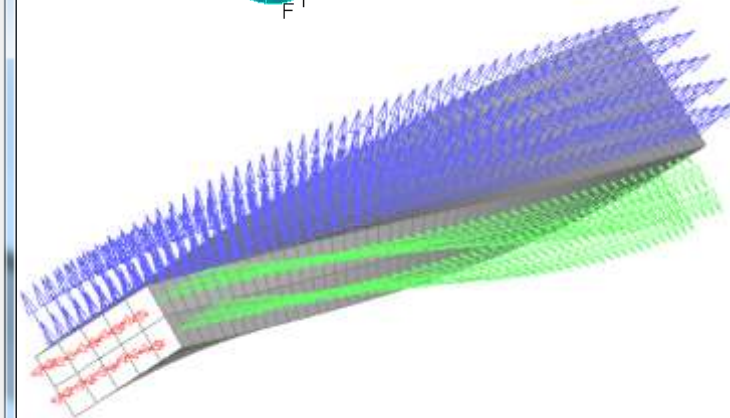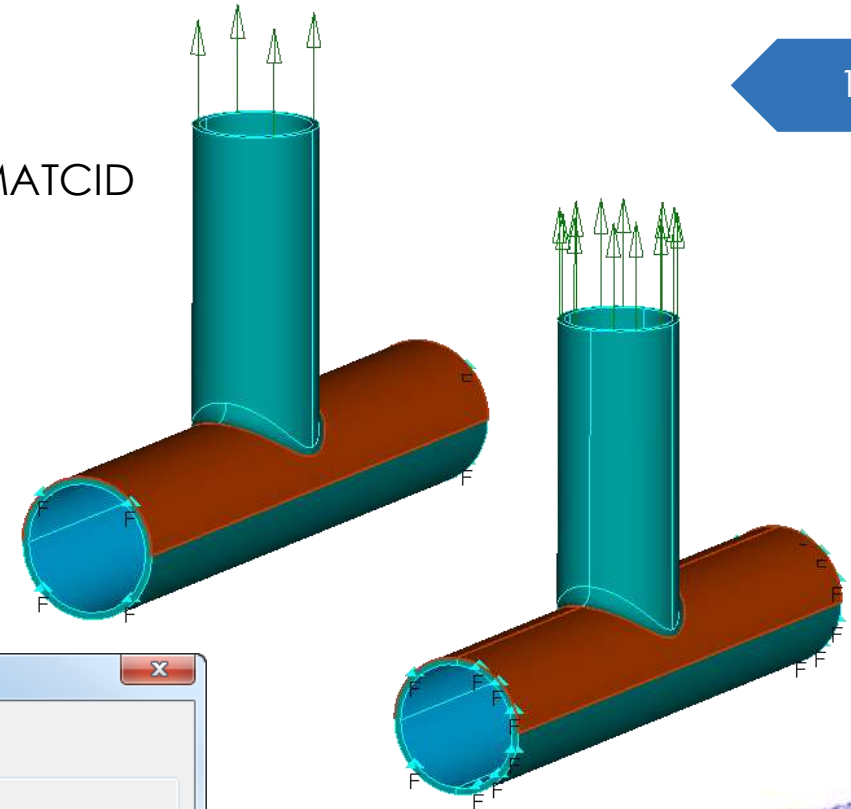Femap 2019.1

Libraries and Project Nodes/Points

➢ Material and property libraries offer greater flexibility
- ➢ Quickly switch between standard Femap libraries, personal libraries, and shared libraries

➢ Project points and nodes radially about an axes or spherically about a point
- ➢ Modify > Project > Node/Point
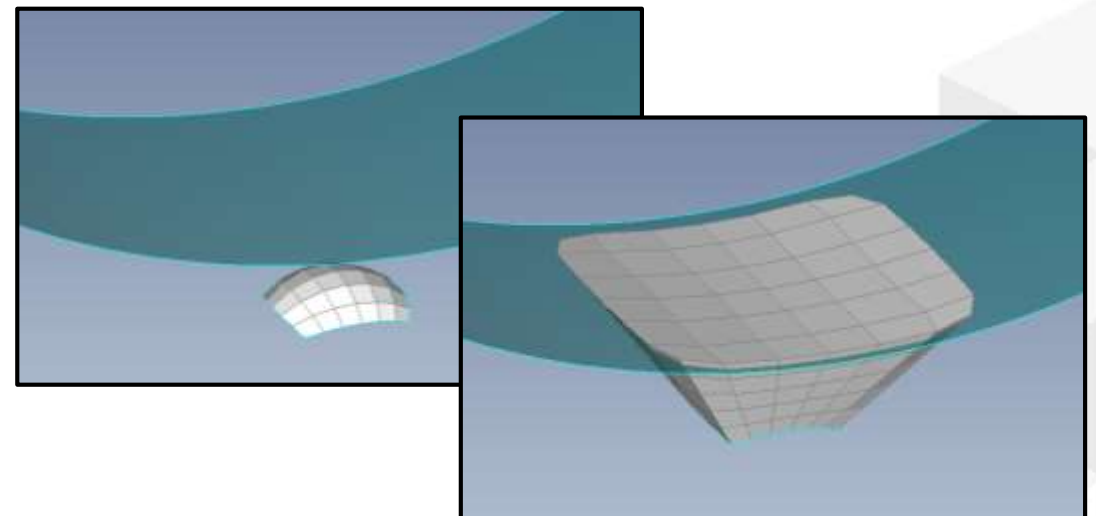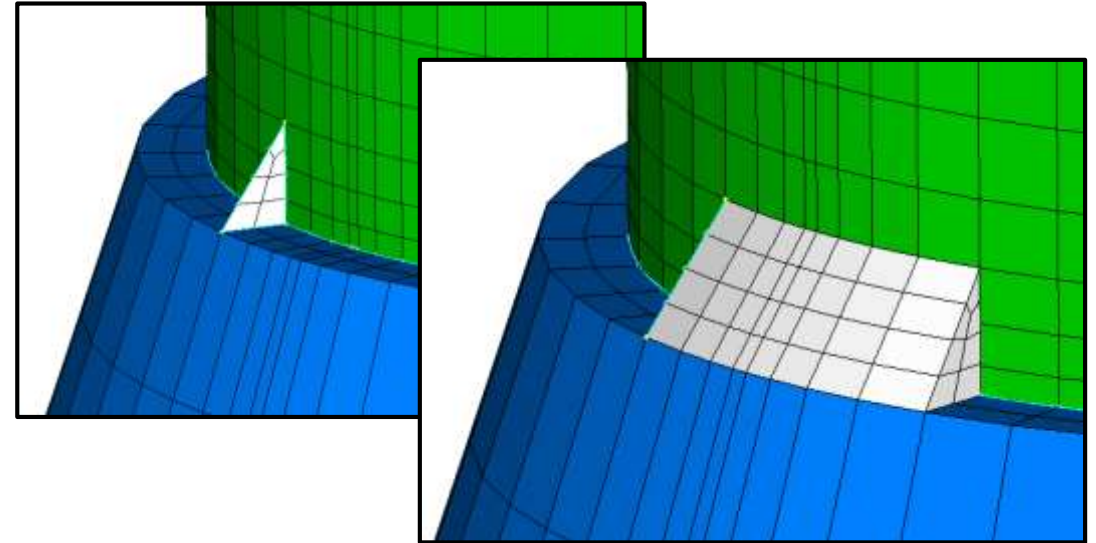
Geometry Split, Force on Curves, and MATCID

➢ Geometry-based definitions are propagated during geometry split operations:
  - ➢ Loads, Constraints and Regions
  - ➢ Colors, layers, and solid attributes

➢ Bearing Force on Curve and Torque on Curve loads are available
  - ➢ Similar to Bearing Force/Torque on Surface commands

➢ Material Coordinate Systems can be specified on solid elements (MATCID card)
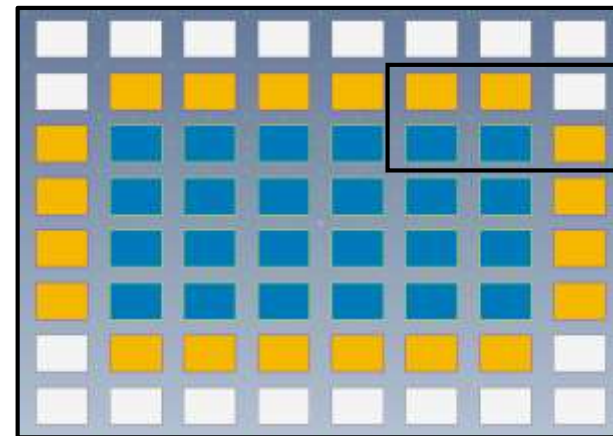 - ➢ Modify > Update Elements > Solid Material CSys...
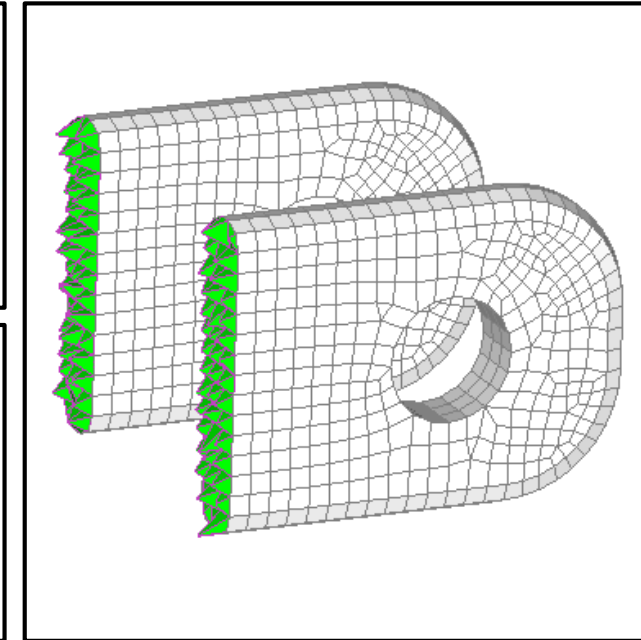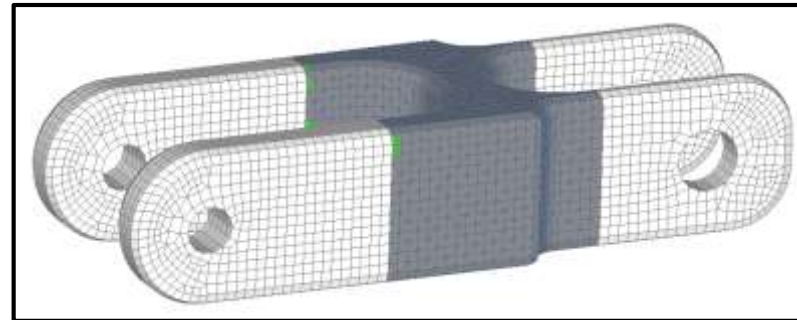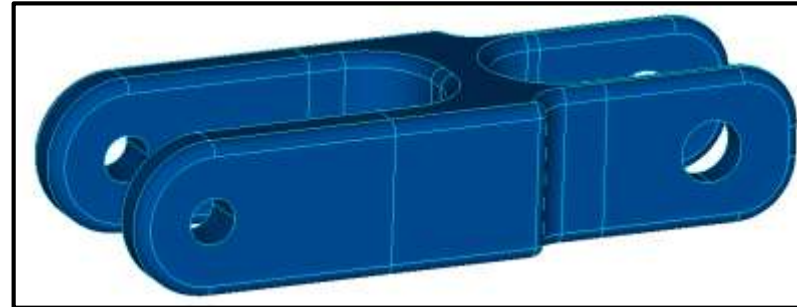
Element Sweep and Extrusion

➢Sweep along element edges
  ➢Select a single element edge
  ➢Sweep operation continues until it connects back on itself, the end is reached, or a stop node is specified
  ➢As elements are swept, the mesh morphs to maintain a match with existing mesh edges

➢Extrude elements along element normals to selected surfaces

➢Extrude shell elements in both directions

Element Transitions

- ➢ Automatically generate pyramid elements to transition between hexagonal and tetrahedral elements
  - ➢ Femap will manage node connections between linear bricks and parabolic tetrahedral elements
- ➢ Connect mixed linear and parabolic elements
  - ➢ Choose between removing midside nodes on parabolic side or adding them on the linear side
  - ➢ Modify > Update Elements > Connect Linear/Parabolic Elements

Solver Support

Simcenter Nastran

➤ Femap 2019.1 comes with Simcenter Nastran 2019.1 (January 2019)

➤ New SOL 401 and SOL 402 sequences:
  - ➤ Nonlineaer dynamics
  - ➤ Linearized buckling
  - ➤ Arc-length methods in statics

➤ SOL 401 allows for bolt preload on 1D elements, supports PCOMP and PCOMPG, offers more robust contact modeling, and has a new transient dynamic subcase

➤ SOL 200 supports frequency response analysis

➤ Additional enhancements:
  - ➤ Numerical improvements to RDMODES solutions
  - ➤ Extension of shared memory parallel (SMP) to element matrix assembly (EMA) operations
  - ➤ Automatic selection of numerical improvements for frequency response

➤ The STATSUB case control command gives greater flexibility when setting up SOL 105 buckling analyses.
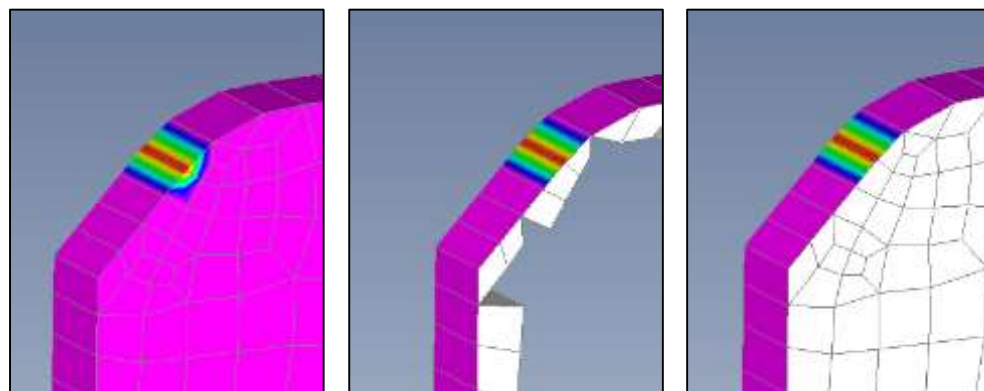
Solver Support

ANSYS and ABAQUS

➢ ANSYS translator adds read/write support for:
 ➢ Non-normal pressure definitions
 ➢ Beam preload and solid preload
 ➢ Additional commands for random vibration analysis, including SPOPT, PSDUNIT, PSDFRQ, PSDVAL, PFACT, and PSDCOM
 ➢ Output control KEYOPT(8) for laminates and solid laminates

➢ ABAQUS translator allows users to specify thermal options for gap elements and contact
 ➢ Accessed via the ABAQUS Thermal button

Contour Plots and Flutter

- ➢ New contour plot options for partial results like contact forces
- ➢ List > Output > Format consolidates:
  - ➢ List > Output > Standard
  - ➢ List > Output > Use Format
- ➢ Aero mesh displacement results from flutter analysis can be displayed
  - ➢ Modes data is read from the .f06 output file
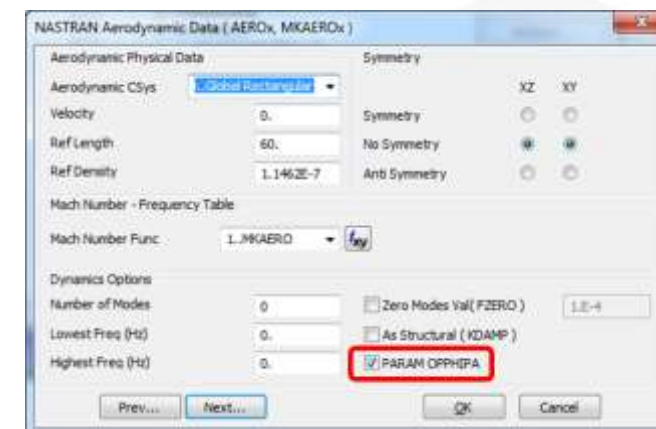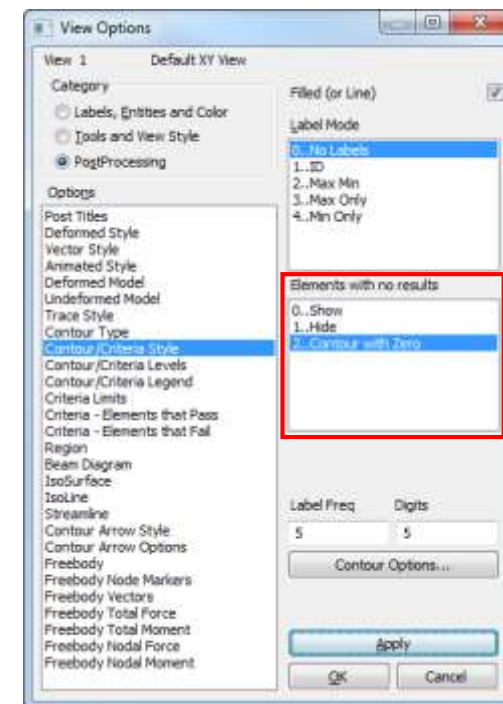 - ➢ PARAM,OPPHIPA must be set to 1 for splined aero data
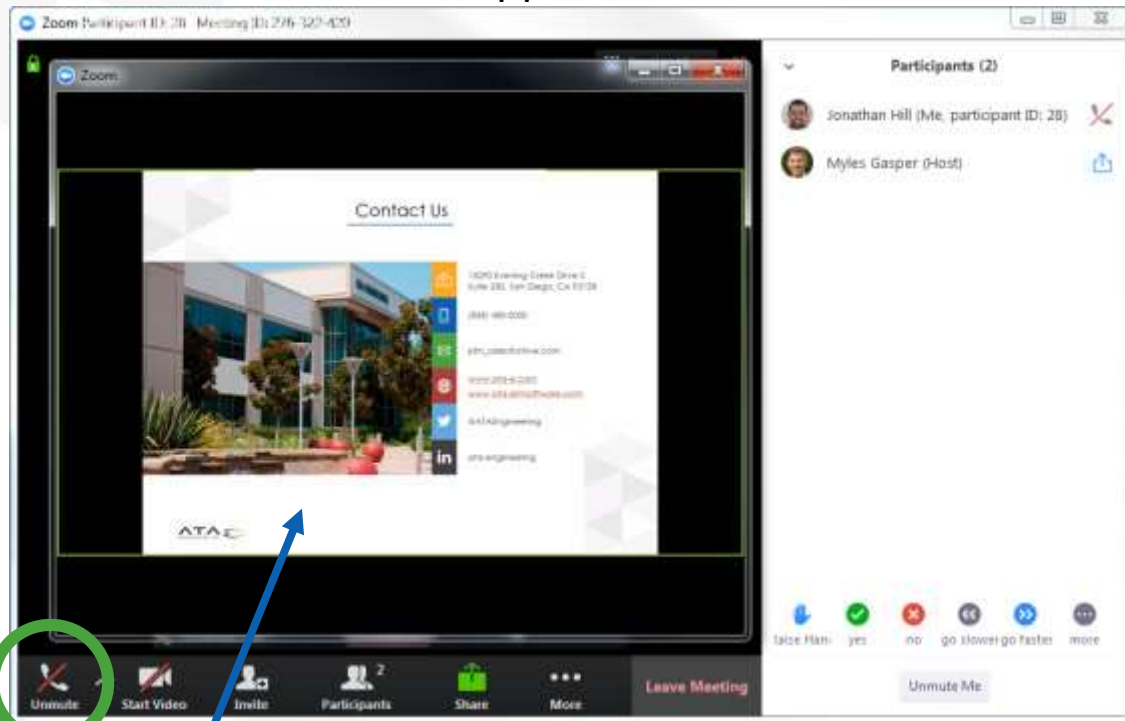

Deformed Structural Mesh

Deformed Aerodynamic Mesh

Deformed Combined Mesh

Questions?

Submit questions in the chat or unmute yourself now

Zoom Application

Web Interface
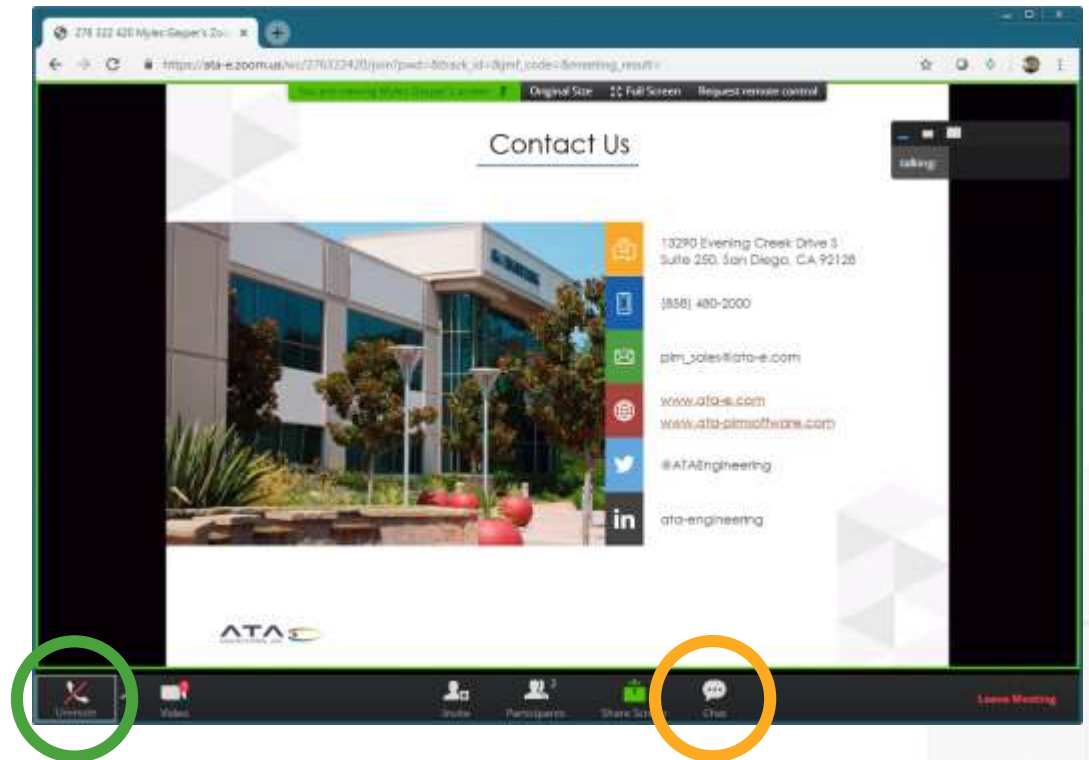

Screenshare in separate window

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