




Webinar: **Modeling Bolted Joints in Femap**


Rachel Backes, ATA Engineering
March 31, 2021

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

 (858) 480-2000

 ata-engineering

 www.ata-e.com

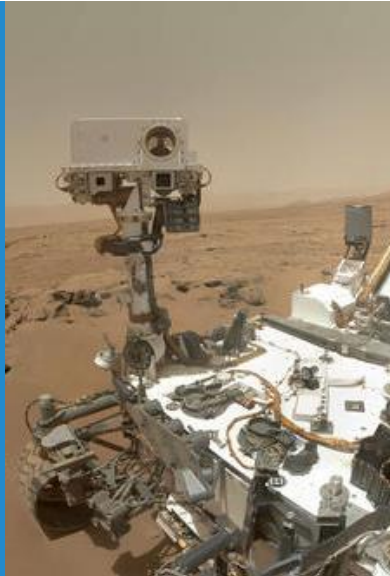
 @ATAEngineering

ATA Provides High-Value Engineering Services 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



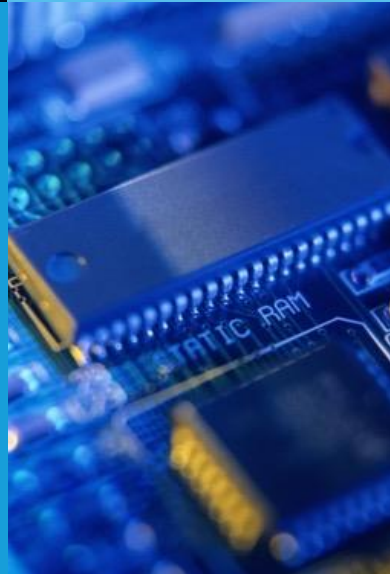
Defense



Industrial & Mining Equipment



Consumer Products



ATA is a Value-Added Reseller for Siemens Digital Industries Software

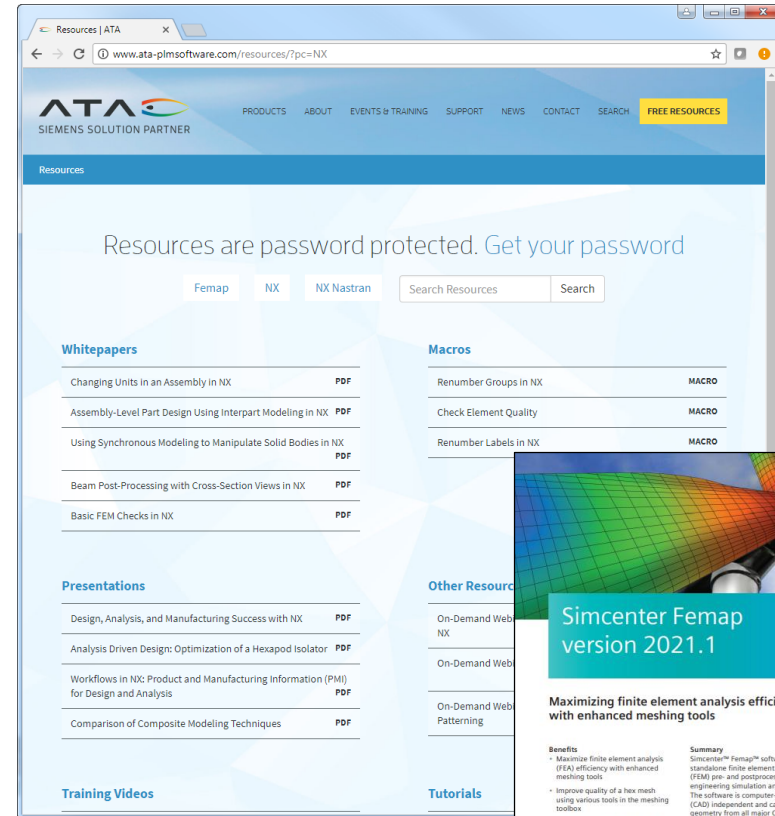
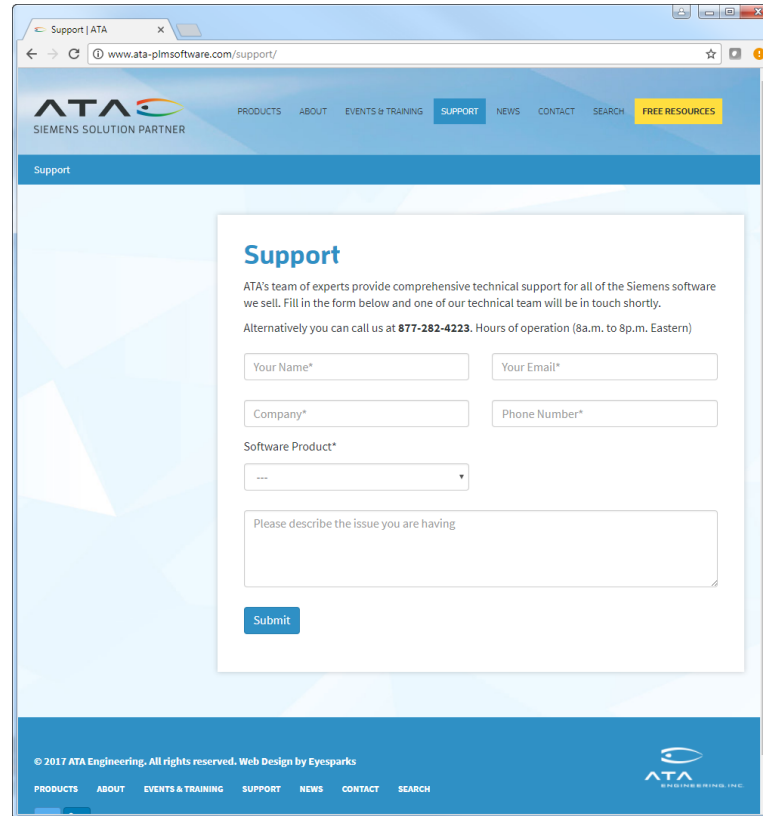
ATA offers training, free resources, and hotline support for a variety of Siemens products.



- 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, STAR-CCM+, and Simcenter 3D

Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com



Maximizing finite element analysis efficiency with enhanced meshing tools

Benefits:

- Maximize finite element analysis (FEA) efficiency with enhanced meshing tools
- Improve quality of a hex mesh using various tools in the meshing toolbox
- Simplify user interface to display only available options for selected solvers via analysis filtering
- Use the familiar interface and depth of functionality of Excite to update data in the functionalable editor in real time or in bulk

Features:

- Mesh control explorer pane to assist in creating complex shell and solid meshes
- Customizable options for mesh propagation in various sizing commands
- Improved performance during entity selection, especially when area or front picking
- Comprehensive support for MSC Nastran SOL 400 to perform implicit nonlinear analysis

Summary

Simcenter™ Femap™ software is a standalone finite element modeling (FEM) pre- and postprocessor for engineering simulation and analysis. The software is computer-aided design (CAD) independent and can import geometry from all major CAD platforms. It supports most CAD data formats. Simcenter Femap also works in combination with a wide variety of FEA solvers, including Simcenter Nastran™ software.

Simcenter Femap, which is part of the Xcelerator™ portfolio, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software, is now being released on a biannual schedule in the spring and the fall, which began with version 2019.1 and continues with version 2021.1. The software is now referred to as Simcenter Femap to reflect that it is a part of the Simcenter portfolio of Siemens computer-aided engineering (CAE) products. For the same reason, NX™ Nastran software is now Simcenter Nastran software. The latest release provides a variety of enhancements that will improve your productivity across the simulation

Workflow: A large amount of functionality added to 2021.1 focuses on facilitating mesh creation and modification. First, matching mesh sizes and determining mesh propagation between disconnected geometric entities can be specified based on the proximity of edge curves to one another. To determine where these mesh propagation options will be enforced, tools were created in the new mesh control explorer pane to provide visual feedback for edge curves, which are fully paired, partially paired or unpaired, along with other tools to visualize which surfaces are paired for solid meshing or have a surface mesh approach assigned. In addition, multiple tools in the meshing toolbox can be used to update an existing hex mesh. Another area of expanded capabilities is support for MSC Nastran SOL 400. From a modeling standpoint, items have been added to specify property extensions, nonlinear materials and contact options designed for use with SOL 400. At the same time, a new analysis type may be selected that offers user interface (UI) elements to define critical and useful solver parameters for SOL 400. Finally, updates have also been implemented for Simcenter Nastran and Abaqus.

Preprocessing enhancements

Improved entity selection performance. Entity selection (picking) now uses OpenMP technology to improve performance, especially for area picking (box, circle, polygon and hexahed) and front picking. In addition, the area within the standard entity selection dialog box

siemens.com/simcenter




Webinar: **Modeling Bolted Joints in Femap**


Rachel Backes, ATA Engineering
March 31, 2021

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

 (858) 480-2000

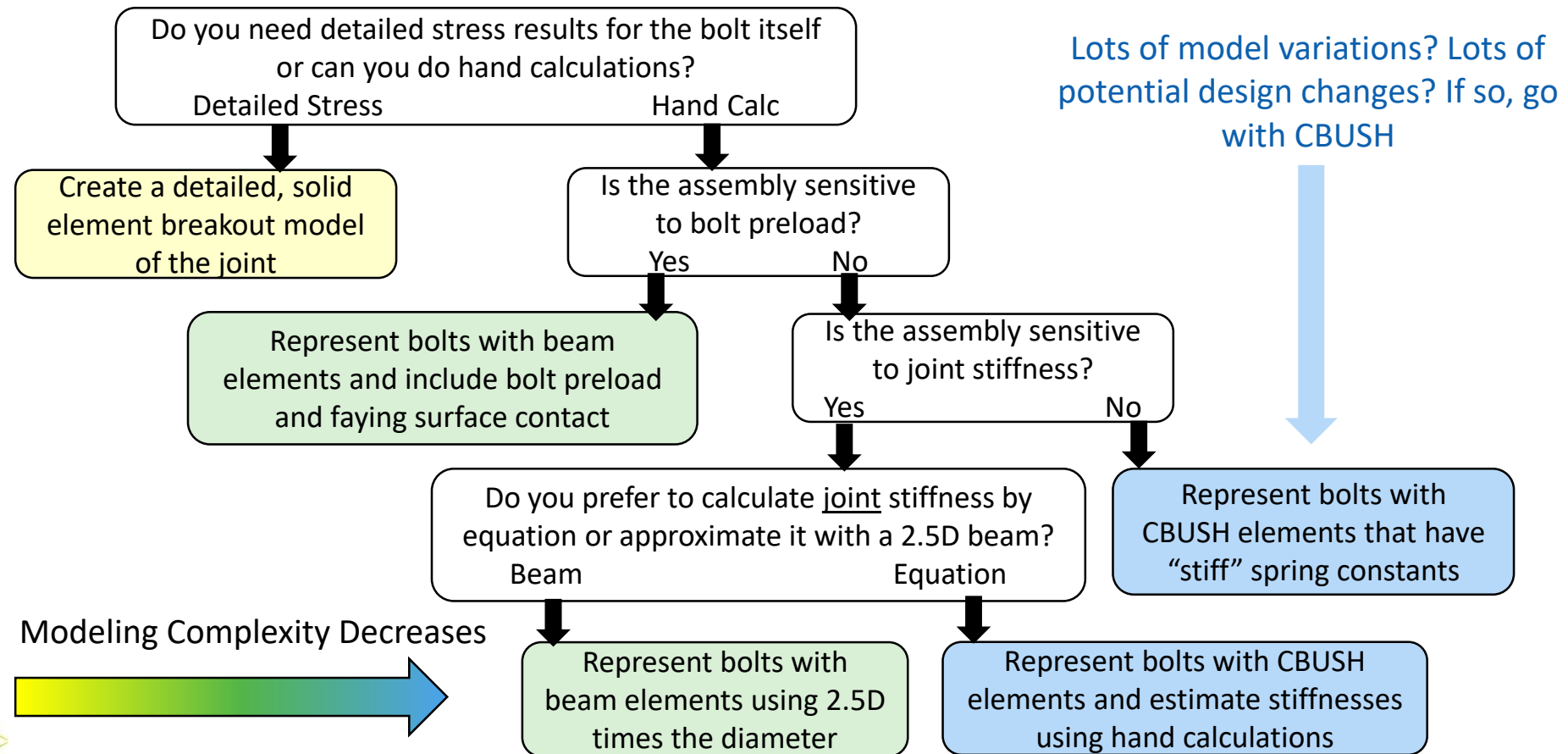
 www.ata-e.com

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

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

Fasteners – Stiffness Element

- Fasteners are almost always simplified in FE models. The decision tree below provides general guidelines for modeling fasteners.



Fasteners - Configuration

Typically, Joint Diameter, D_j is the smallest value of:

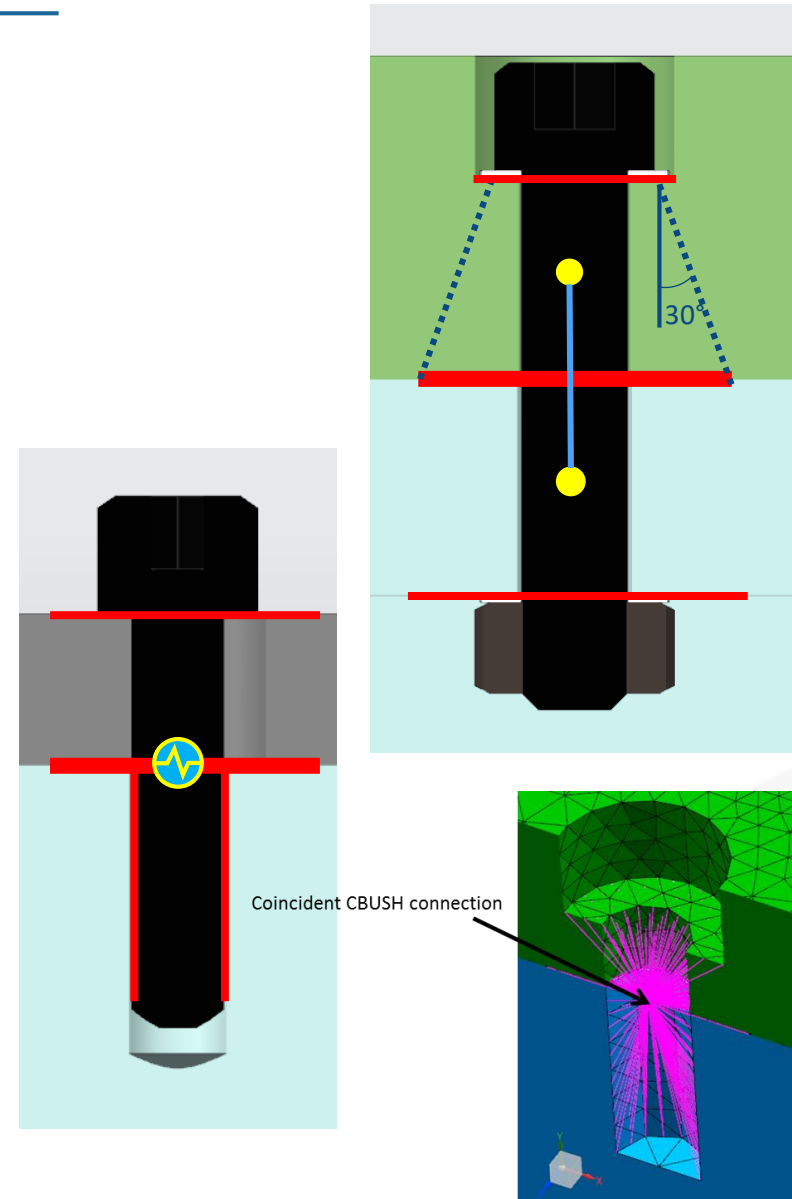
- Twice the radial distance to the closest edge
- Distance between holes
 - Avoid overlapping spiders
- Diameter resulting from 30 deg cone through plate thickness from bolt head diameter

Use D_j or available material to determine size of spiders at head, nut, and faying surface

Spiders only include surfaces along fastener shaft if there are threads (inserts) or direct shear

Locate Beam nodes at center of part

Locate CBUSH nodes at interface (zero length)



Fasteners – “Rigid” Spiders

- You have several options for connecting your stiffness elements to your mesh
- Good rule of thumb is to mix something overly soft with something overly stiff to balance the system
- “Soft” and “Stiff” joints are not necessarily bad.
- **USE ENGINEERING JUDGEMENT!!!**

Warning!

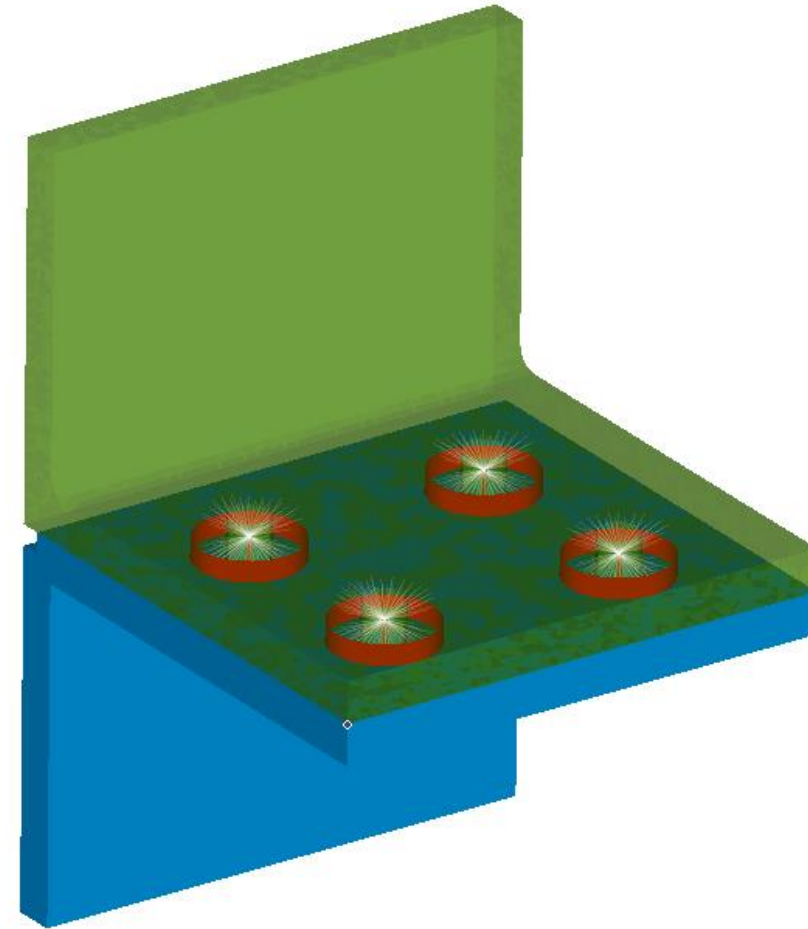
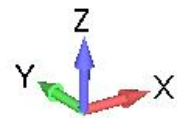
This slide provides general suggestions only.
There are MANY opinions on modeling fasteners. If your company/customer/reviewer has a preferred methodology, then use that as the default for that project and vary as necessary for accurate modeling.

Stiff Beams are the best choice for any **thermal** model
RBEs should be used only when necessary and with extreme caution

	Beams 1.0xD	CBUSH k_bolt	Beams 1.0xD w Preload & Contact	Beams 2.5xD	CBUSH k_joint	CBUSH k_stiff
RBE3	Very Soft	Very Soft	Soft	Kinda Soft	Kinda Soft	Good
RBE2	Soft	Soft	Good	Good	Good	Stiff
Stiff Beams	Soft	Soft	Good	Good	Good	Stiff

Femap Demo

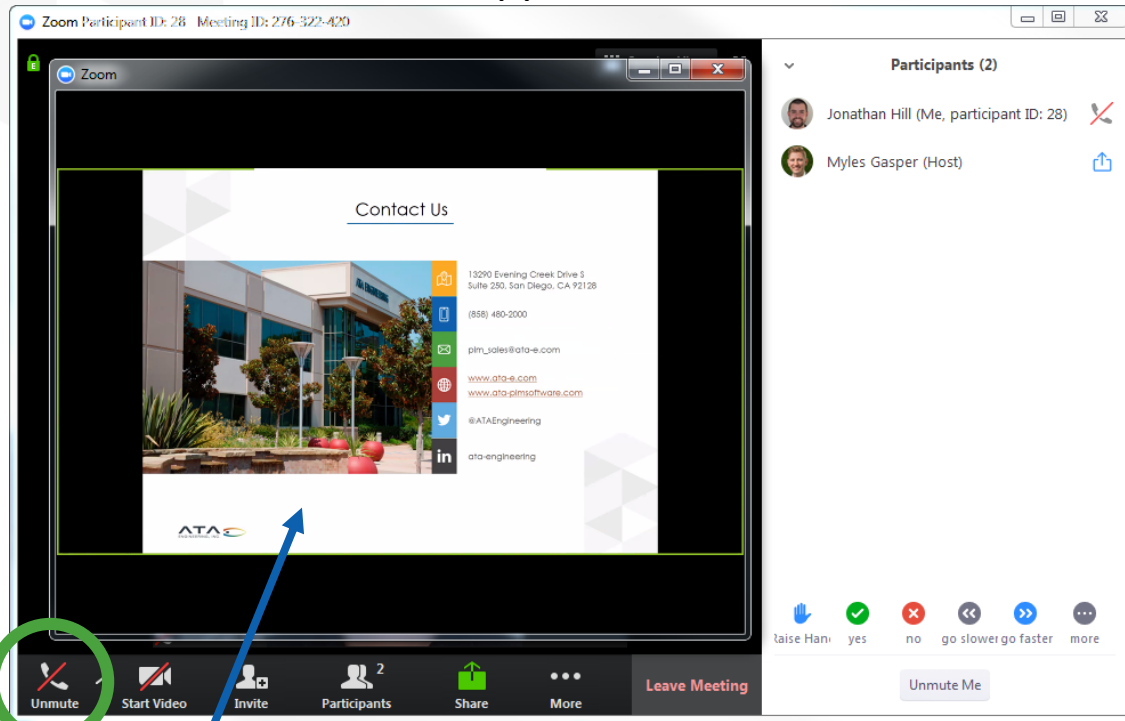
- Create bolt mesh connecting two brackets
 - RBE2 spiders
 - 2.5D Beam stiffness elements



Questions?

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

Zoom Application

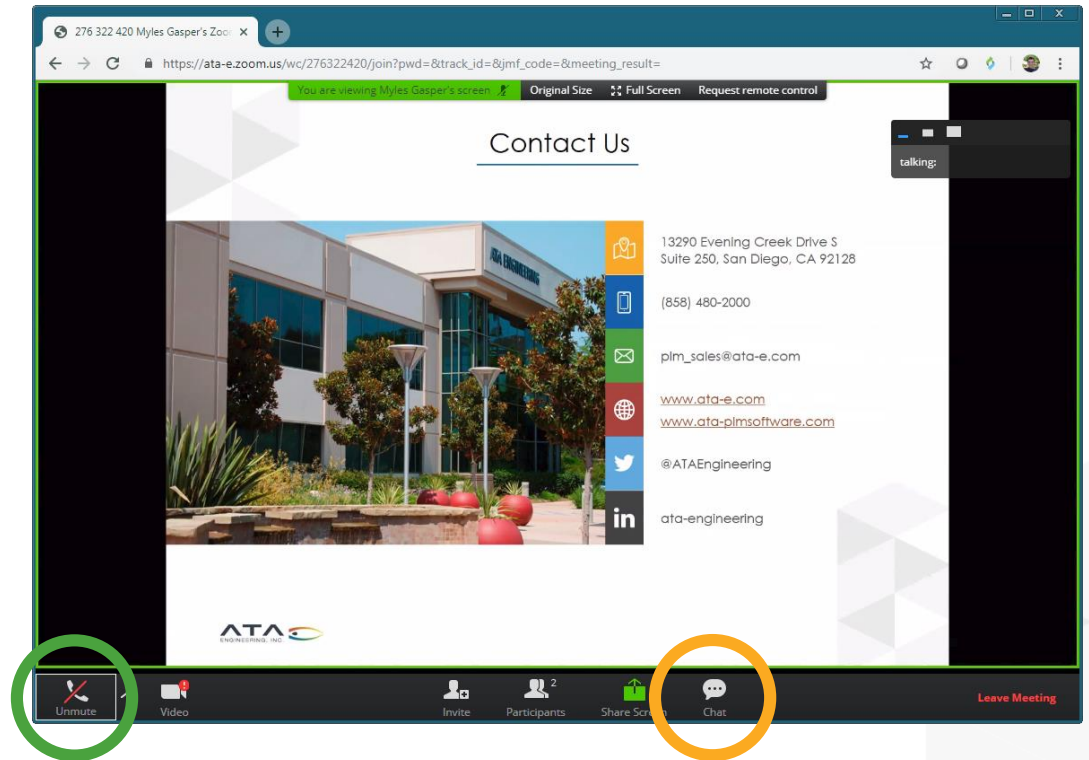


Screenshare in separate window

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