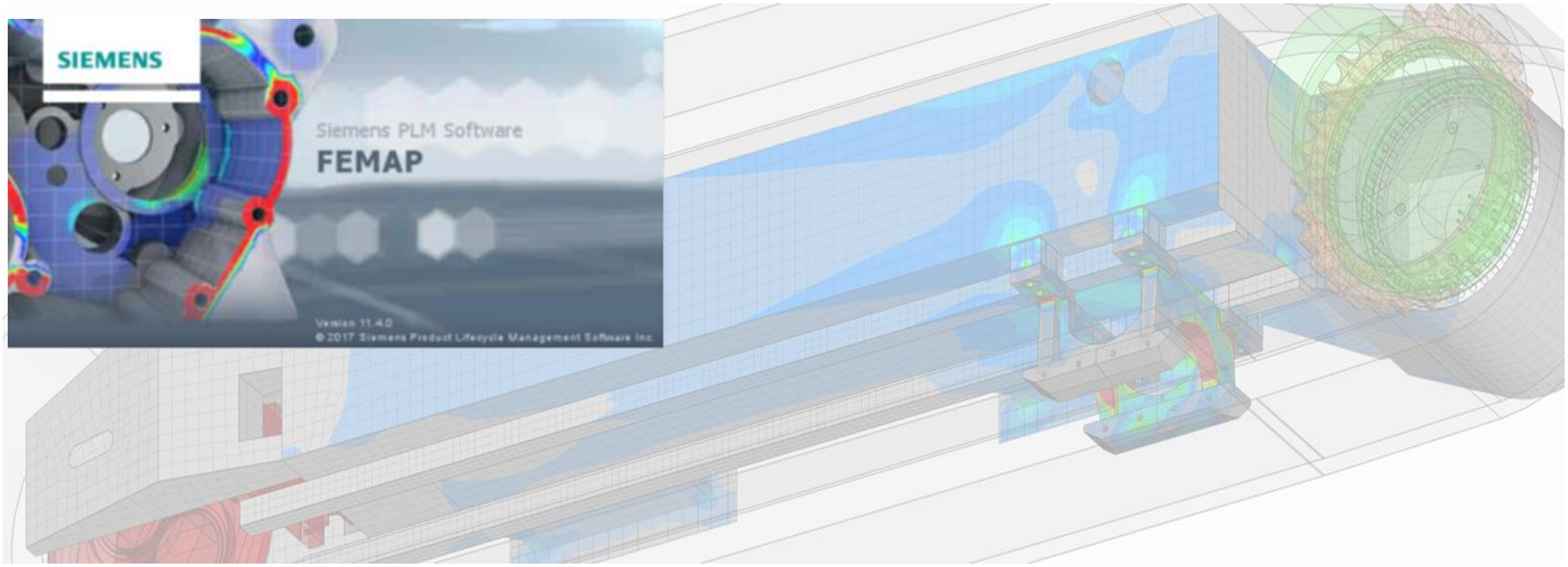


Global-Local Modeling

A Seminar for FEMAP and NX Nastran Users

Adrian Jensen, PE - Senior Staff Mechanical Engineer

Brian Kolb – Staff Mechanical Engineer

Our Services

Software Licenses:

FEMAP • LS-DYNA • Simcenter 3D

Fibersim • NX CAD-CAM • Teamcenter

Experienced technical support

Ongoing training

We welcome your business

Support Review:

”Adrian and Applied CAx consistently get back to me within a few hours with helpful tips or insights. Engineers know that you can't extend a deadline because your FEA support was late in getting back to you. This is why I truly appreciate the level of support Applied CAx provides... They are an excellent resource and I’m glad to have their support.”

Allen Foulstone
Senior Systems Engineer
Stratolaunch Systems

Upcoming Training

FEMAP Training: April 23 – 27, 2018

LS-DYNA Training: May 14 – 18, 2018

FEMAP API Course: Available as an on-demand online course

Interested in becoming a client or in training?

Please contact Andy Whitesides
(503) 962-0287
Andy.Whitesides@AppliedCAx.com

Applied CAx
We do this *every day*

TABLE OF CONTENTS

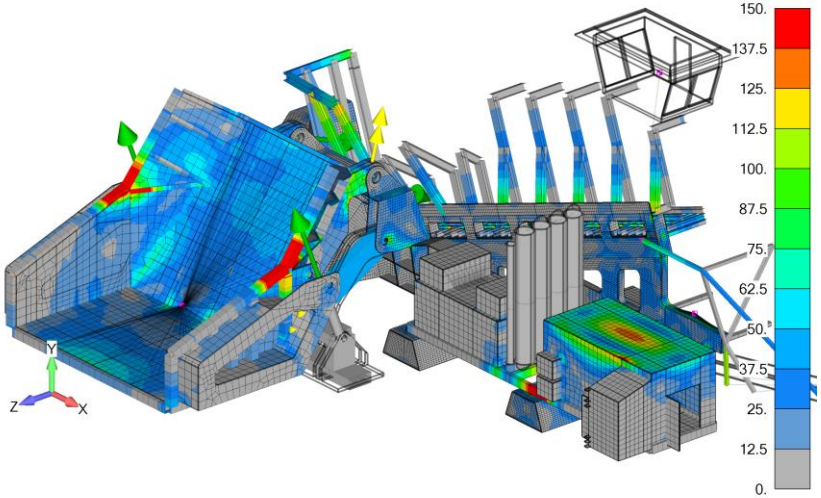
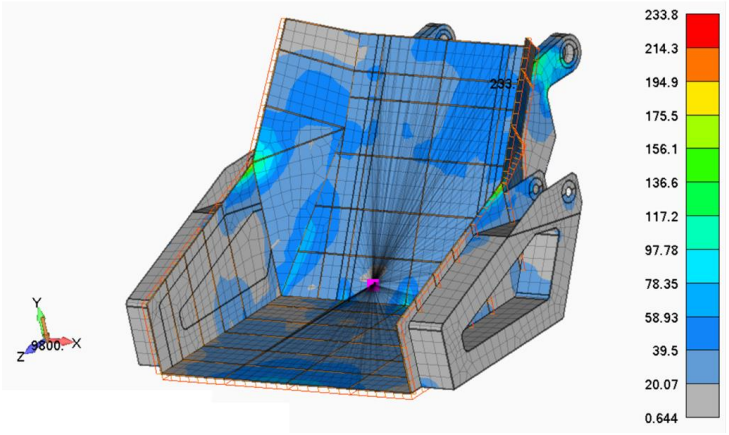
1. AN INTRODUCTION TO GLOBAL-LOCAL MODELING

What is it?

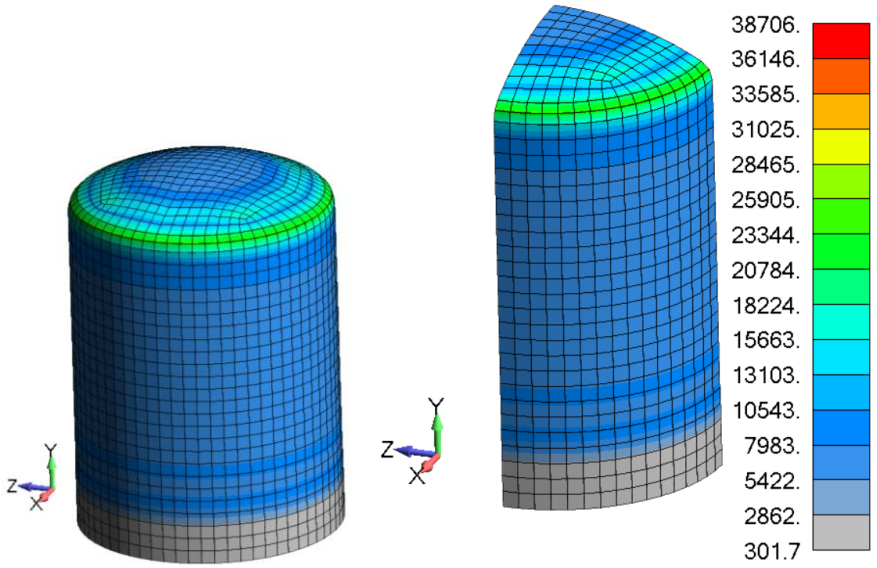
- Global-local modeling takes a very large assembly and reduces it to something that is quick and easy to optimize.

Why do you want to do it?

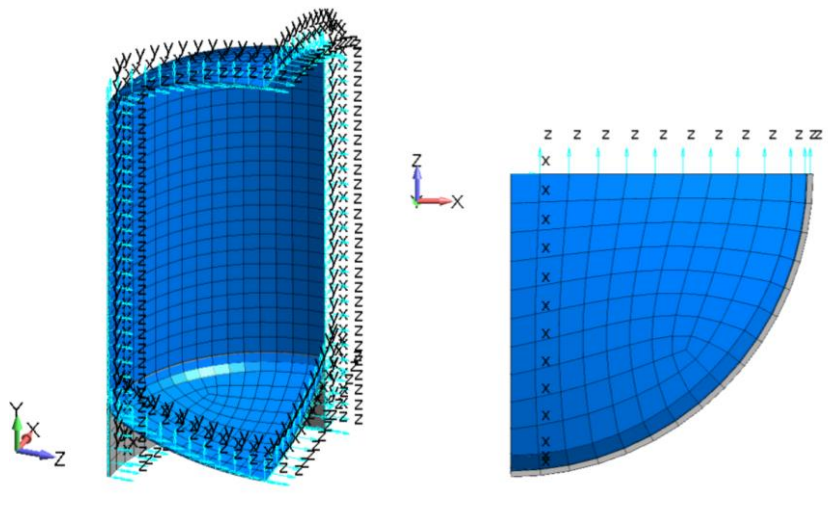
- Despite continuous improvements in software and hardware large assemblies, models with large node counts, and non-linear analysis techniques still require long solve times and large amounts of computational resources.
- It is possible but not practical to pull in a complex CAD assembly, put a fine tet-mesh on it, set up automatic connections and let it spin for a few hours.
- Global-local modeling decreases solve times and reduces computational resources required by allowing the large models and assemblies to be analyzed with a coarser meshes and simpler modeling techniques.

	
Global Model	**Local Model**

1.1 Symmetry Modeling: Simple Pressure Vessel



Full and ¼ Symmetry Models

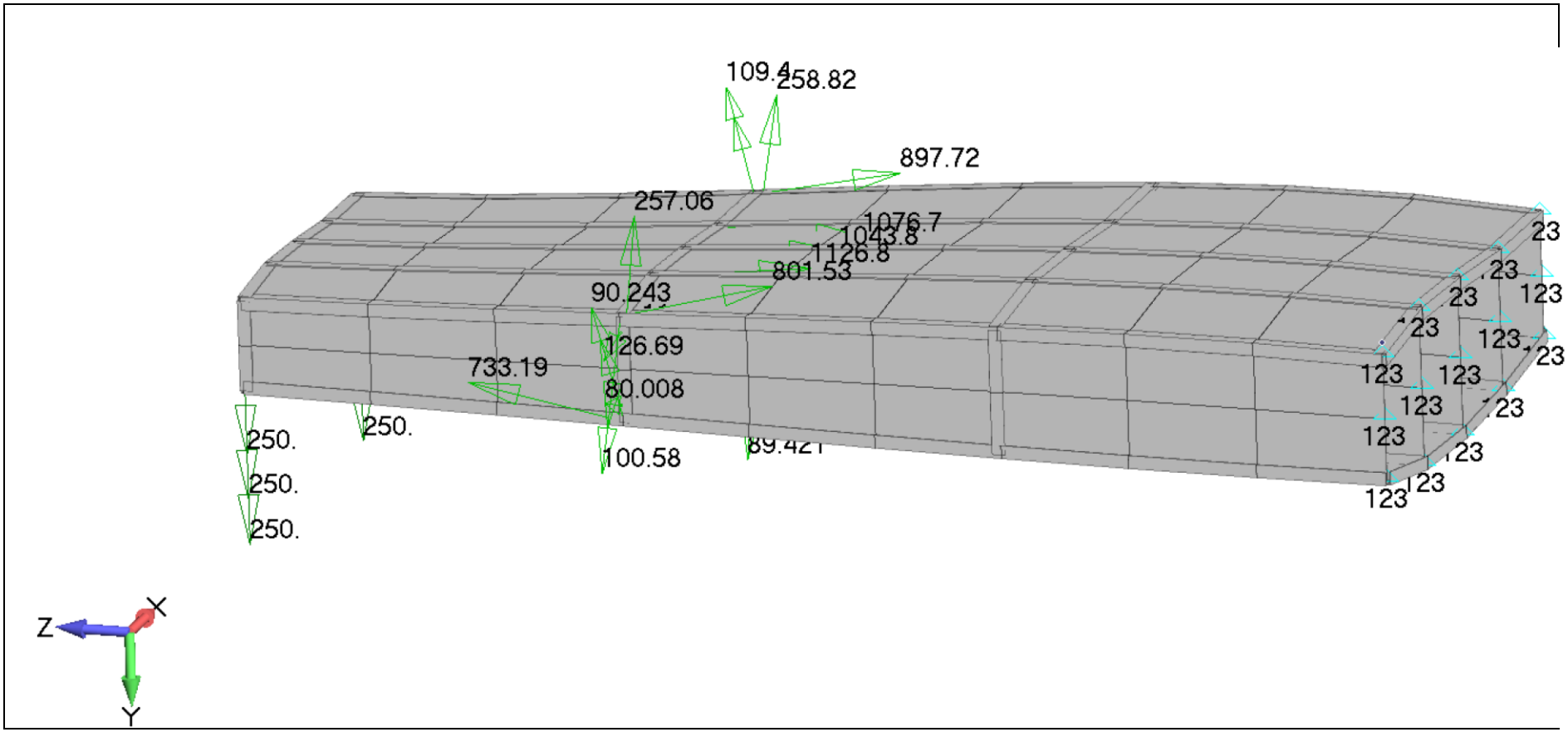


Symmety Boundary Condition

- One of the earliest forms of model simplification.
- If the loads applied to the structure are symmetric relative to the planes of symmetry of the structure's geometry, then the full model can be replaced with a symmetric model.
- Boundary conditions are applied on the symmetric planes constraining out of plane motion but allowing the structure to move freely in the other directions.

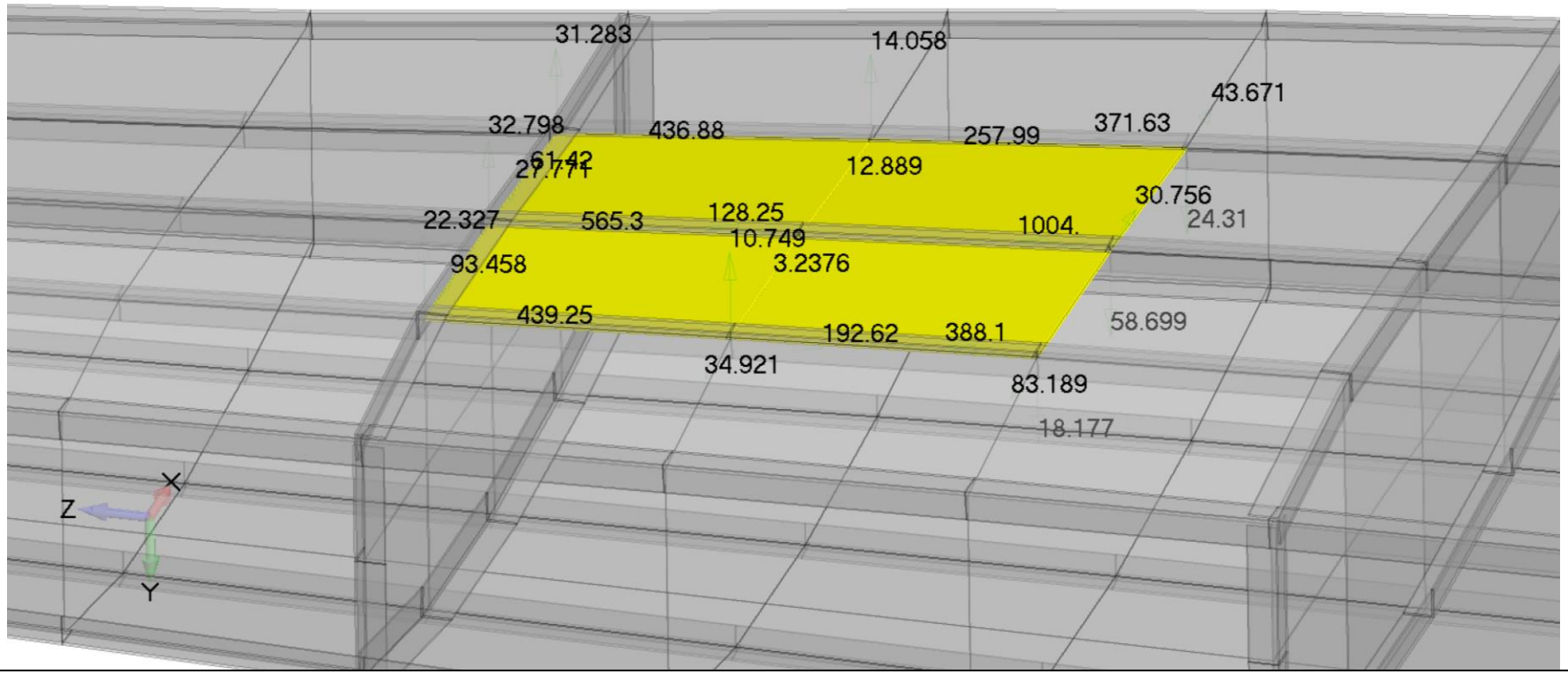
2. FREEBODY DIAGRAMS

2.1 OVERVIEW



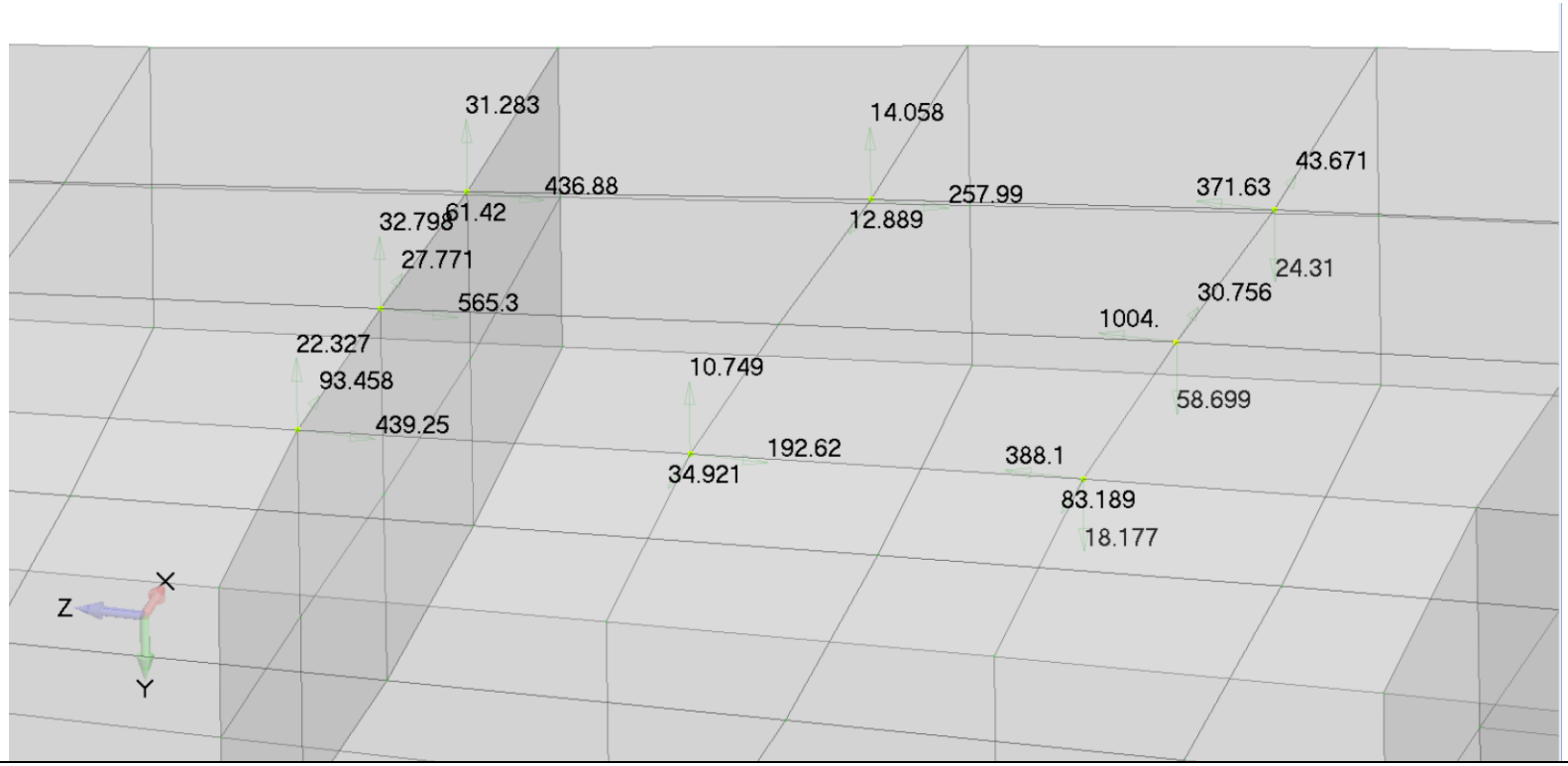
- THE Critical Tool for Global-Local Modeling.
- Powerful tool that is effective in trouble shooting models.
- Used to create the loads applied to the local models.

Freebody Method



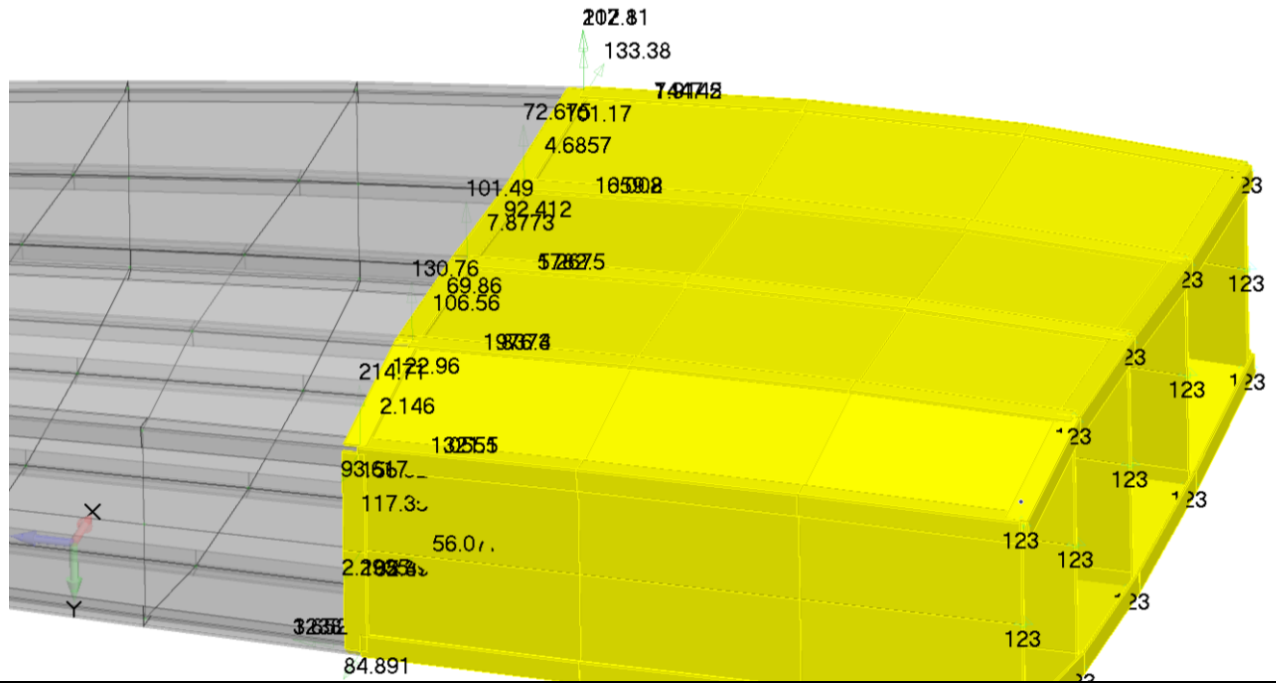
- Requires selecting the elements of interest.
- FEMAP automatically selects related nodes.
- Displays a balanced set of loads on a specific set of elements.

Interface Load Method

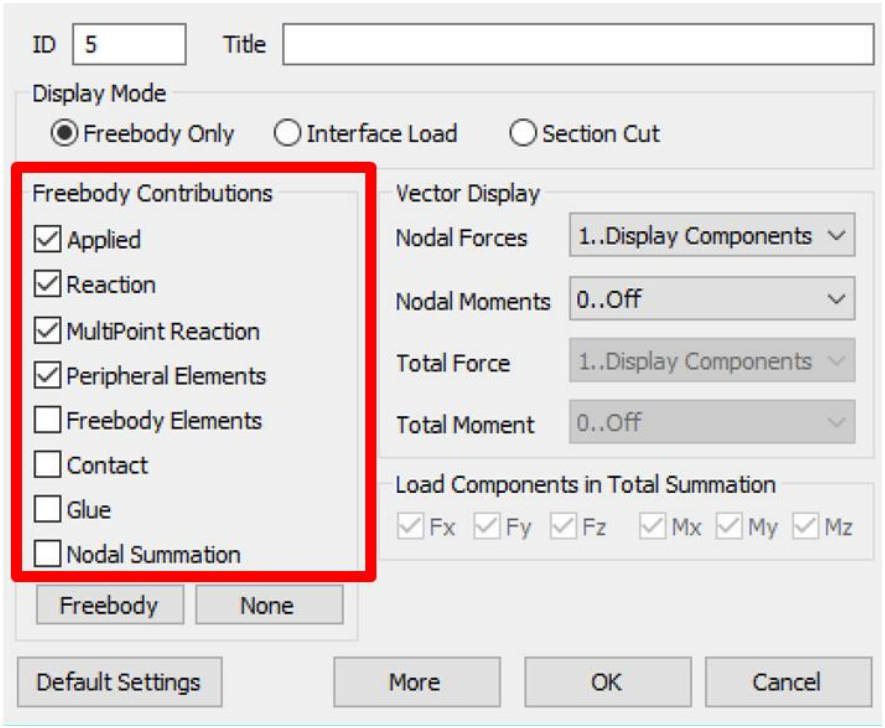


- Requires selecting both the nodes and elements.
- FEMAP calculates a summation of loads and forces across the interface.
- Displays nodal vectors for selected nodes as well as a total summation vector at a selected location.
- Unlike the freebody method, interface load freebodies are not likely to be in equilibrium.

Section Cut Method



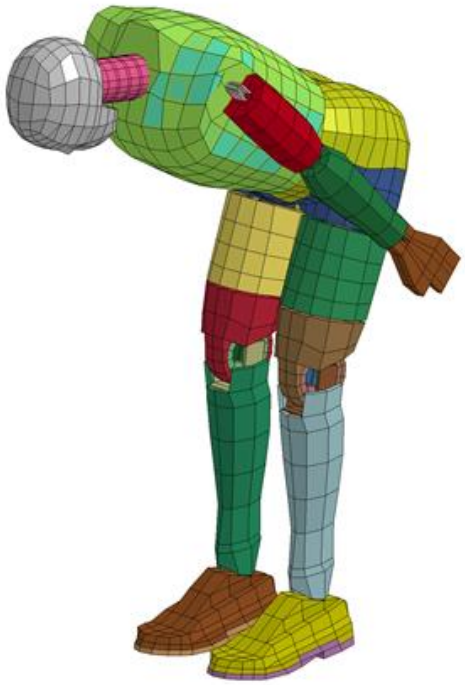
- Requires selecting a “cutting plane”, defined by a plane, vector or a curve.
- The contributing freebody nodes and elements are determined automatically.
- A summed load across the interface is displayed and calculated.
- The total summation location can be placed at the plane path intersection, nodal centroid, or static location
- Nodal and total summation vectors can be aligned tangent to the path without creating additional coordinate systems.
- The cutting plane can be moved dynamically within the model.



- **Applied:** Includes contributions from all loads applied to the model.
- **Reaction:** Includes contributions from all reaction forces and moments at a single point constraint in the model.
- **MultPoint Reaction:** Includes contributions from reaction forces and moments from constraint equations, rigid elements, and interpolation elements in the model.
- **Peripheral Elements:** Includes grid point force and moment contributions from the elements surrounding the selected freebody elements.
- **Freebody Elements:** Includes grid point force and moment contributions from the selected freebody elements.
- **Contact:** Includes contact force contributions from the selected output set.
- **Glue:** Includes glue contact force contributions from the selected output set
- **Nodal Summation:** Includes force and moment contributions from nodal summations. Typically very small unless there is a “non-balanced” force or moment in the model.

3. AND NOW A WORD FROM OUR SPONSORS

Thank You

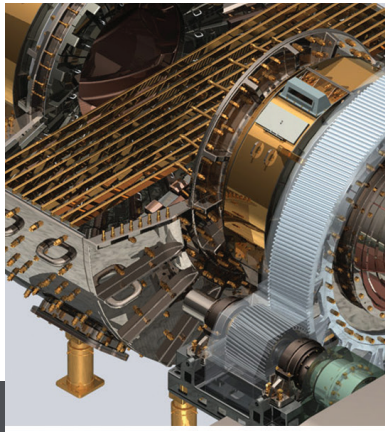


Predictive Engineering is located in Portland, OR

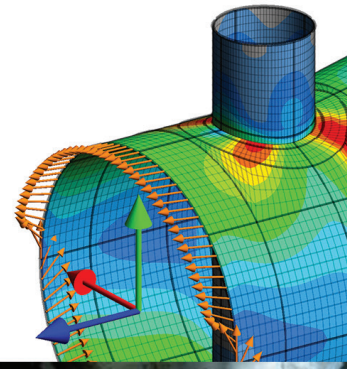
PredictiveEngineering.com

About Applied CAx

Our combination of a mechanical design firm, machine shop and FEA consultancy allows us to provide the strongest support for Siemens PLM Software in the US.



NX CAD • NX CAM • Teamcenter • BCT
Simcenter 3D • FEMAP • Fibersim • STAR CCM+



Software licenses and installation
Premium on-call support from experts
Customized training
Teamcenter architecture
CAM post writing and sim kits

We've helped hundreds of companies stay on track with their important projects. From aerospace to mining to automotive and beyond, we do this *every day.*

Please contact us to learn how we can support you & your team.

AppliedCAx.com
Info@AppliedCAx.com
800-746-8134

AppliedCAx
We do this *every day*