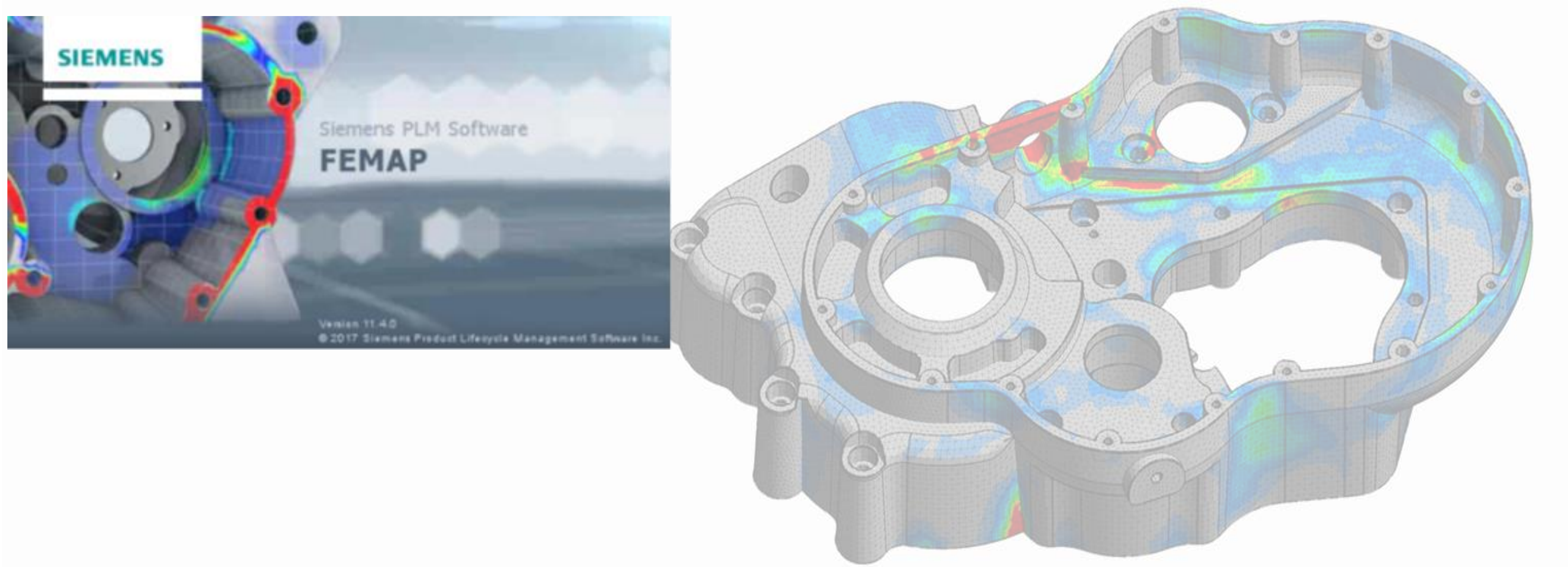


FEMAP Solid Modeling Workflow

A Seminar for FEMAP and NX Nastran Users

Adrian Jensen, PE - Senior Staff Mechanical Engineer
Brian Kolb – Staff Mechanical Engineer

Finite Element Analysis Predictive Engineering
LS-DYNA Sales, Support & Consulting
www.PredictiveEngineering.com

AppliedCAx
Siemens PLM Software Sales & Support
CAD I CAM I CAE I Teamcenter I STAR-CCM+
www.AppliedCAx.com

Our Services

Software Licenses:

FEMAP • LS-DYNA • Simcenter 3D

Fibersim • NX CAD-CAM • Teamcenter

Experienced technical support

Ongoing training

We welcome your business

Upcoming Training

FEMAP Training: October 22 – 26, 2018

LS-DYNA Training: Potentially November, 2018

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

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

Interested in becoming a client or in training?

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

TABLE OF CONTENTS

1. WHY WE ARE HERE

All finite element analysts know this experience – take a single piece of solid CAD geometry, auto-mesh it, apply the boundary and analyze the model. It will be quick, easy and you think you will have some results ready to document in time to leave the office at a reasonable hour… and then the model crashes and the goose chase begins. Sometimes the simplest analysis projects can suck up a ridiculous amount of time.

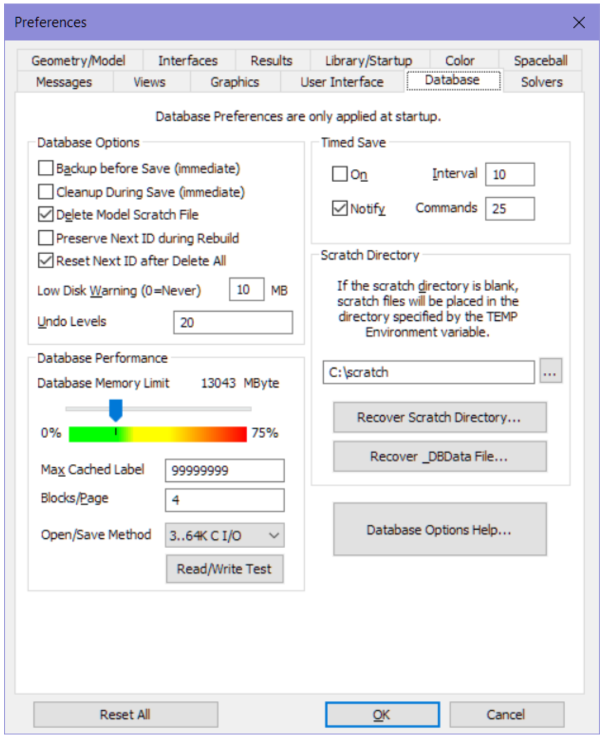
In this technical seminar, we want to get back to the basics and talk about why it's not always so basic. This presentation will walk through the standard analysis workflow from geometry to stress results and take a look at some of the common pitfalls. Along the way, we will share with you some of our time saving techniques. We will be working with solid geometry and tetrahedral meshes as we explore geometry preparation, mesh quality, model verification tools and finally, some recommended post-processing practices.

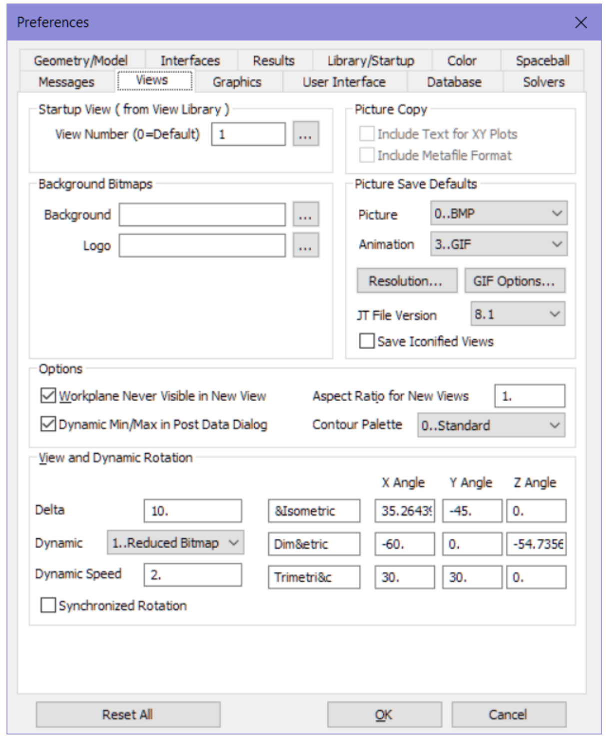
Analysis Workflow:

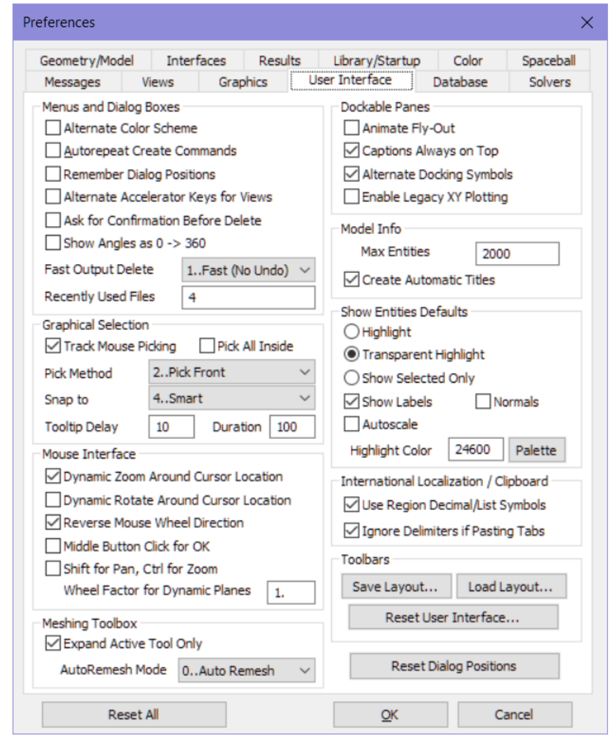
- Geometry
- Material
- Property
- Mesh Sizing
- Meshing
- Loads
- Constraints
- Analyze

1.1 LET'S START WITH FEMAP PREFERENCES

Undo at 20	Resolution at 4x	Pick Front







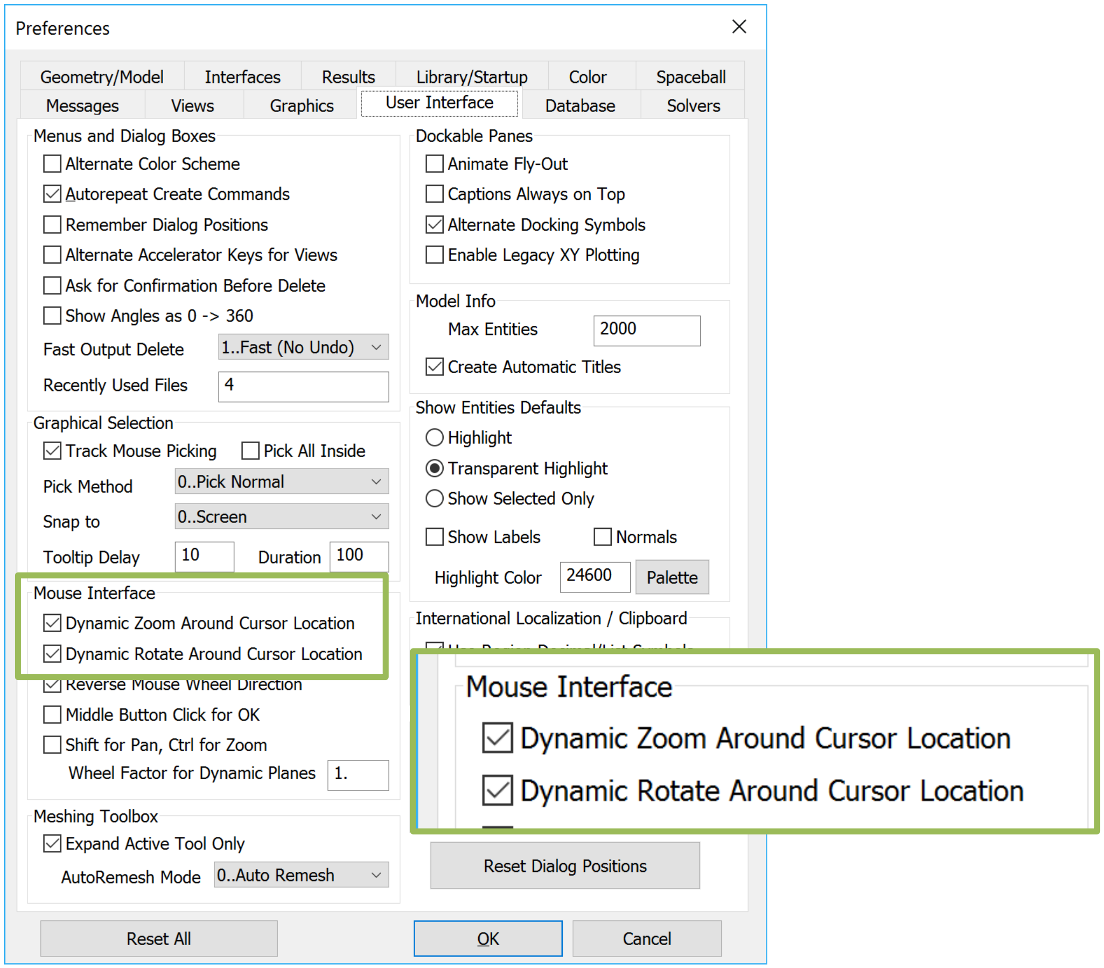
Scratch to C:\scratch

View Number to predefined view

Turn off Ask for Confirmation

Turn off Autorepeat (just use Ctrl+Y)

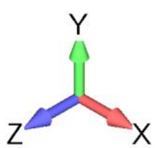
A nice feature added for Femap 11.3 is the ability to configure a dynamic zoom and rotation around your cursor location instead of the center of your screen or view center.

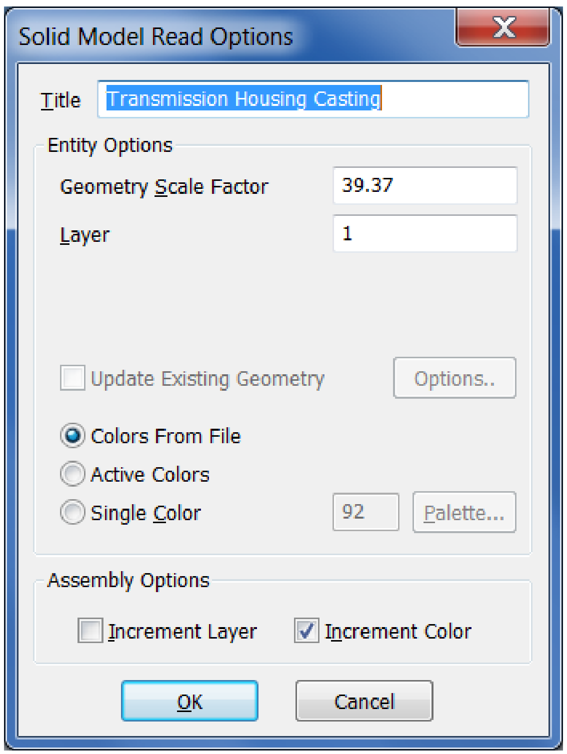


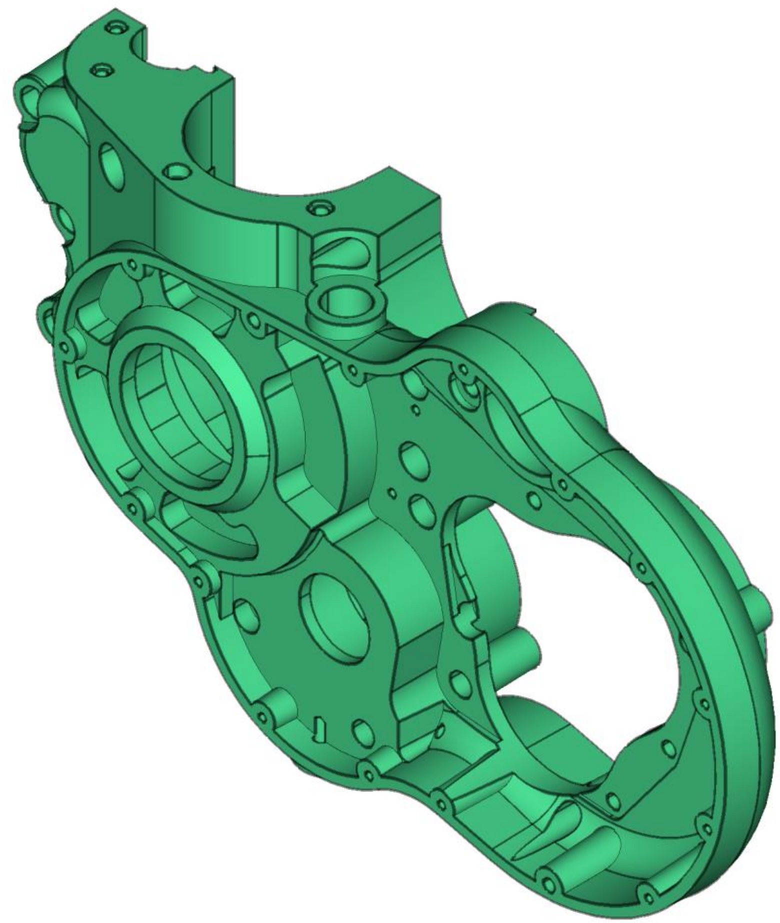
2. GEOMETRY

File > Import > Geometry

Note the scale factor (39.37 in/meter). The scale factor can be controlled after import via ***Modify > Scale > Solid***. Consider model scale factor and part scale factor.

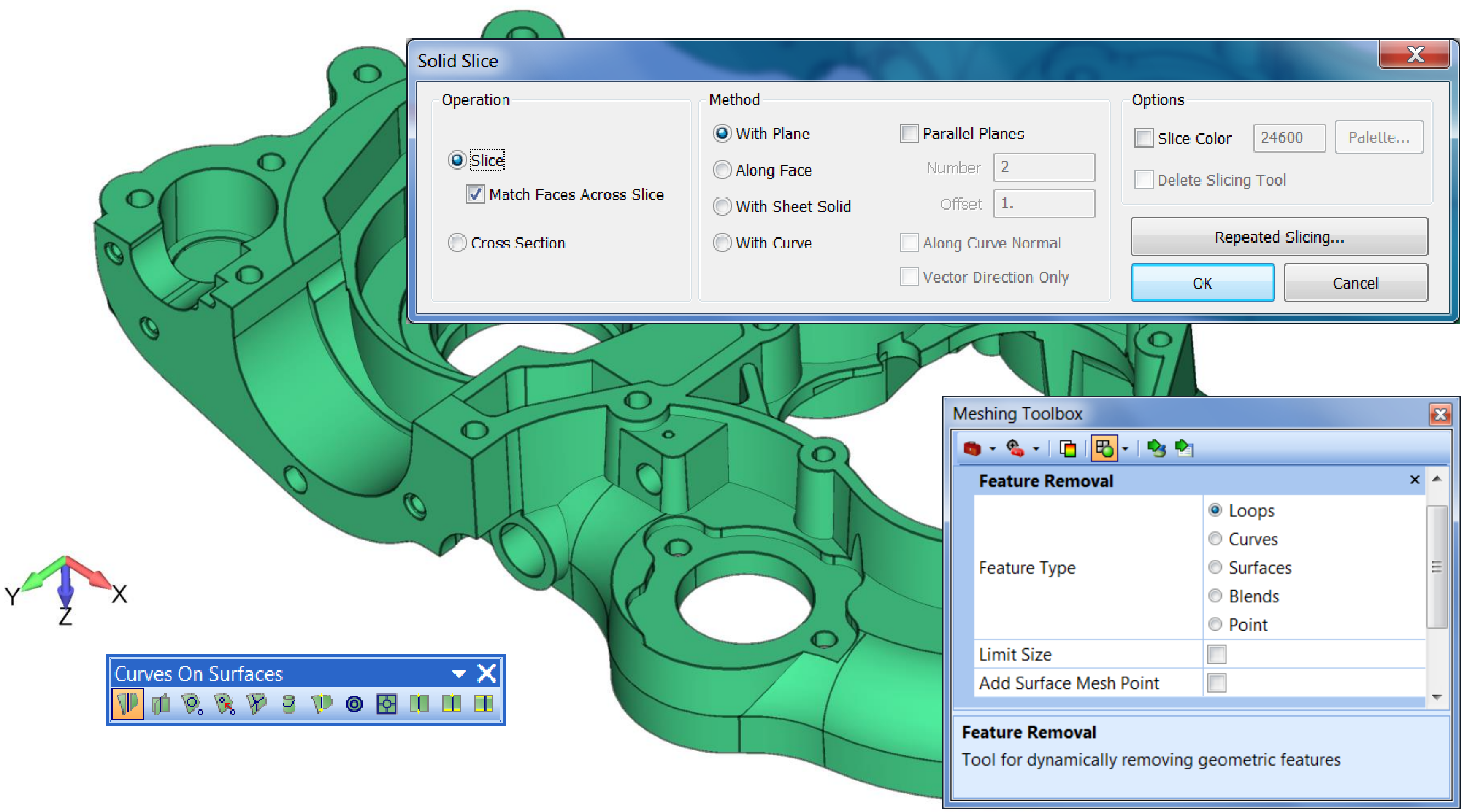






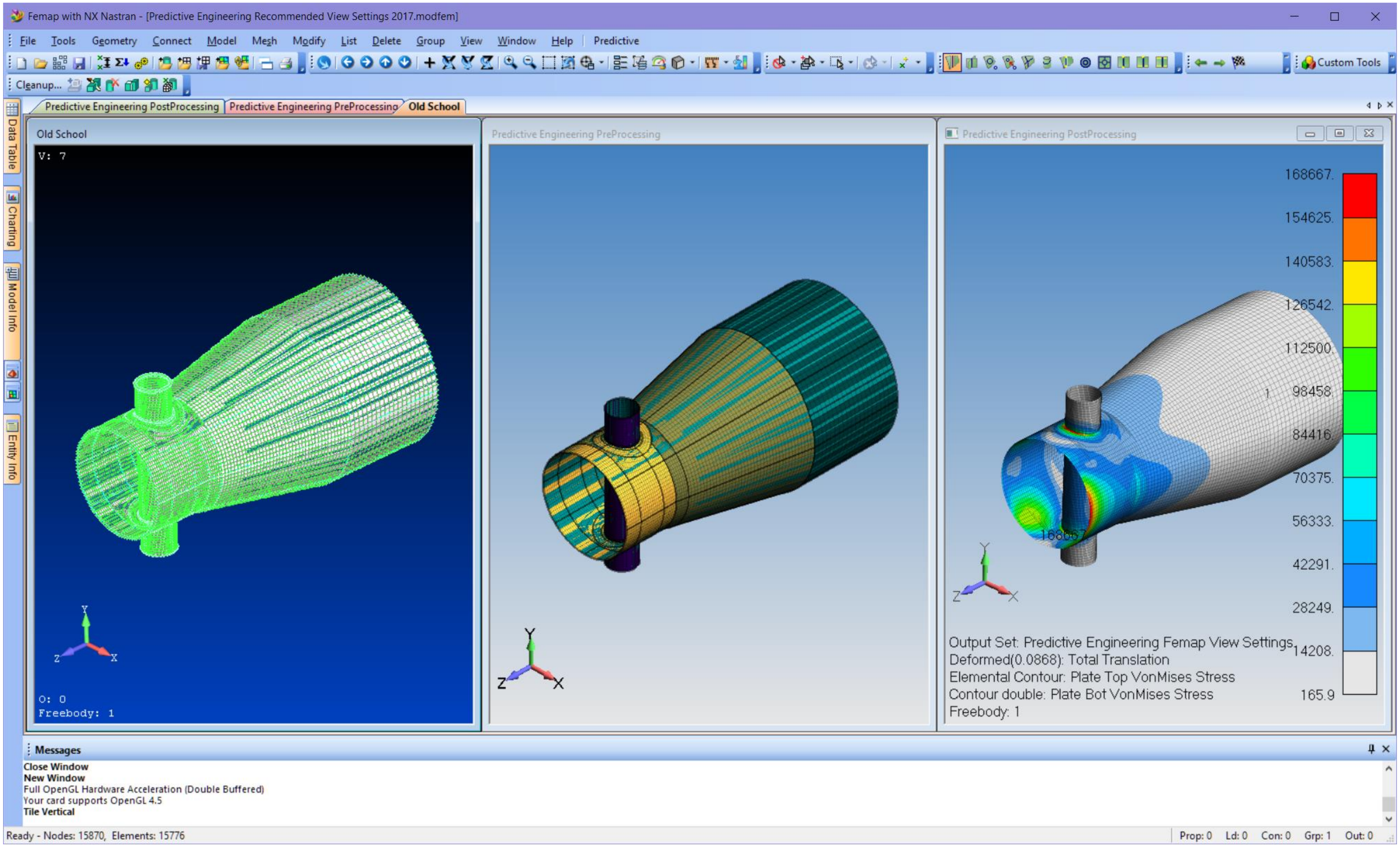
2.1 Geometry Prep – De-Featuring, Slicing, Inscribing and Embedding

It's best to make modifications to the geometry before moving on to other step in the analysis workflow. Let's take a look at a few ways to prepare the geometry for mesh control and application of boundary conditions.



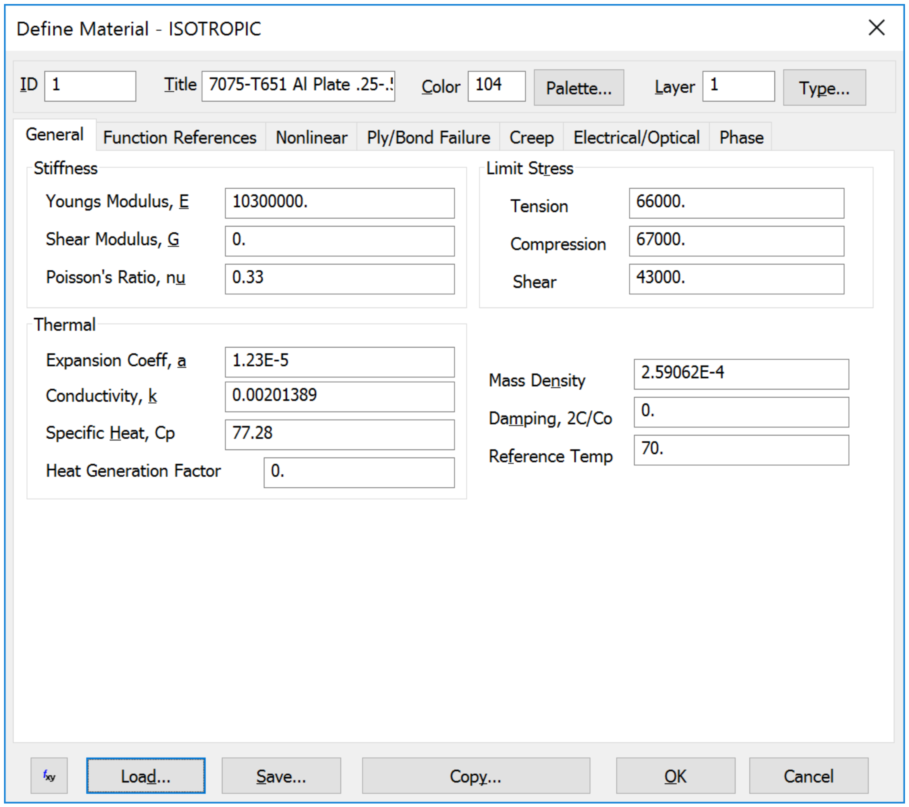
2.2 Panes, Toolbars and Views: The joys of creating a custom work environment

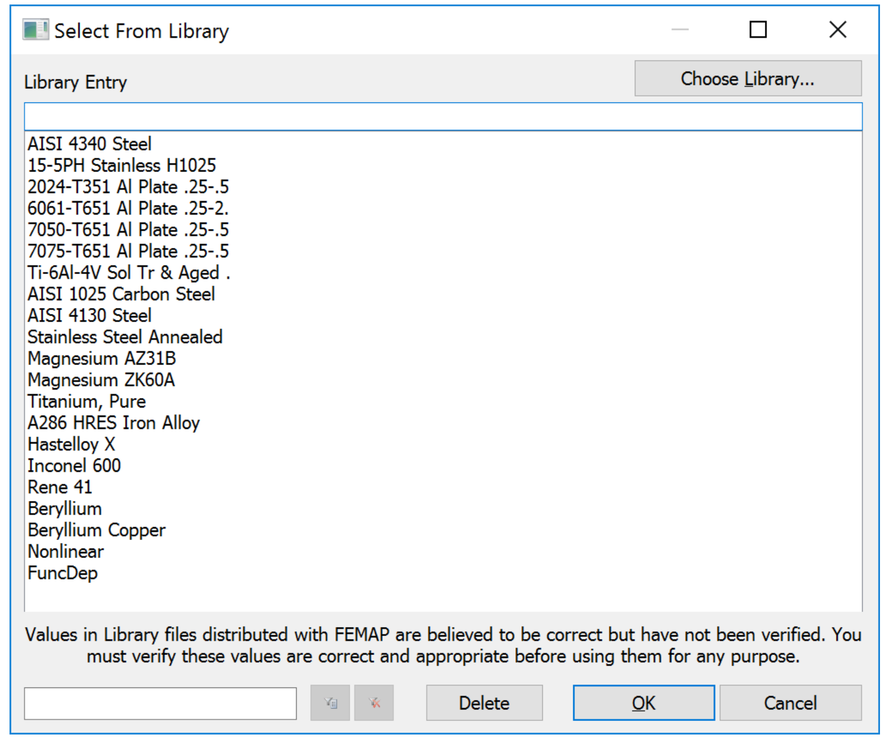
Before we get too far, let's talk about view settings… Nobody wants to see magenta in the contour plots.



3. MATERIALS AND PROPERTIES

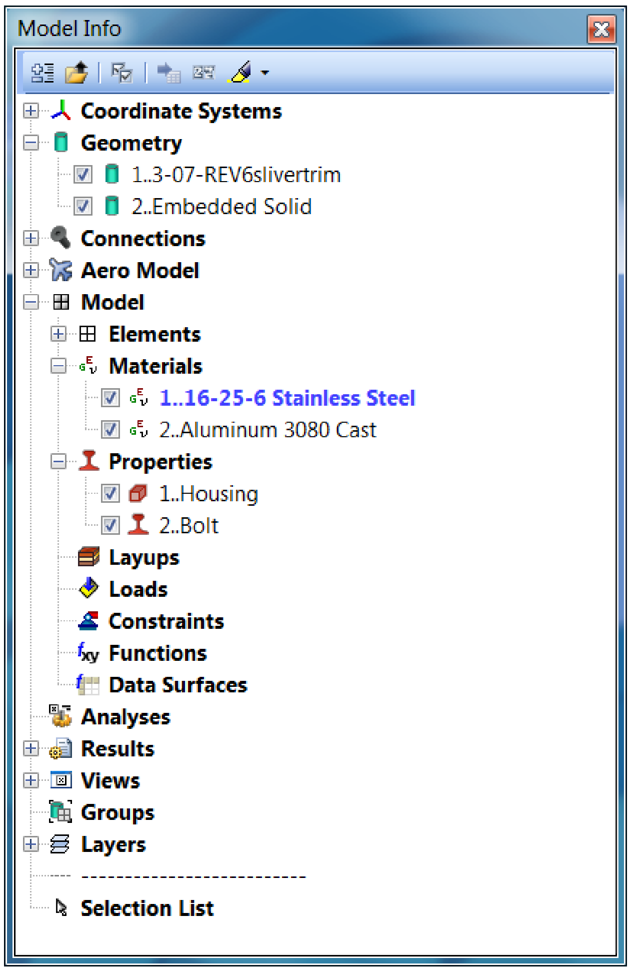
Ok, back to it! Create your own materials or pull from the FEMAP libraries. Or, create your own and save it to the material library. Watch out for auto-repeat!

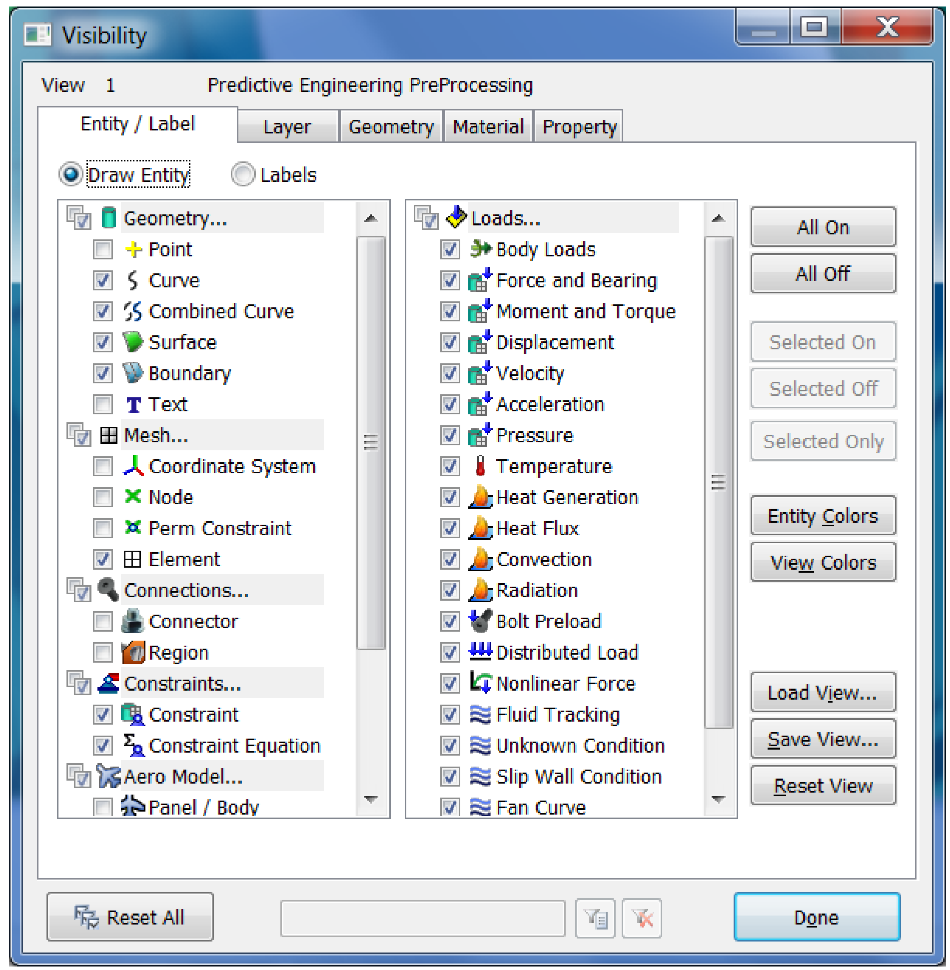




3.1 Blanking from the Model Info Tree and the Visibility Toolbox

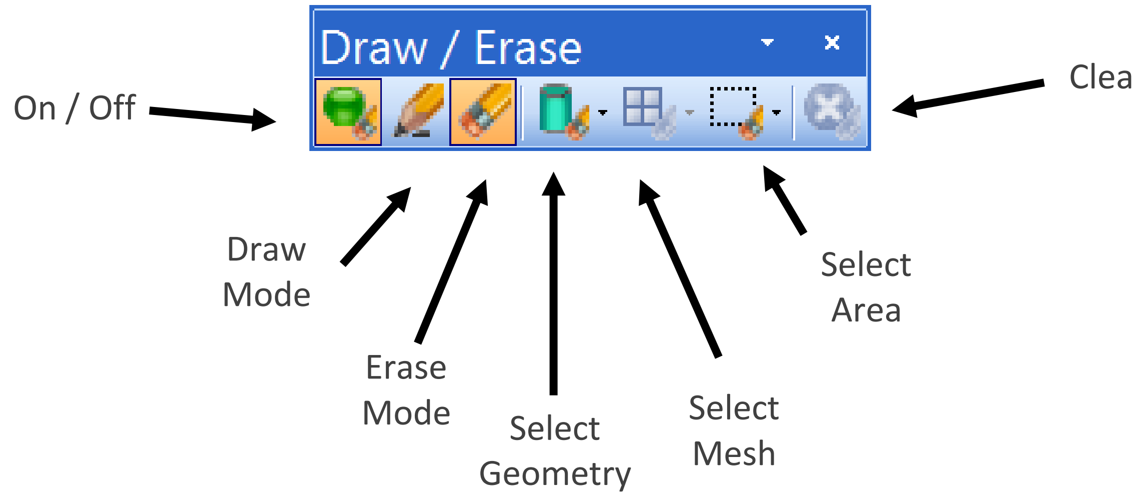
There's lots of ways to control what is displayed in FEMAP. We recommend starting with the Model Info Tree and the Visibility Toolbox. Get used to using Ctrl-Q to access Visibility and your FEMAP workflow will be much quicker.



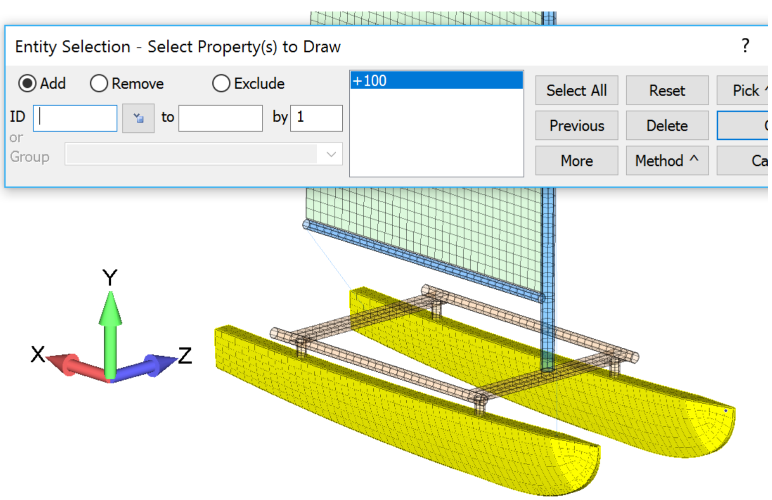


3.2 Draw / Erase Toolbar

The Draw / Erase toolbar is a useful new tool for navigating your model by selecting Entities to show, or to hide. It's even better once your geometry is already meshed.

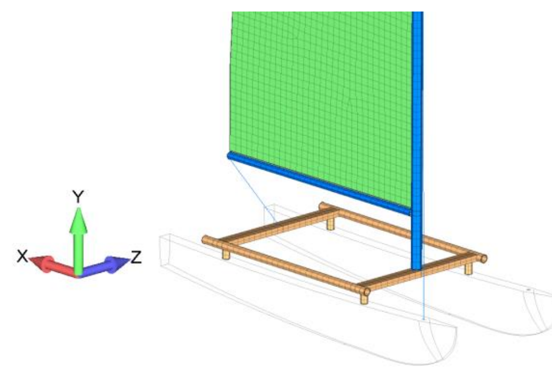


Select by Property



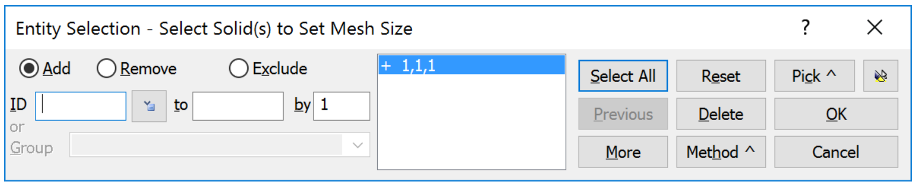
Draw Only

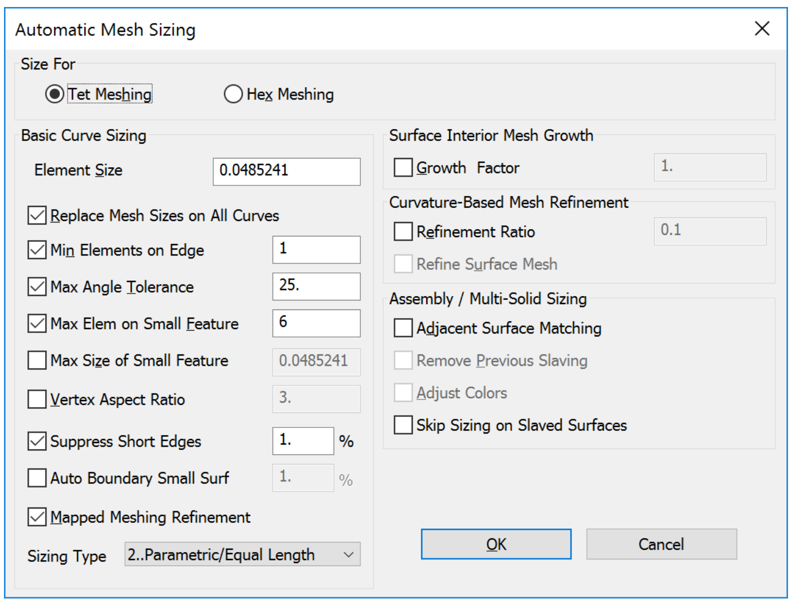
Erase Only

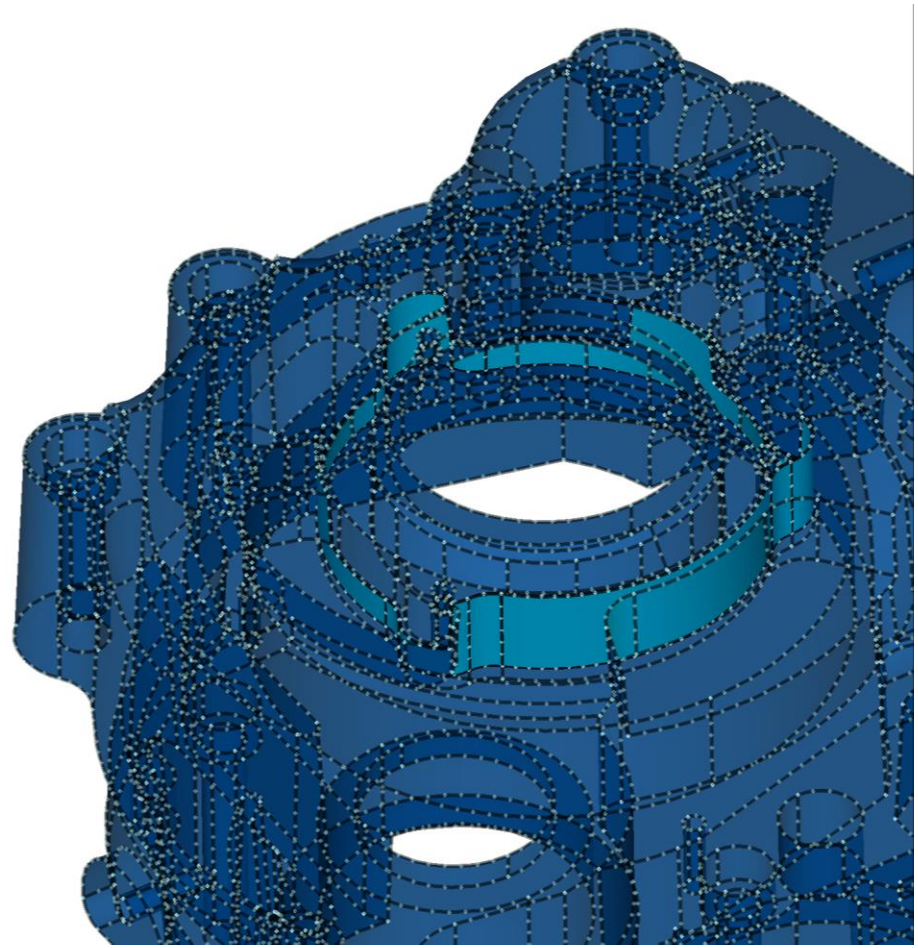


4. MESH SIZING

Choose a mesh sizing approach that is appropriate for your geometry and desired element type. In this case, we will be using ***Mesh > Mesh Control > Size on Solid***. Note the option for “Multi-Solid Sizing” and how the appearance of your curves and surfaces changes once you have applied mesh sizing. Also pay attention to the message window.







4.1 Why Do We Care About Tet or Hex?

We'll skip the lecture about good mesh / bad mesh and just get to the point about why hexing is has some advantages:

Simply Supported Bending = 12,000

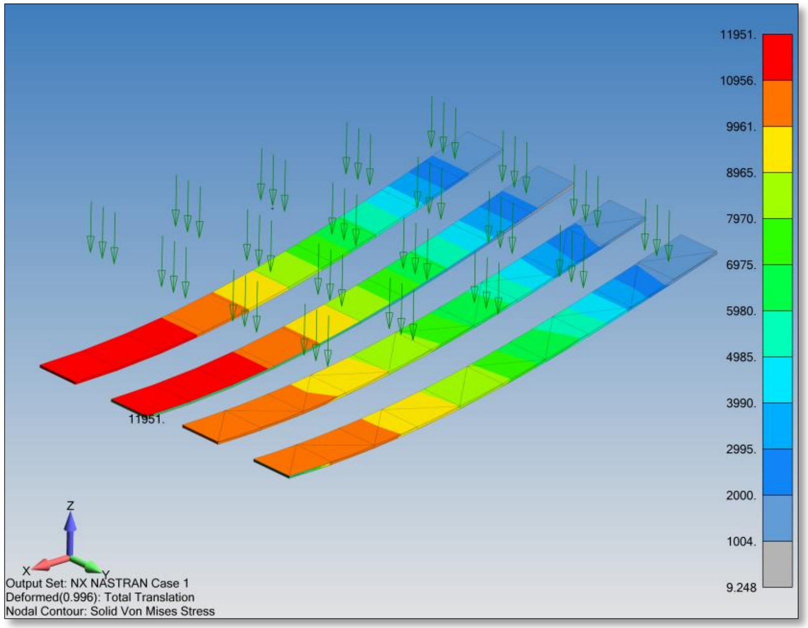
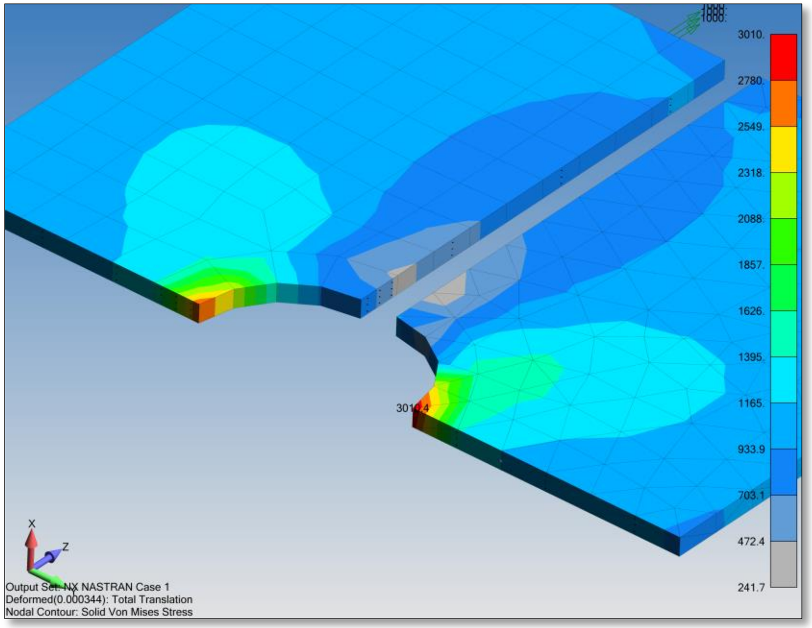


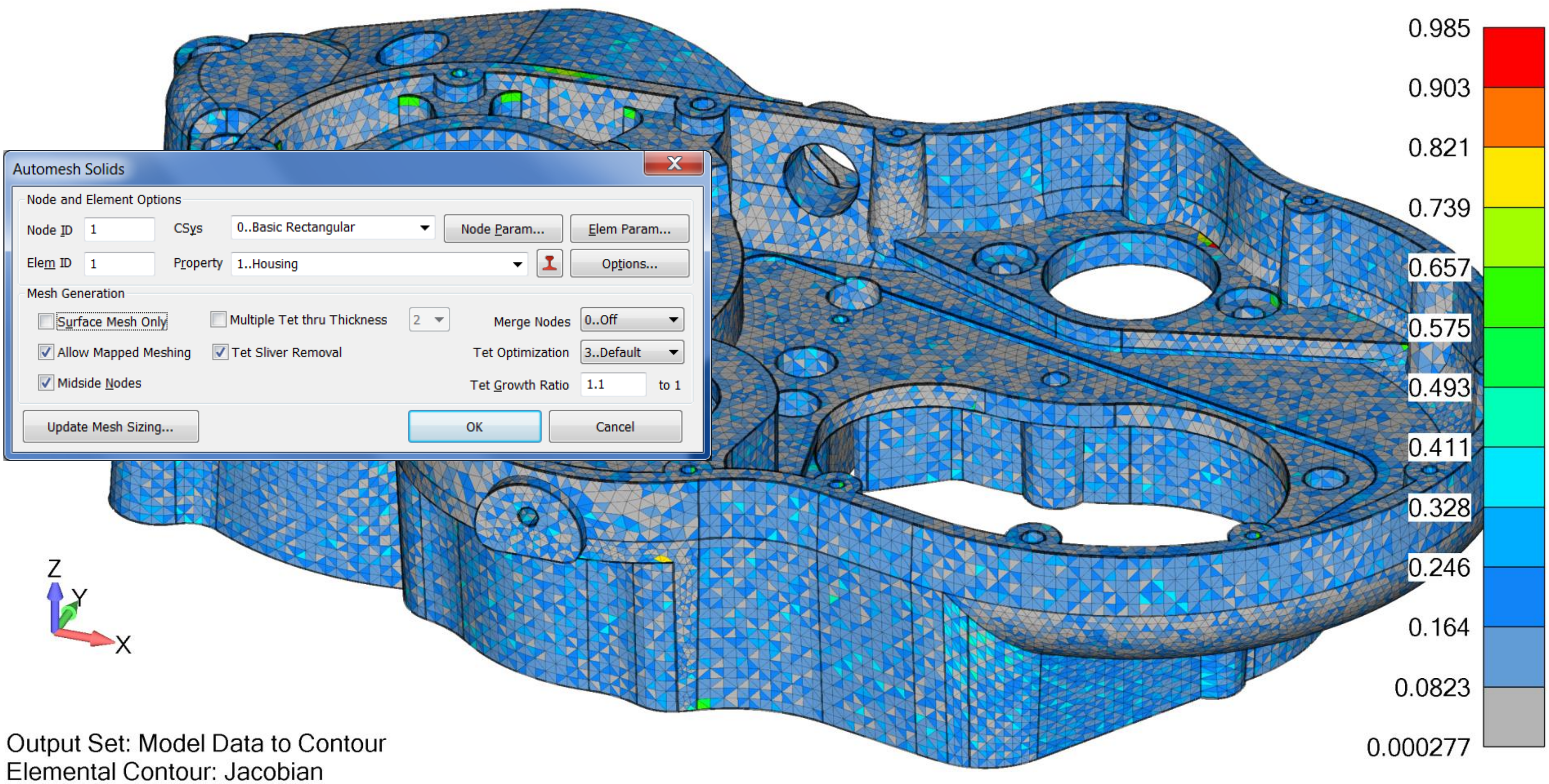
Plate with Hole under Uniform Tension K_t = 3.0



Pros	Cons
• Tet is "easy" and for linear analysis hard to justify hex	• Tet is heavy (to fill a cube: 26 vs. 8 nodes)
• Tet provides acceptable linear elastic stress results	• Often unsuitable for material nonlinear analysis
• Good Tets = Good Stress (watch Jacobian and Tet Collapse – no free lunch)	• Can be unwieldy in large models that require normal modes solution (e.g., PSD)

5. MESHING

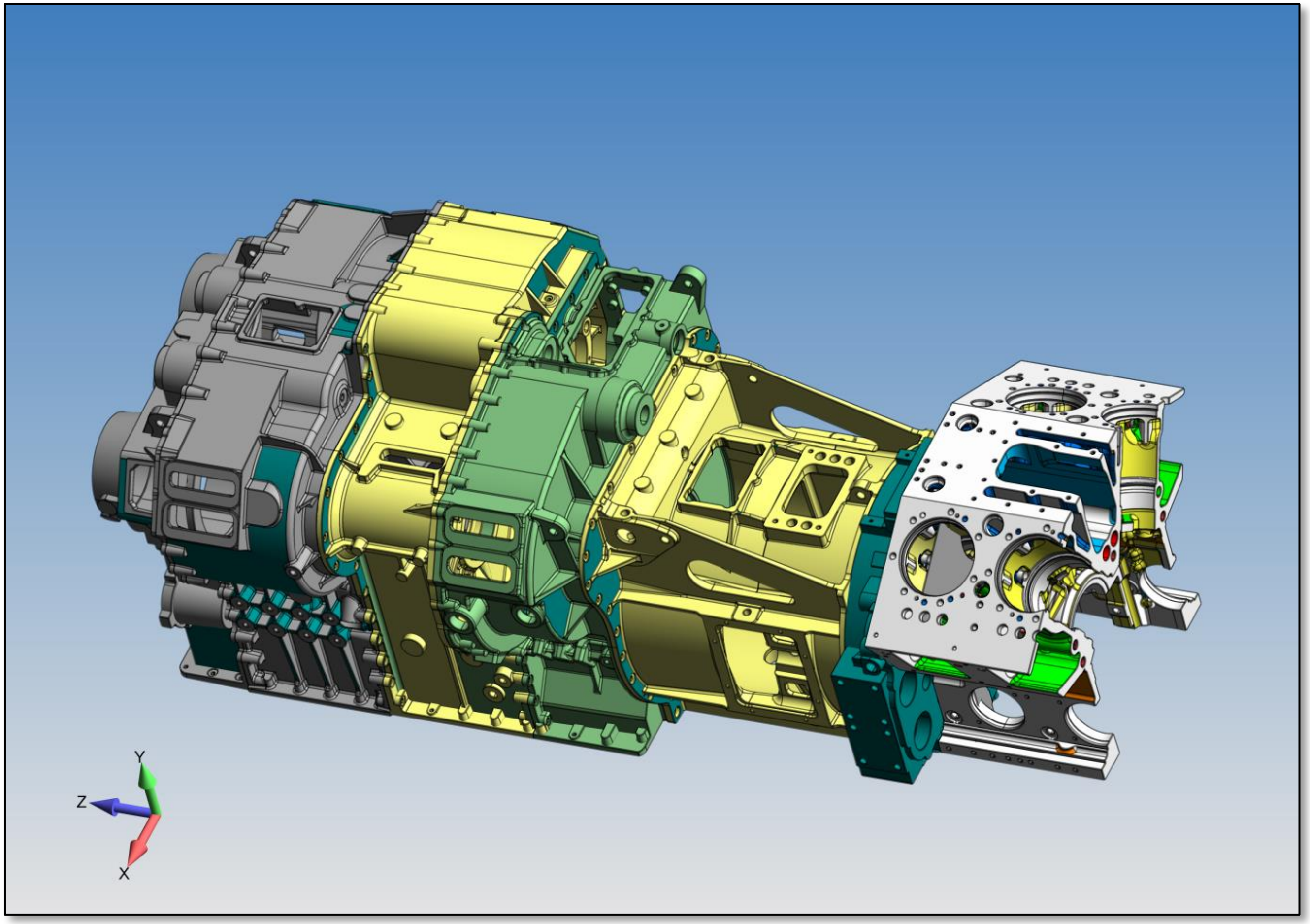
Choose a meshing method that is appropriate for your geometry and desired element type. For this model, we will be using ***Mesh > Geometry > Solids***. Just like mesh sizing, the defaults are the recommended starting point but take some time to look at the options and the corresponding help section. It's really quite useful and it will save time in the long run. As always, pay attention to the message window; there's good information there.



5.1 Tetrahedral Meshing Fundamentals - How it Works

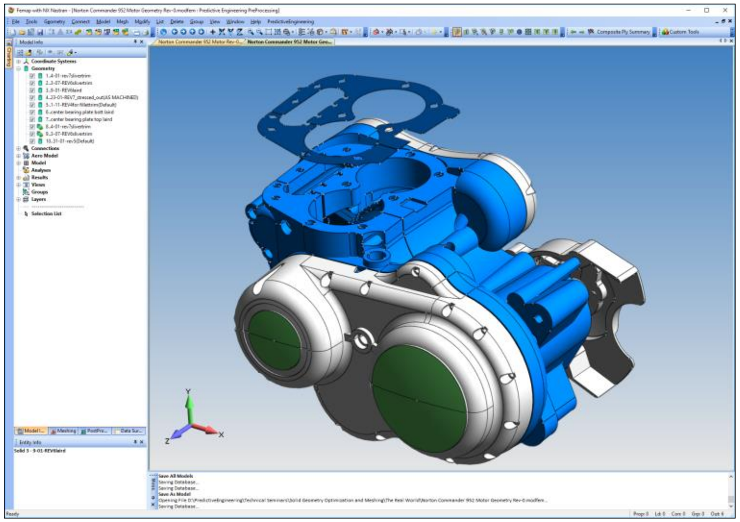
Everybody does it the same: (i) surfaces are meshed with three-node triangular elements (FEMAP plot-only), (ii) the surface mesh is "sealed" (nodes at edges are merged and elements are checked) and (iii) the surface mesh is submitted to the solid tet mesher.

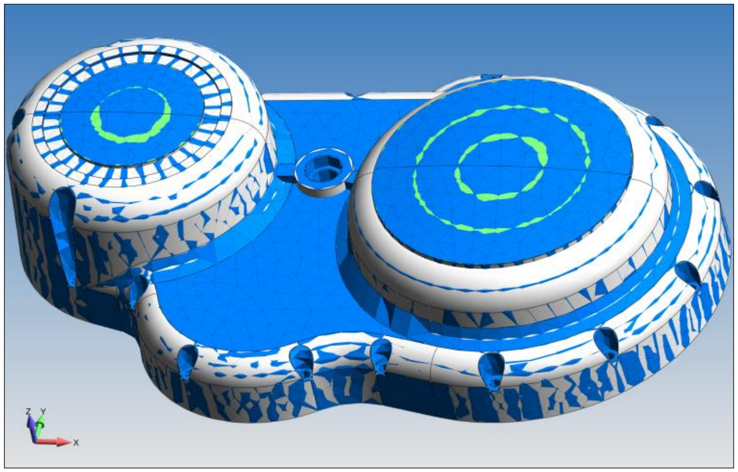
In this example, we have done things simply not too much cleanup or geometry prep. But what happens when you have to deal with a beast like this?

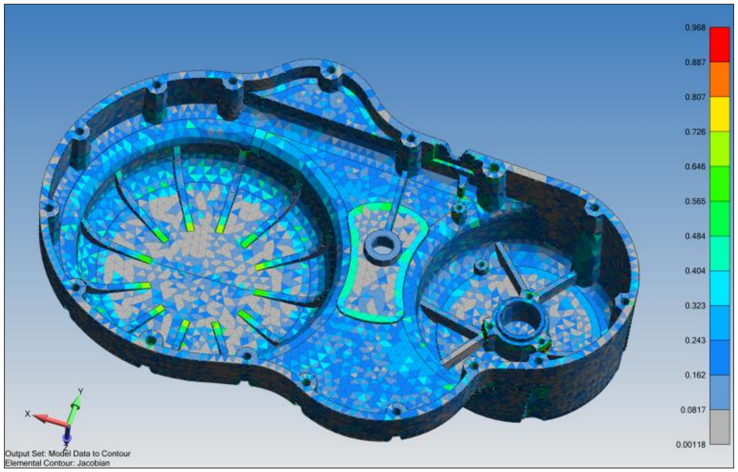


5.2 How to Improve the Quality of Your Tet Mesh

- Pick a part and then start with a reasonable mesh size
- Read the messages and think a bit
- Investigate to see if it seems reasonable
- Try ***Mesh > Geometry Preparation***
- What is it doing in the background?
  - Trimming and splitting
  - Features suppression
  - Boundary surfaces and composite curves
- Check Mesh Quality.

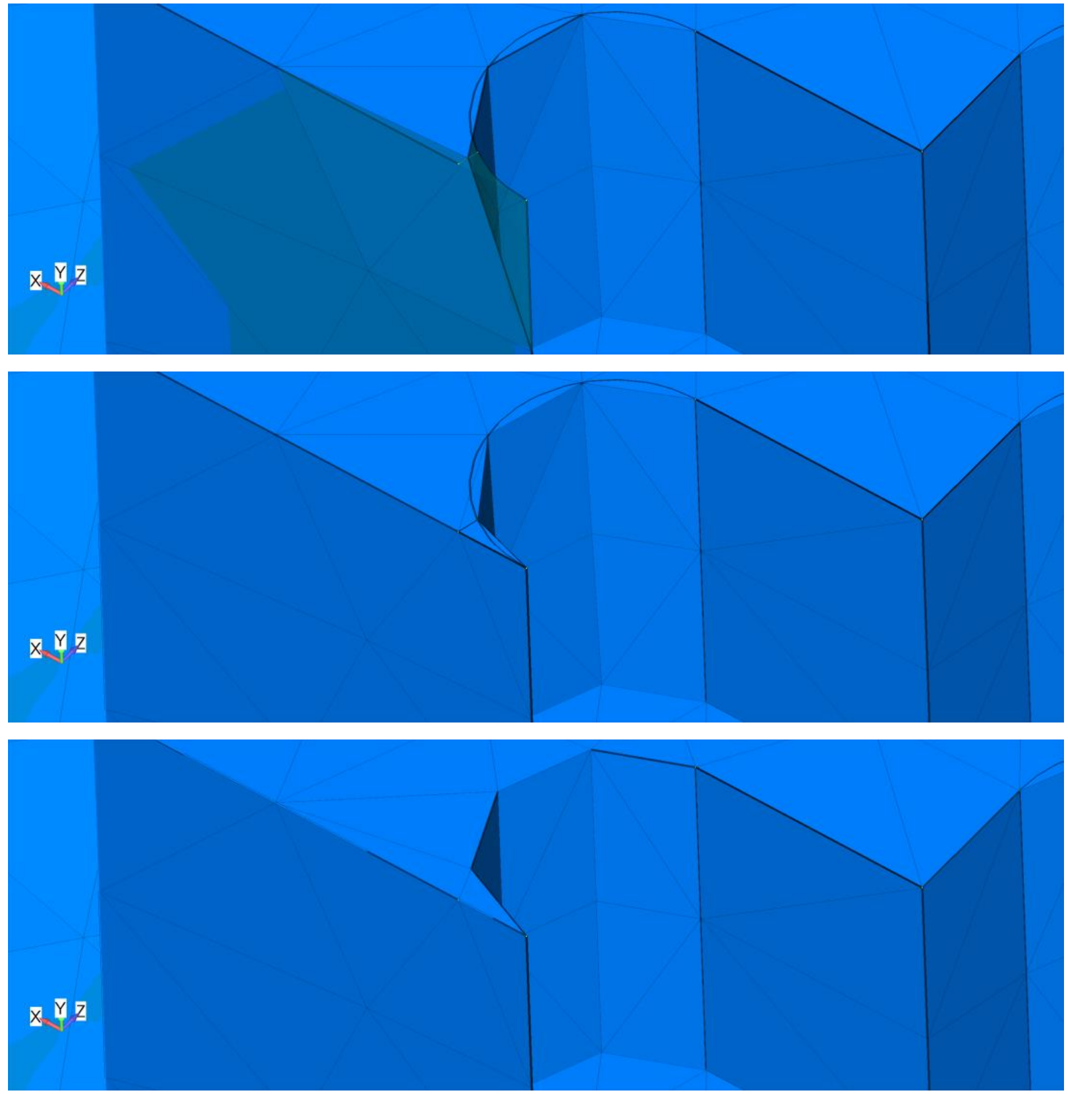






5.3 Troubleshooting Bad Geometry - Brute Force Tet Meshing

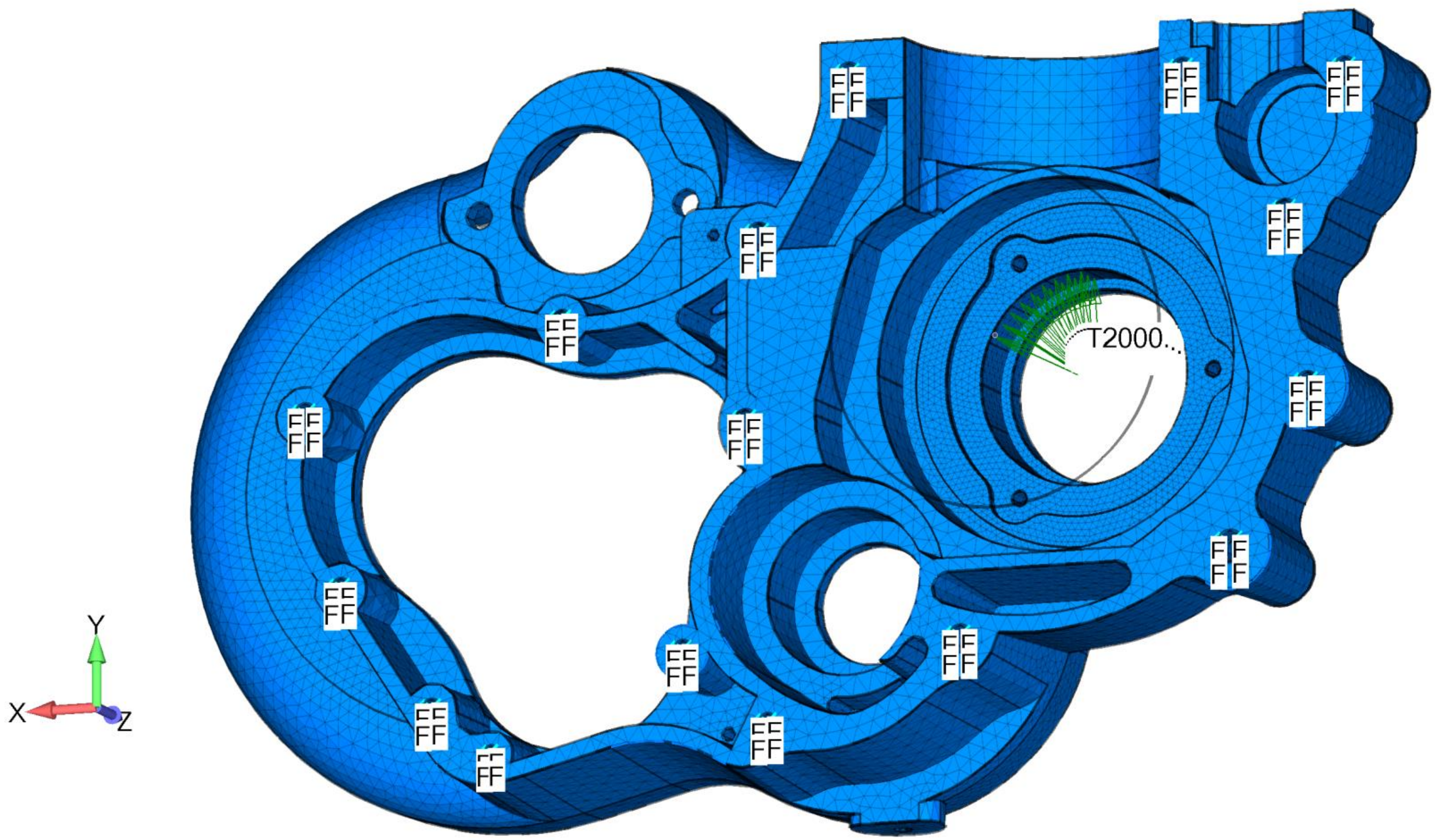
- Try to be systematic
- Look at mesh - identify the geometric features that lead to bad mesh
- Play with the changes generated by Geometry Preparation
- Sometimes, Geometry Preparations creates unwanted changes. Protect these regions with groups.
- Modify the mesh sizing and geometry using the Meshing Toolbox
- Use plot-only elements to test model changes quickly
- Try ***Geometry > Solid > Cleanup***



No magic bullet - No perfect system – Still jobs for simulation engineers ☺

6. BOUNDARY CONDITIONS

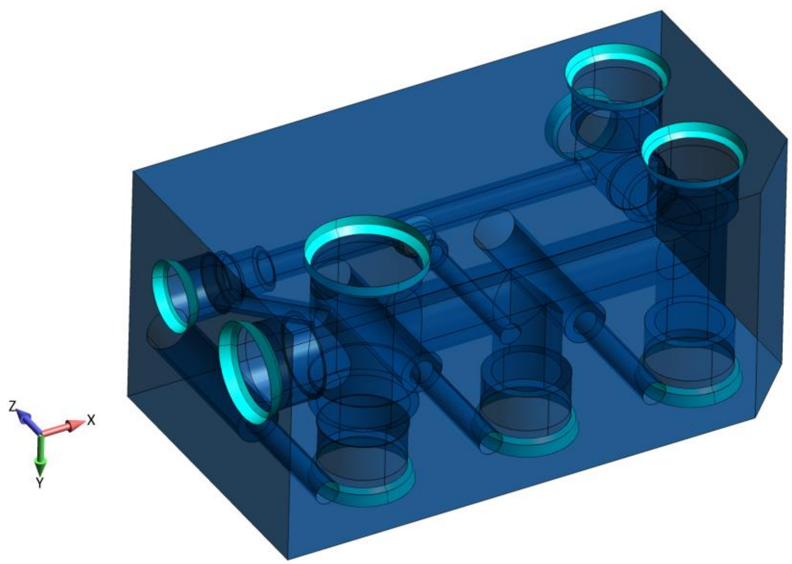
Let’s apply loads and constraints to geometric features. FEMAP will identify certain features and allow for advanced boundary conditions (e.g., cylindrical constraints or bearing loads).

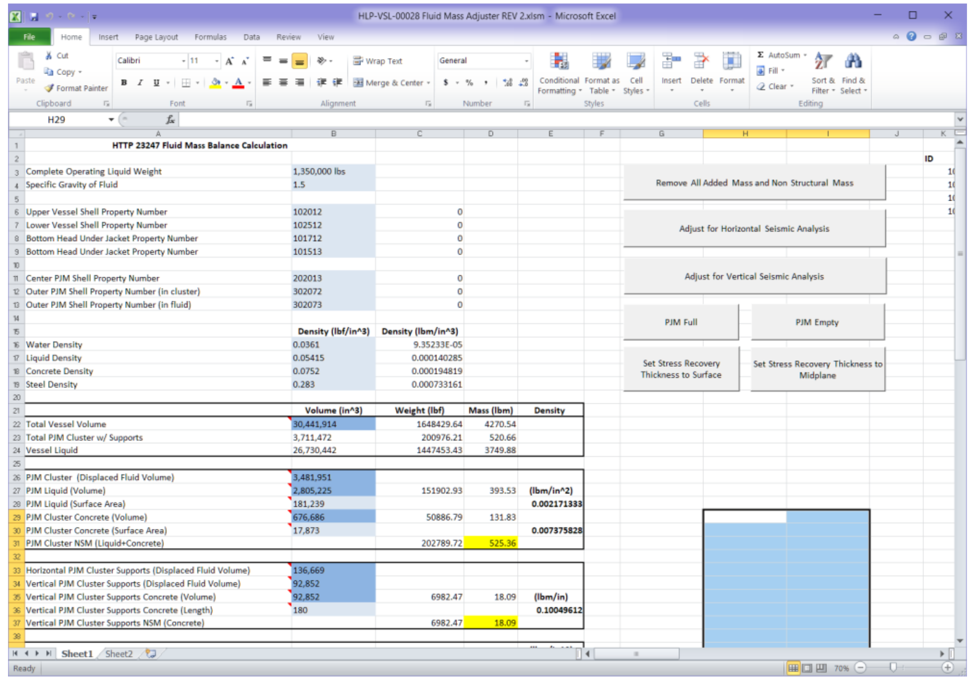


6.1 Application Programming Interface (API)

Over the years, we have saved our clients hundreds of thousands of dollars using FEMAP's API to expedite project work.

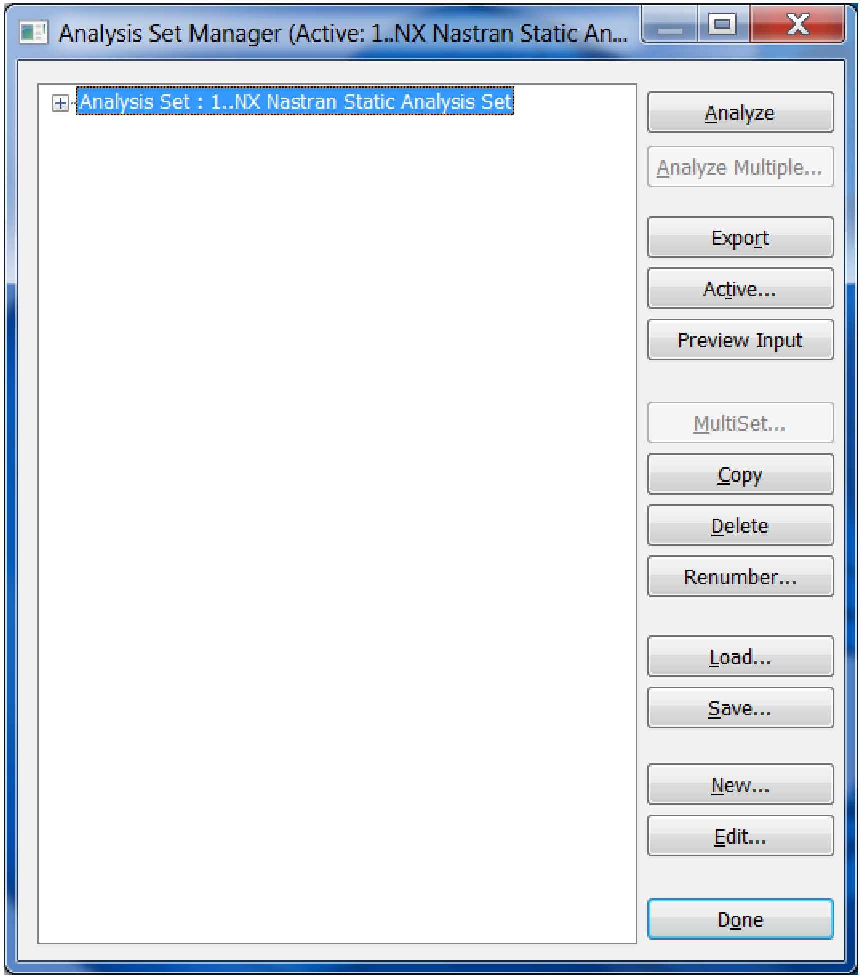
Unique Load Applications	Driving FEMAP via Excel

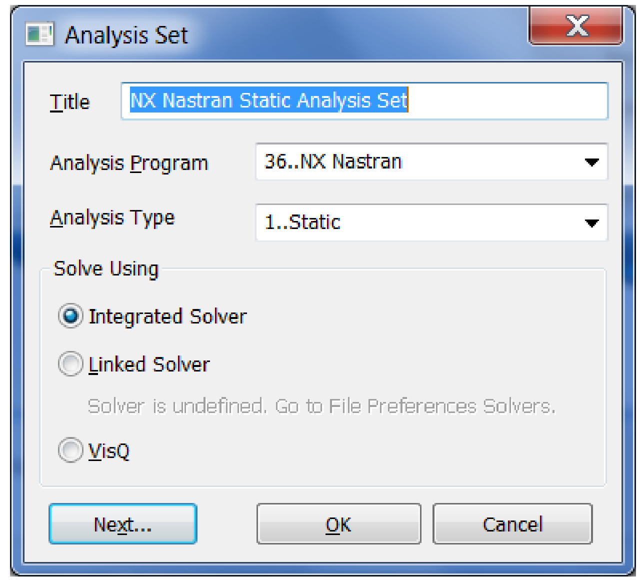




7. ANALYZE

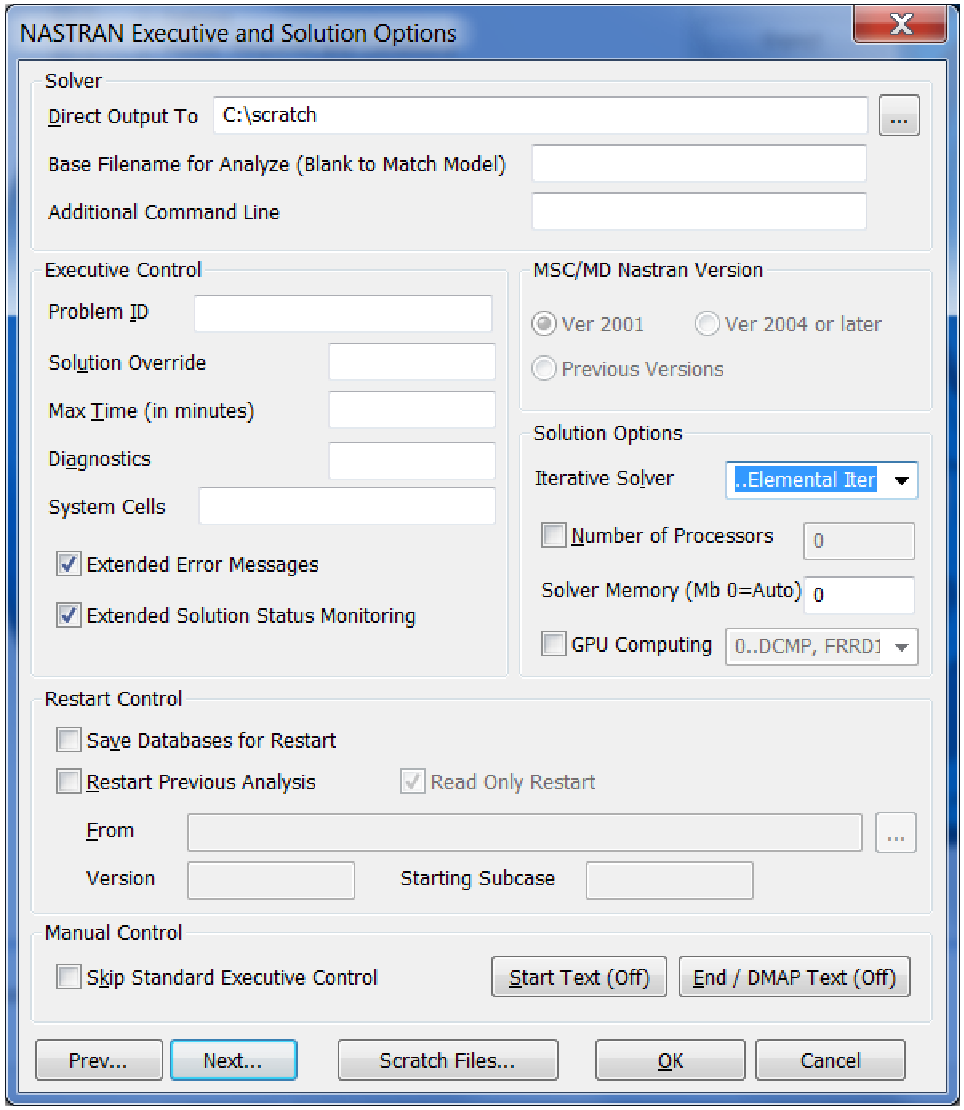
Model > Analysis (36..NX Nastran, 1..Static)





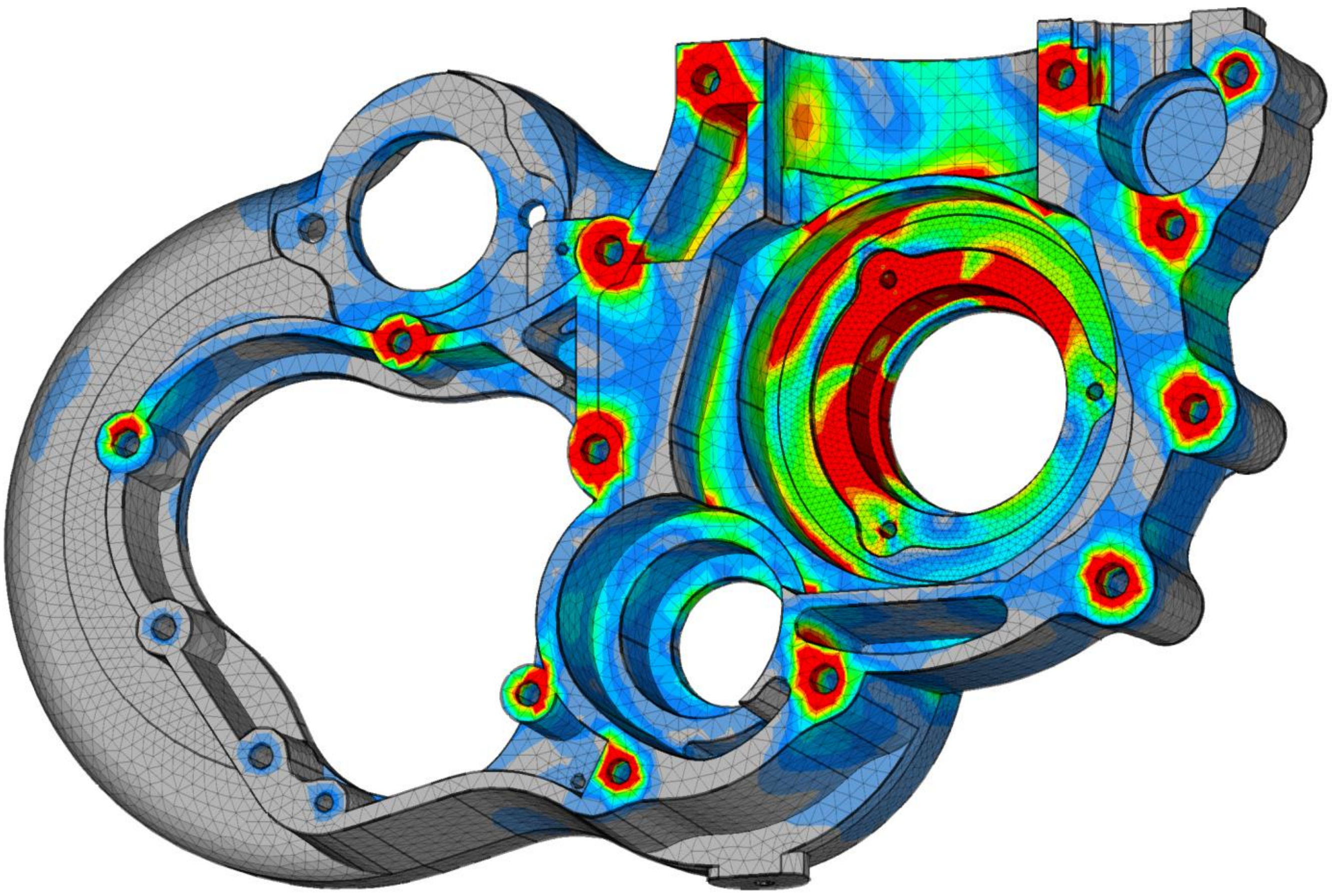
7.1 When Working with Tets, Use the Iterative Solver

When working with a model dominated by tet10 elements, one should use the iterative solver to reduce solve time.

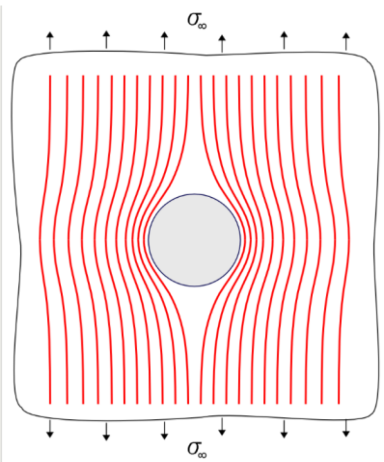


8. POST PROCESSING

Post Processing Toolbox (Contour, Deform)



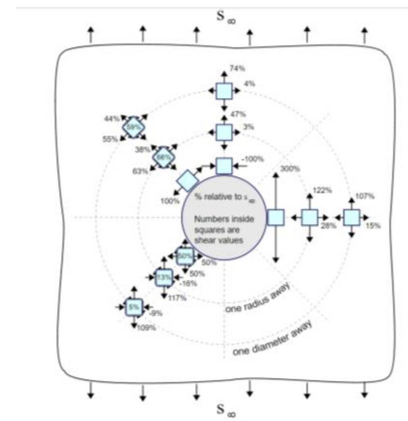
8.1 JUST A QUICK NOTE ABOUT STRESSES



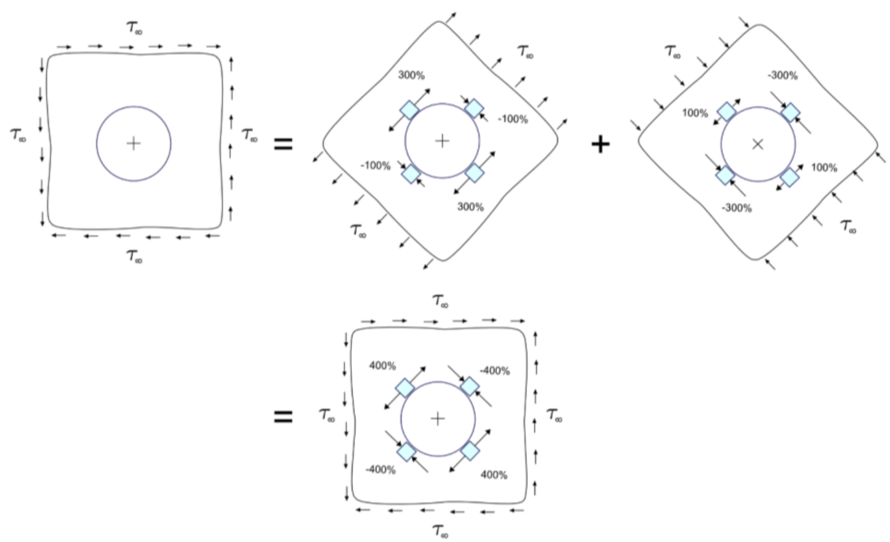
Stress Flows

Stress (2D) in an isotropic solid can be described using the Airy Function that satisfies the *Biharmonic Equation:*

$$\frac{\partial^4\phi}{\partial x^4} + 2\frac{\partial^4\phi}{\partial x^2\partial y^2} + \frac{\partial^4\phi}{\partial y^4} = 0$$

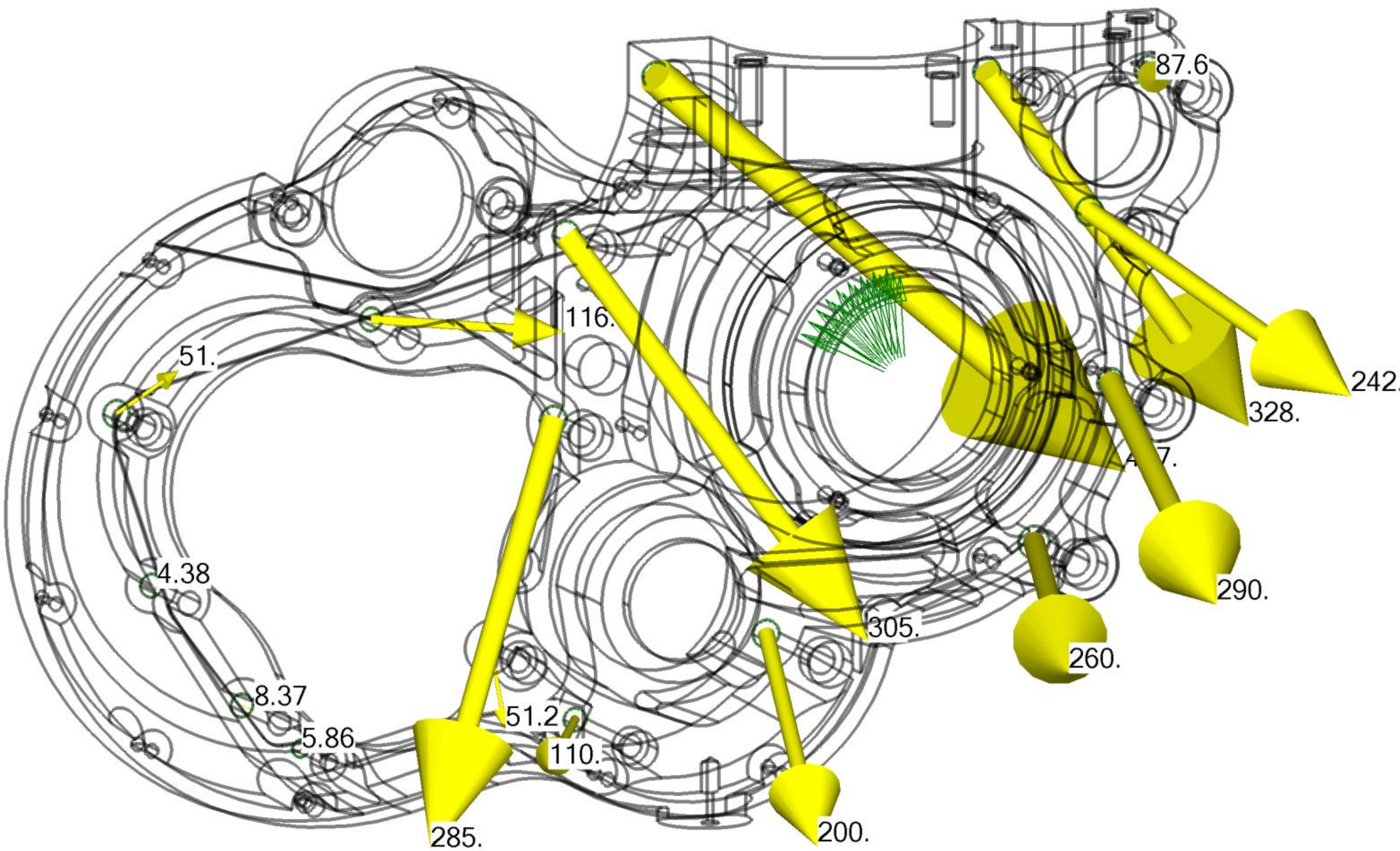


Stresses around holes have stress concentrations of -1 and +3 under pure tension and +4 under pure shear



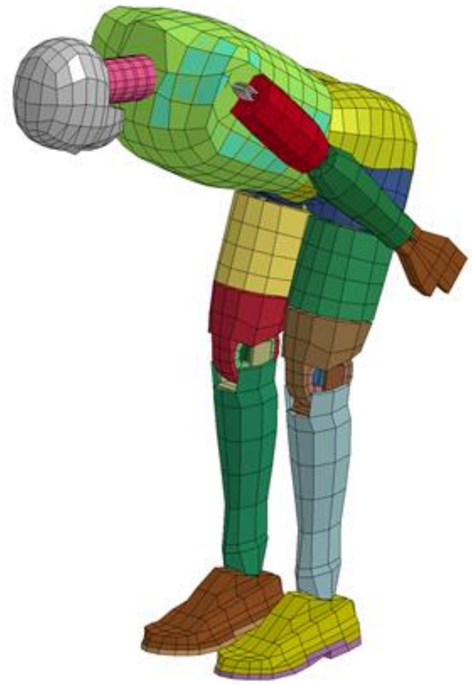
Images courtesy of www.FractureMechanics.org

8.2 FREE BODY DIAGRAM (FBD): WHAT EVERY SIMULATION ENGINEER TRULY LOVES



AND NOW A WORD FROM OUR SPONSORS

Thank You



Predictive Engineering is located in Portland, OR

PredictiveEngineering.com