Femap and NX Nastran Best Practices

Technical Seminar for Femap and NX Nastran Users

Hosted by: George Laird, Ph.D., P.E., Principal Mechanical Engineer

Adrian Jensen, EIT, Senior Staff Engineer

FEA, CFD & LS-DYNA Training, Support and Consulting

Siemens NX CAD, CAM, CAE, Teamcenter and Femap and NX Nastran Sales

1. INTRODUCTION

We plan to cover our short list of model and analysis procedures from building a model to running it. We are keeping it short since a longer list is rarely implemented since one rarely remembers "everything". Thus we are focusing on the big stuff that will make the *difference* in your modeling efforts and the *quality* of your analysis product.

1.1 MODELING

- Avoid complexity with a passion.
- Small is beautiful
- Clean geometry is your friend
- A perfect quad is craftsmanship at its finest
- Force, mass, dimension checks are your first defense

1.2 ANALYSIS

- Stress just flows
- What goes in has to equal what goes out
- Vibration is about mass and constraints

1.3 ELEGANT SIMPLICITY IS DECEPTIVELY DIFFICULT TO ACHIEVE

2. MODELING

2.1 AVOID COMPLEXITY

2.1.1 WHEN ONE CAN JUST GLUE IT

When you build simple, one should build simple. What does this exactly mean? If the structure is appropriate, use glued contact to join up meshes to create assemblies of larger parts. NX Nastran has very good technology for joining tets-to-hexs, plates-to-solids and likewise plates-to-plates.

Hex-to-Tet

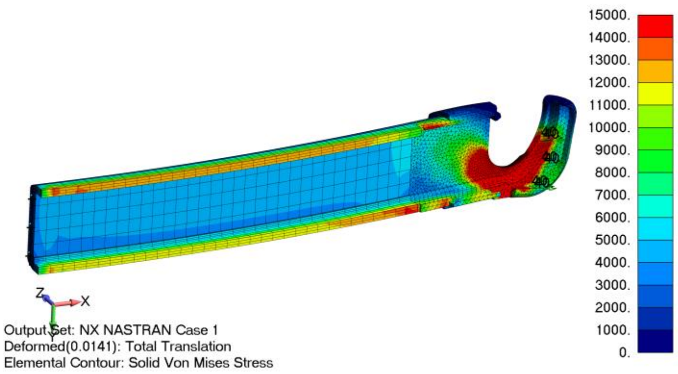


Plate-to-Solid

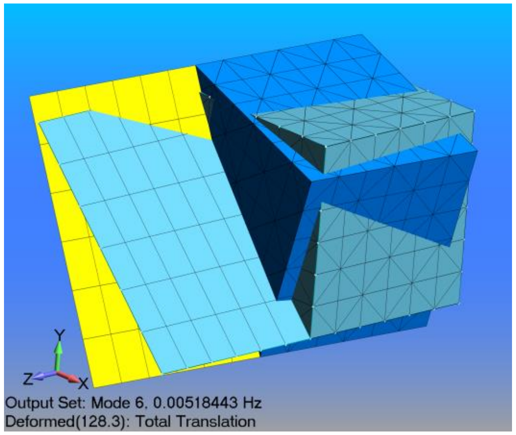
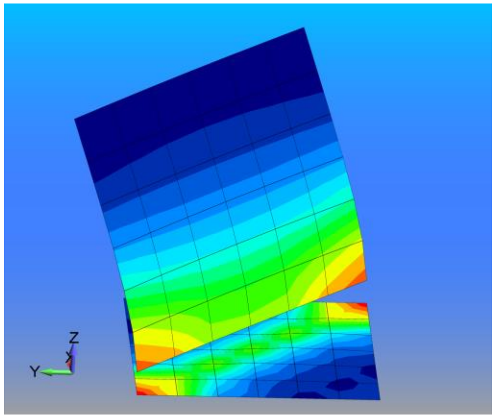


Plate Edges-to-Whatever



For more information see our Seminar: Hex Meshing

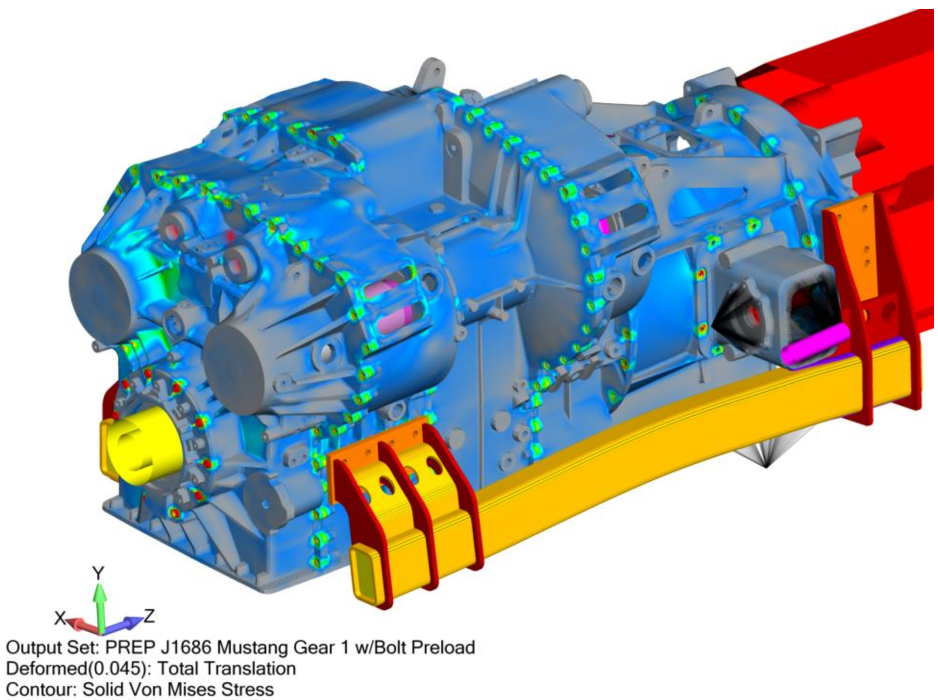
Assembly Modeling for Optimization

2.2 SMALL IS BEAUTIFUL

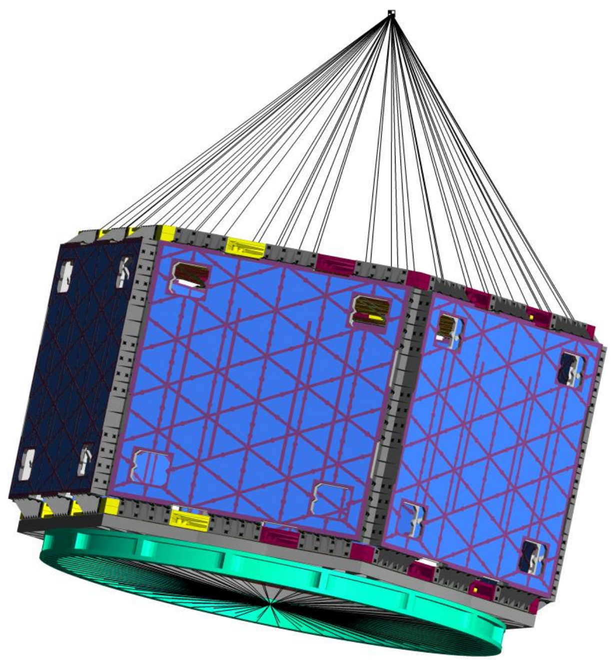
2.2.1 TRY NOT TO MODEL BOLTS BUT USE RBE2 AND BEAM ELEMENTS

Although this might sound obvious but one of the real benefits is that one can directly extract bolt axial and shear forces for downstream bolt calculations. And RBE2 and their cousin, RBE3 are great for opening doors to idealization efficiency.

RBE2 with Beam Elements: Bolt Preload



RBE3 to Smear Mass and Force

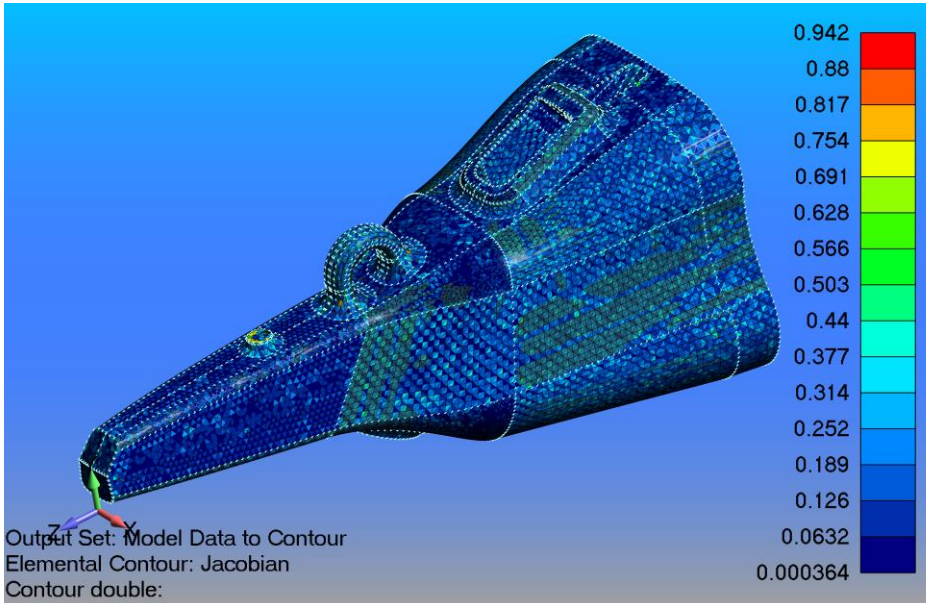


For more information see our Seminar: Connections 2013

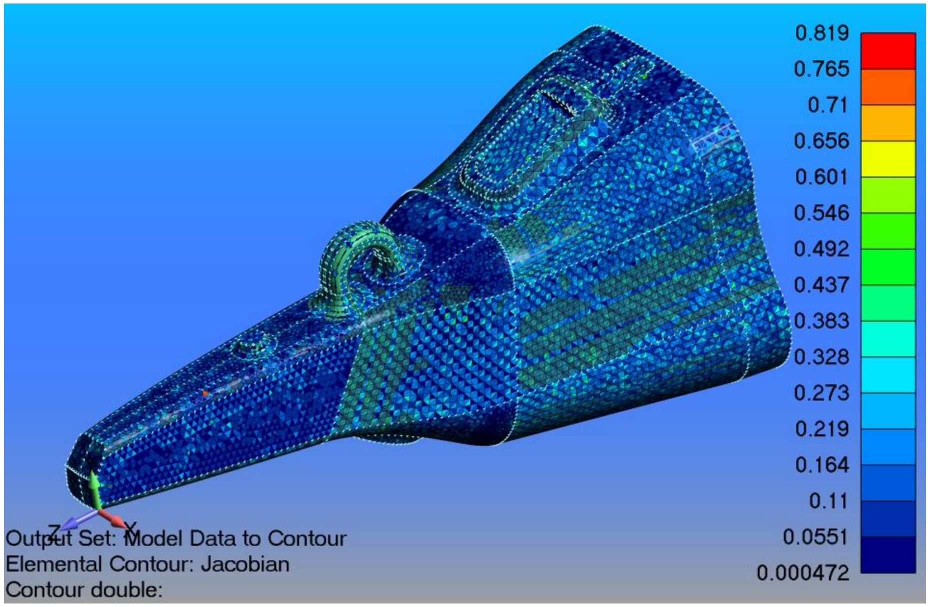
2.2.2 EVEN IF YOU CAN SOLVE 10,000,000 NODES, WHAT THEN?

Even though you can go gigantic, use Mesh / Geometry Preparation to create better elements and then force two elements through thickness without having to add god-zillon elements.

Always Jacobian (J > 0.942 Nastran Problems)

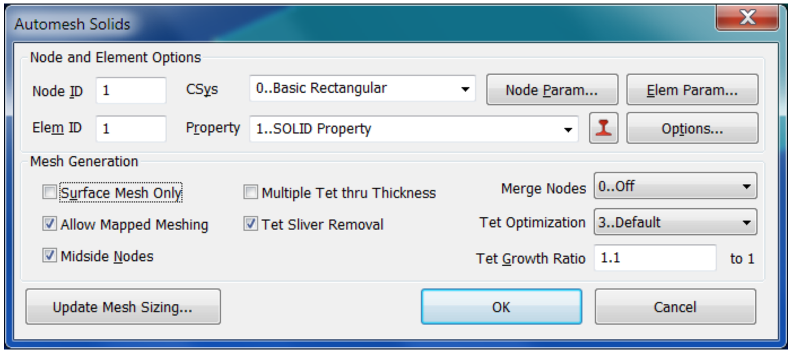


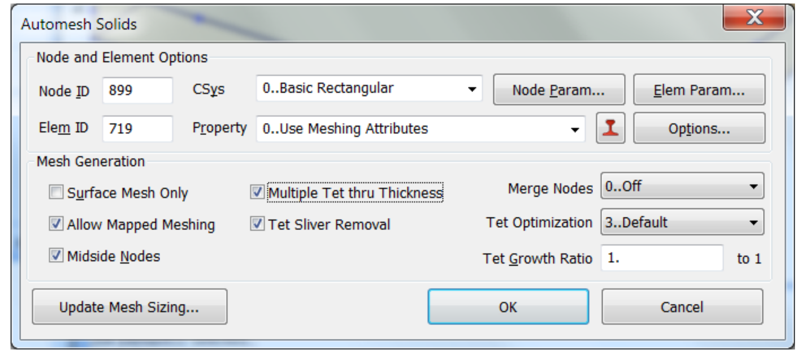
Mesh / Geometry Preparation (J < 0.9 Runnable)

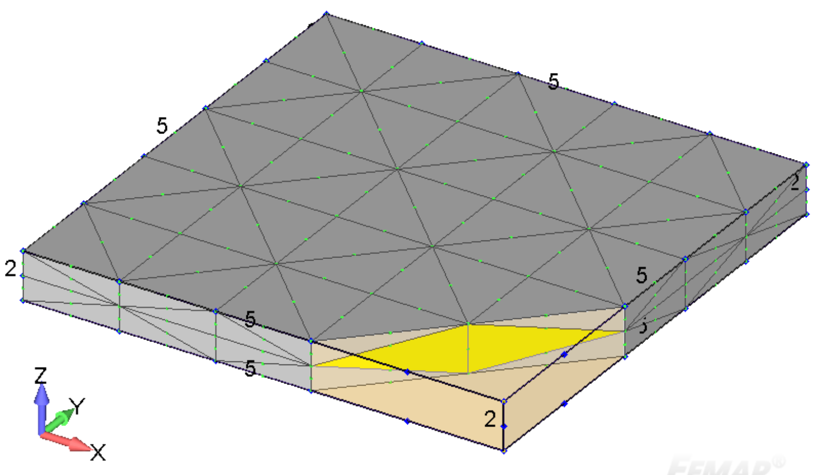


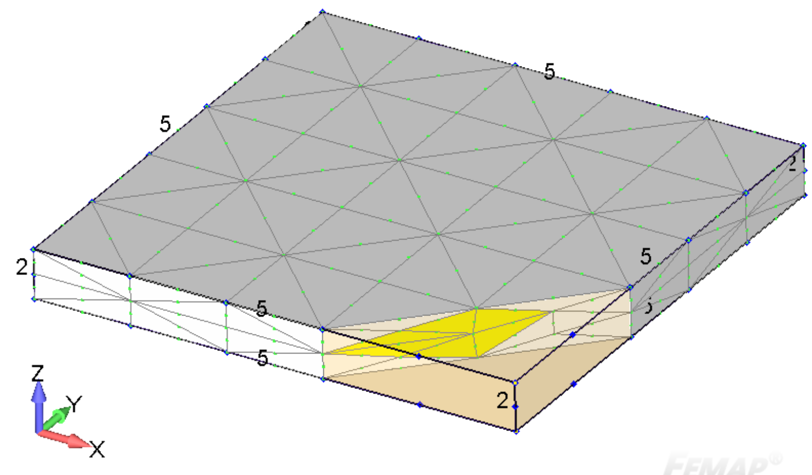
For more information see our Seminar: Femap and NX Nastran Update

But the real question is whether or not you are getting the mesh that you need?









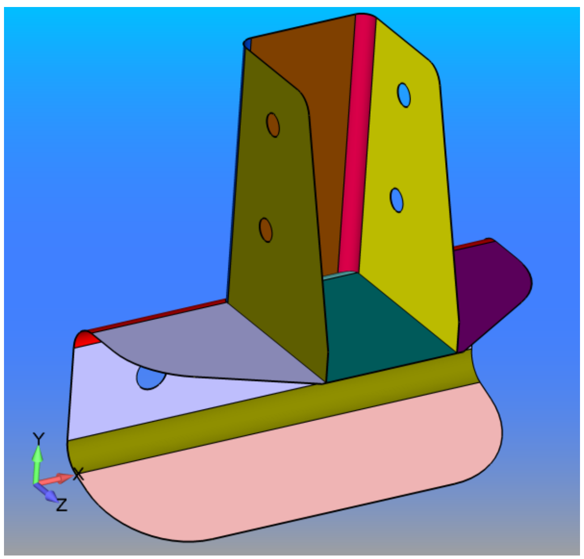
For more information, stay tuned for the Femap v11.1 seminar (~December 2013).

2.3 CLEAN SKIN GEOMETRY PREVENTS MESH PATHOLOGIES

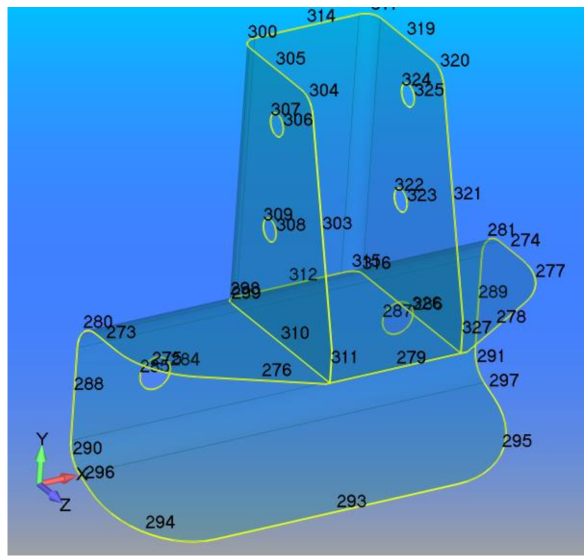
2.3.1 MANIFOLD ALWAYS AND NON-MANIFOLD AT THE END

We know it sounds simple but it requires up-front work for down-stream payoffs. We won't sermonize too much but a clean foundation ensures many other benefits than just a beautiful mesh.

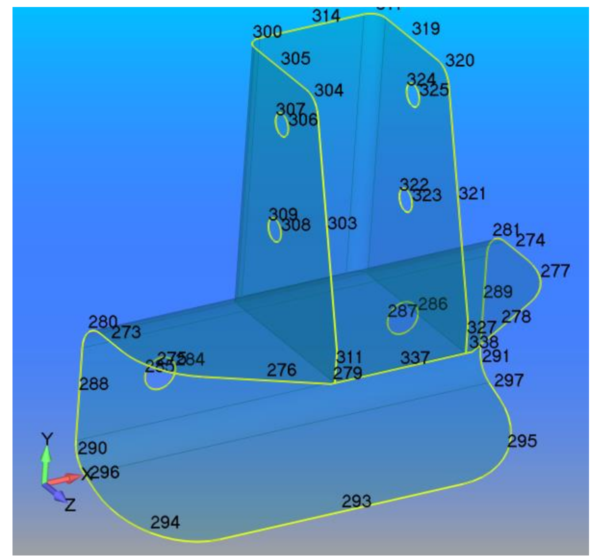
A bunch of Surfaces



Manifold



Then Non-Manifold Add

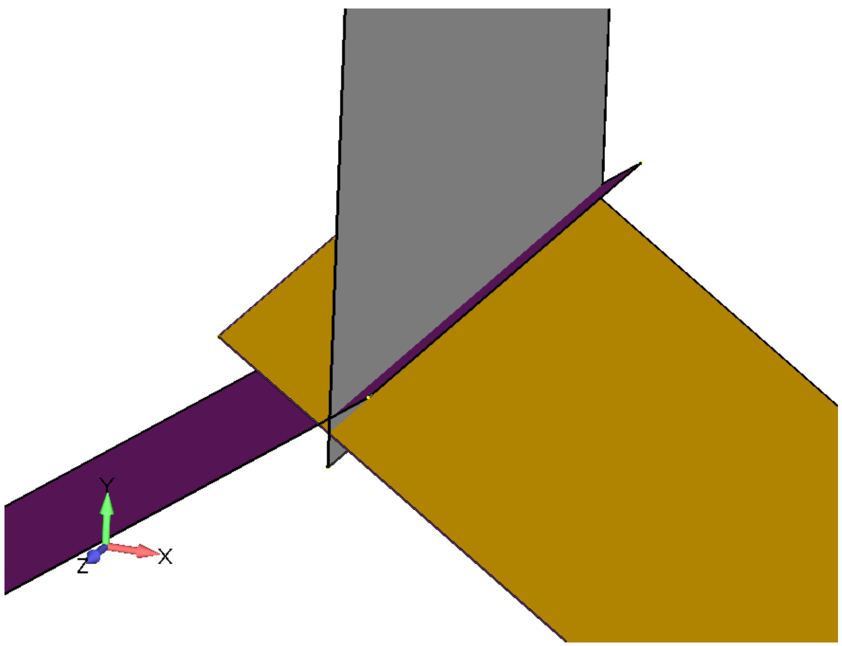


For more information see our Seminar: Surface Modeling
Working with Surfaces

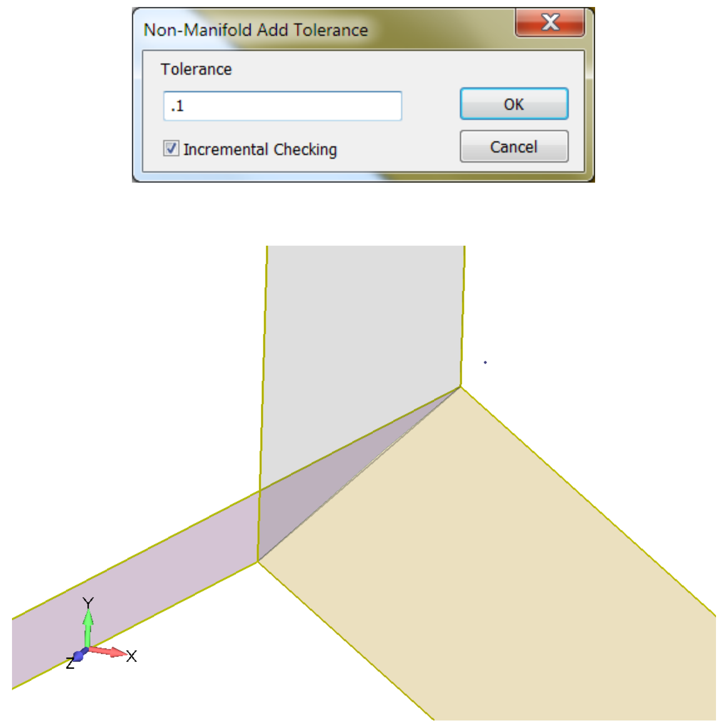
2.3.2 STAY ABREAST OF NEW DEVELOPMENTS

Femap v11.1 has solved many problems with non-manifold add

A Chunk of Junk



Non-Manifold the Impossible

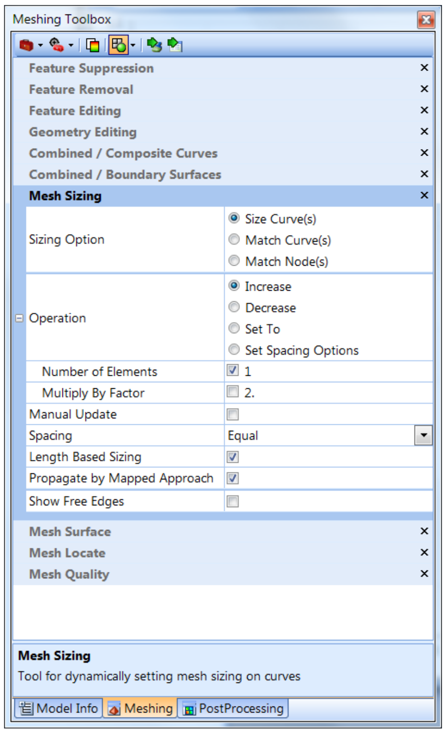


For more information, stay tuned for the Femap v11.1 seminar (~December 2013).

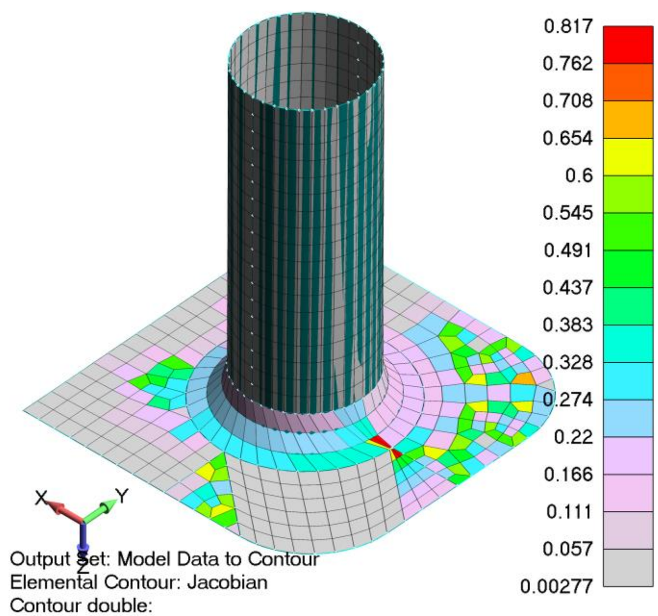
2.4 A PERFECT QUAD IS FEA CRAFTSMANSHIP AT ITS FINEST

Stresses are calculated at integration points and that clean stresses ensure stress visualization clarity and helps one debug your model as to reality or artifact. Or in other words; "***If it looks good, it is good***." Nevertheless, always check your Jacobian. To get to quad nirvana, embrace using the Mesh Toolbox.

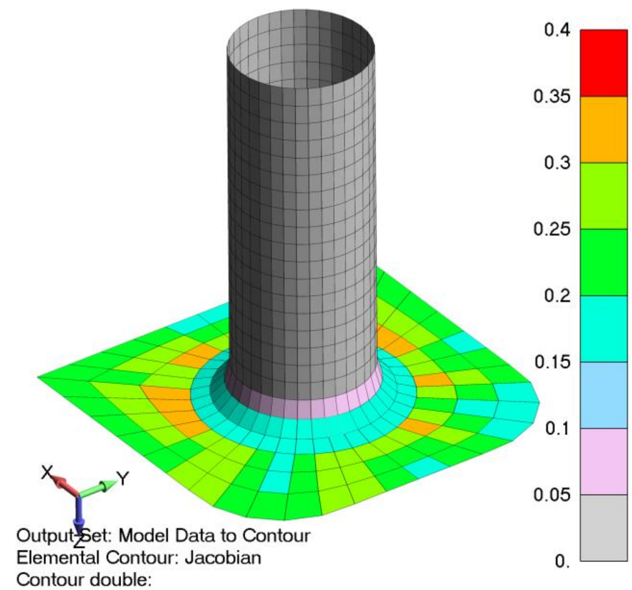
The Mesh Toolbox



Raw Mesh



Quad Perfection



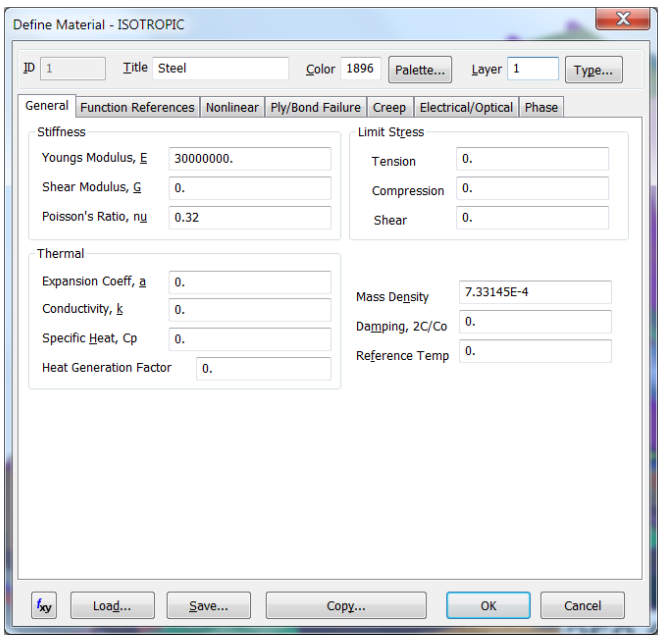
For more information see our Seminar: Stress Visualization and Validation
Surface Modeling

2.5 FORCE, MASS, DIMENSION CHECKS ARE YOUR FIRST DEFENSE

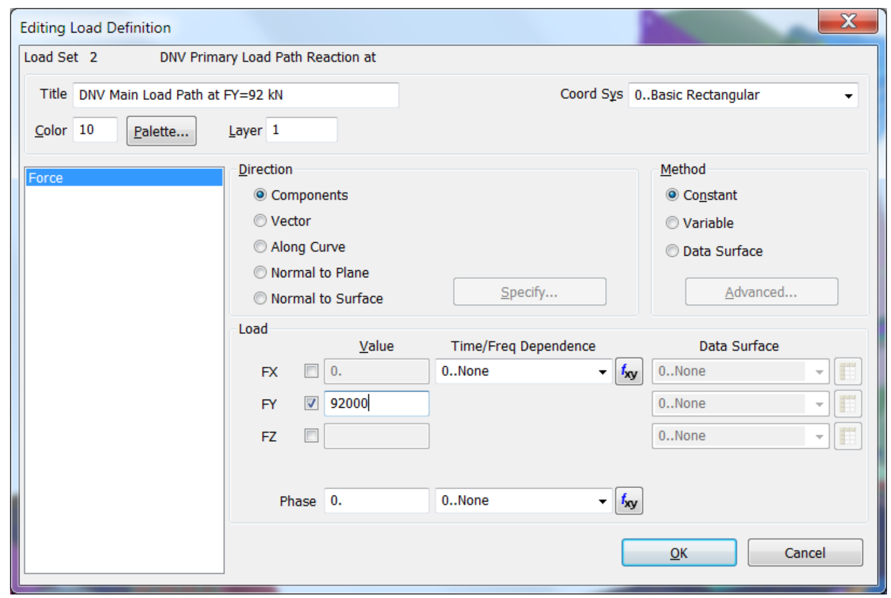
2.5.1 UNIT CHECK AND SIGNIFICANT DIGITS

Verify that force, elastic moduli, acceleration, mass density and dimensions represent a consistent set of units. It sounds obvious but one might be surprised how a digit or two can get slipped in among friends. We also recommend to adhering to significant digits in all analysis entries. Why? It makes debugging easier and implicitly demonstrates that you are aware of the accuracy of your analysis work.

Material Card is "Bare-Bones"



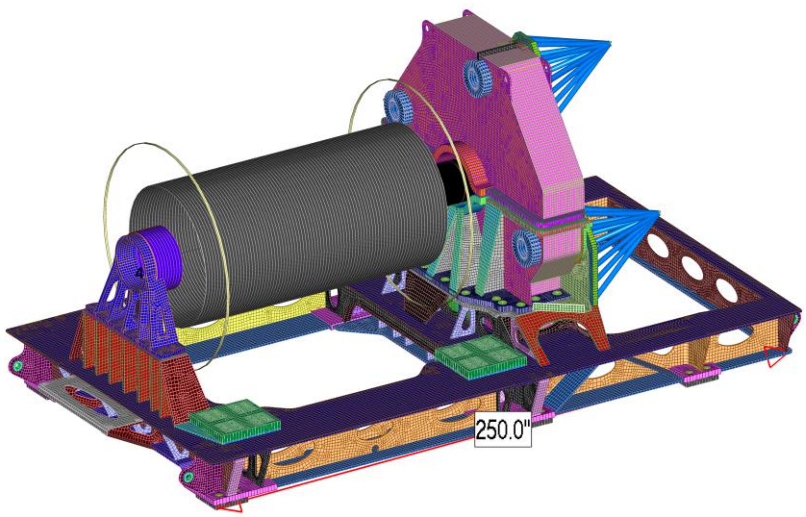
Descriptive Load Names
"DNV Main Load Path at FY=92 kN"



2.5.2 CHECK MENU

We always provide dimensional, force and mass verification in our reports using direct Femap tools.

The GFX Measure Tool for Quick Dimensioning



Check Everything and Document It

Check Sum of Forces

```
Summation of Forces, Moments and Pressures for Set 2 (CSys 0)
  Nodal Force                  FX =           0.  FY =      366000.  FZ =            0.
  Nodal Moment                 MX =     5607750.  MY =           0.  MZ =            0.
  Pressure Force               FX =           0.  FY =           0.  FZ =            0.
  Body Translational Accel     FX =           0.  FY =     -116217.  FZ =            0.
  Body Varying Trans Accel     FX =           0.  FY =           0.  FZ =            0.
  Body Rotational Accel        FX =           0.  FY =           0.  FZ =            0.
  Body Rotational Velocity     FX =           0.  FY =           0.  FZ =            0.

Totals (CSys 0)
  About Location                X =           0.   Y =           0.   Z =            0.
  Forces                       FX =           0.  FY =      249783.  FZ =            0.
  Moments                      MX =    12901551.  MY =           0.  MZ =     14351206.
```

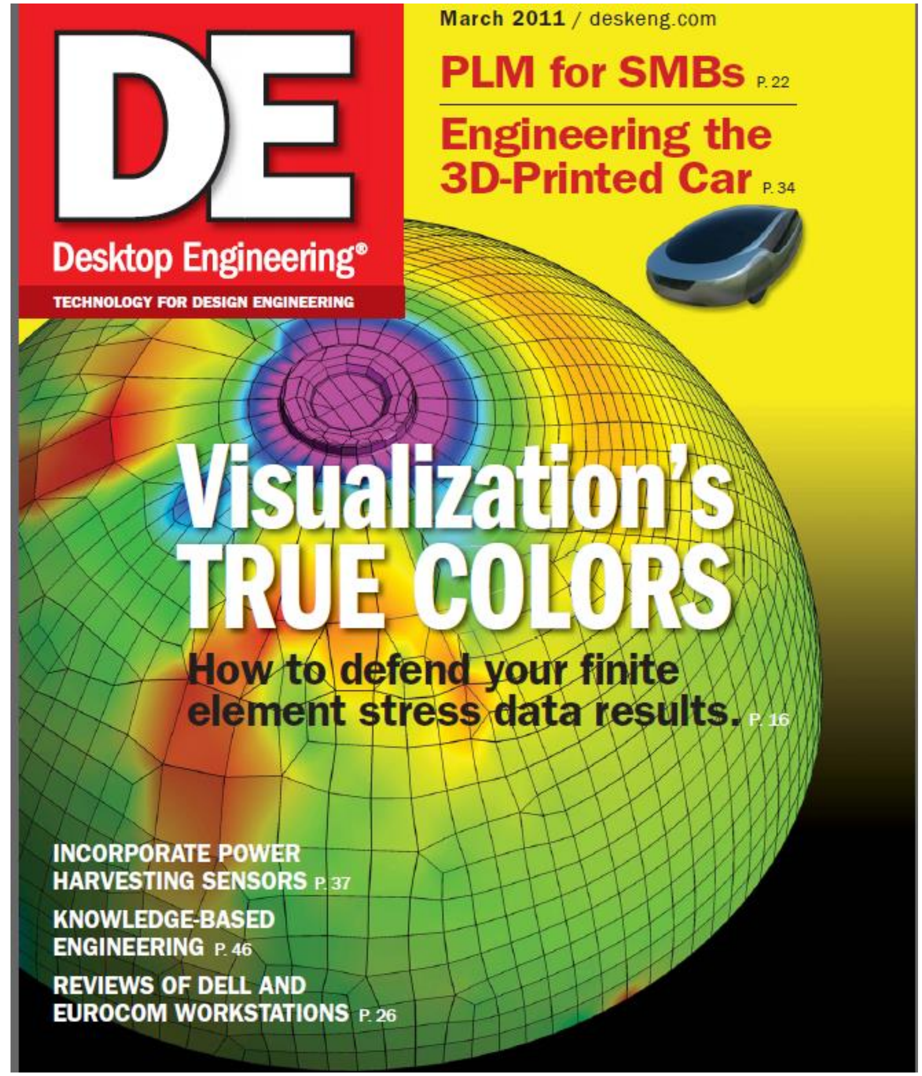
To obtain the GFX tool (it is an API), it is buried in our Seminar: Analysis Workflows (See our website www.PredictiveEngineering.com)

3. ANALYSIS CHECKOUT

If you have done everything up to this point, your model should be really tight and you shouldn't have too many surprises. But knowing full well the quote: ***"All models are wrong, but some models are useful."*** We have a few more checks to perform.

3.1 STRESS JUST FLOWS

- If it looks good, it is good
- Stress should never look uneven since it just flows
- Stress is computed at Guass points and then extrapolated to nodes and averaged
- Artifacts can be explained via basic principles and don't need a fancy explanation



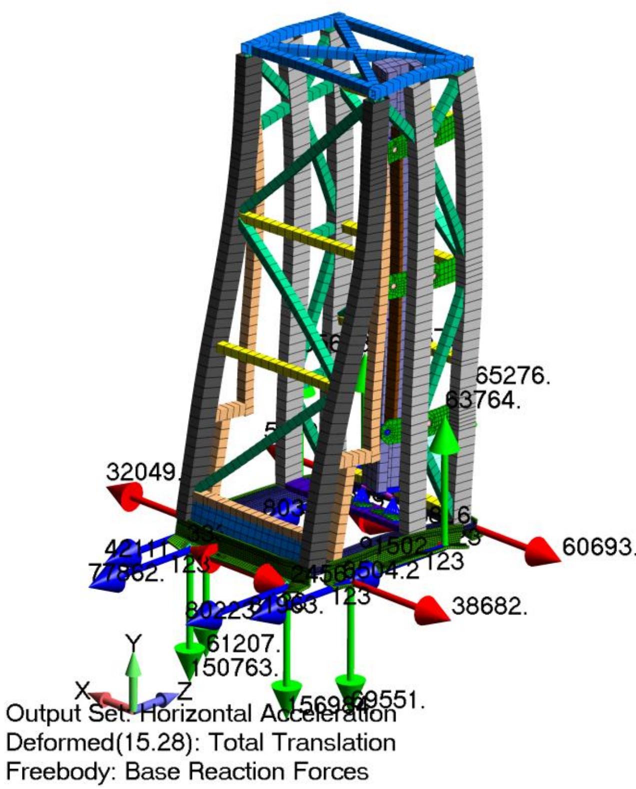
For more information see our Seminar: Stress Visualization and Validation

3.2 WHAT GOES IN HAS TO EQUAL WHAT GOES OUT

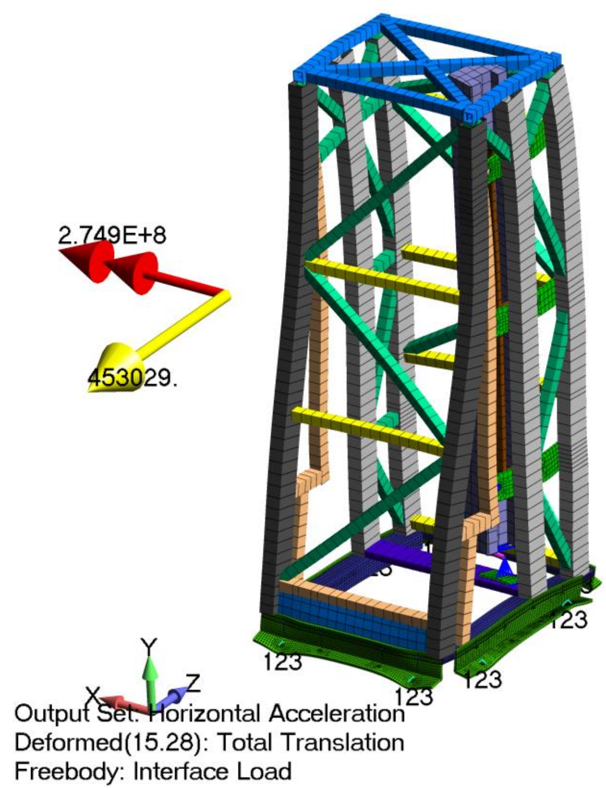
3.2.1 FBD AS IF YOUR LIFE DEPENDED UPON IT

Never underestimate the utility of a simple FBD check based on reaction forces. This is the basic check and one doesn't need to request any special outputs and it works for linear and nonlinear analysis.

Just the Reaction Forces



Summing All Reaction Forces



For more information see our Seminar: Femap and NX Nastran Update

3.3 VIBRATION IS ABOUT MASS AND CONSTRAINTS

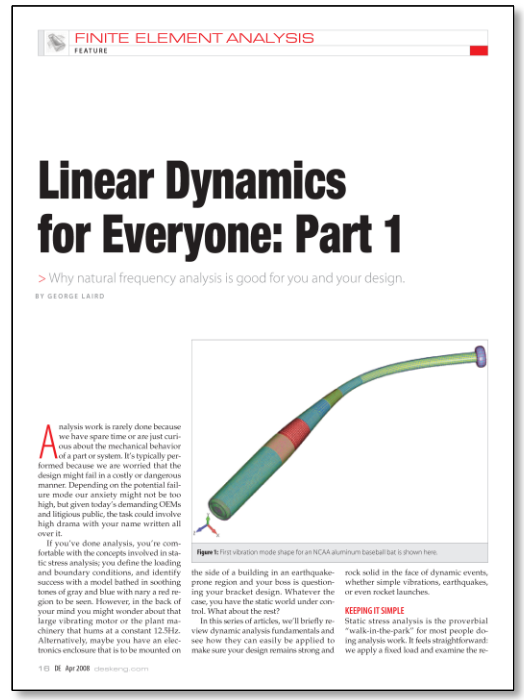
3.3.1 CHECK F06 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

Although this is just another check, we wanted to let you guys know

FO6 Check-Out Basics

- Do the element types and numbers make sense?
- Does the model mass exactly match that reported in the "OUPUT FROM GRID POINT WEIGHT GENERATOR"?
- Error and Warning Messages?

Vibration Analysis White Paper



FINITE ELEMENT ANALYSIS
FEATURE

Linear Dynamics for Everyone: Part 1

> Why natural frequency analysis is good for you and your design.

BY GEORGE LAIRD

Analysis work is rarely done because we have spare time or are just curious about the mechanical behavior of a part or system. It's typically performed because we are worried that the design might fail in a costly or dangerous manner. Depending on the potential failure mode our anxiety might not be too high, but given today's demanding OEMs and litigious public, the task could involve high drama with your name written all over it.

If you've done analysis, you're comfortable with the concepts involved in static stress analysis; you define the loading and boundary conditions, and identify success with a model bathed in soothing tones of gray and blue with nary a red region to be seen. However, in the back of your mind you might wonder about that large vibrating motor or the plant machinery that hums at a constant 12.5Hz. Alternatively, maybe you have an electronics enclosure that is to be mounted on the side of a building in an earthquake-prone region and your boss is questioning your bracket design. Whatever the case, you have the static world under control. What about the rest?

In this series of articles, we'll briefly review dynamic analysis fundamentals and see how they can easily be applied to make sure your design remains strong and rock solid in the face of dynamic events, whether simple vibrations, earthquakes, or even rocket launches.

Figure 1: First vibration mode shape for an NCAA aluminum baseball bat is shown here.

KEEPING IT SIMPLE

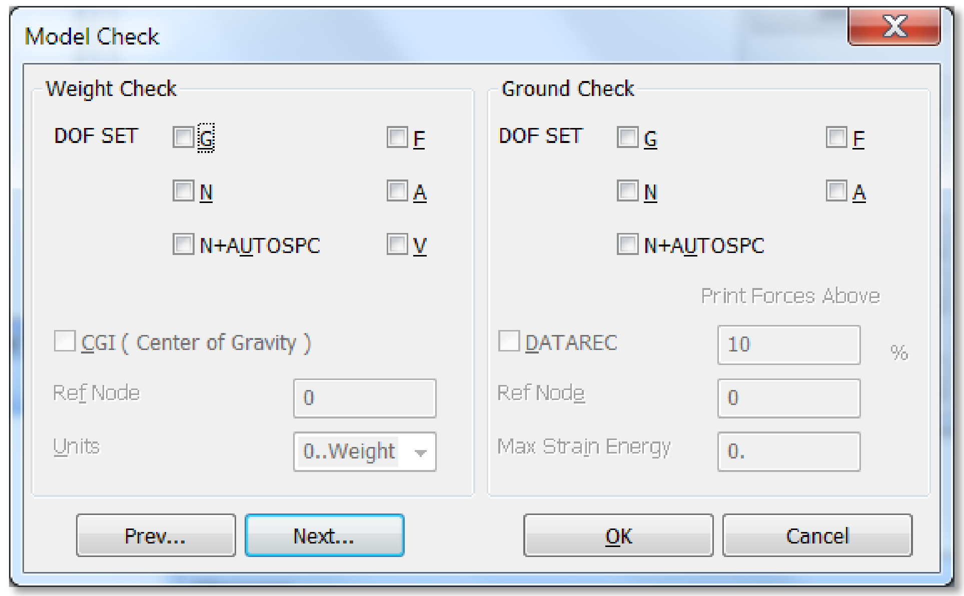
Static stress analysis is the proverbial "walk-in-the-park" for most people doing analysis work. It feels straightforward: we apply a fixed load and examine the re-

16 DE Apr 2008 deskeng.com

For more information see our Seminar: Normal Modes Analysis

3.3.2 GROUND CHECK IF YOU ARE DOING AEROSPACE QUALITY WORK

This check-out technique provides a numerical proof that your stiffness matrix is up-to-snuff. It is a rather dry subject and we'll leave it up to the seminar to flesh-out exactly how to do Ground Check, but if you have ever wondered what this screen does – this is your opportunity.



For more information see our Seminar: What is Groundcheck?

4. ELEGANT SIMPLICITY IS DECEPTIVELY DIFFICULT TO ACHIEVE

A good modeler:

- Saves their model locally to a unique name every couple of hours. We append “WIP” to our model names and then save them as WIP A, WIP B, WIP C, etc. Moral of the story, disk space is cheap, LAN is slow and one can always recover quickly from heading down the wrong path or an “accident”;
- Performs all checks prior to releasing the model to their colleagues and of course, production;
- Uses symmetry even if they have to spend 15 minutes explaining that it have no effect upon the solution accuracy since it enforces a verification check on geometry and loads;
- Employs beam elements since they are the ultimate in optimization flexibility and enforces a certain modeling discipline to use them effectively (and likewise for plates elements);
- Is comfortable with RBE2 and RBE3 elements and understands that they are multi-point constraints and not suitable for geometrically nonlinear analysis but just fine for most mildly non-linear material analyses;
- Strives to hex mesh but is also quick to pull the plug and tex-mesh that which belongs to tet meshing and then use Glued Connections to handle the tet-to-hex transition;
- Avoids the use of Contact for Multi-Part Assemblies but when absolutely necessary, does it locally, simply and with full-knowledge of the cost/benefit analysis since the solution is going to take a big performance hit.
- Reads the manuals, attends technical seminars, creates stupid simple “pilot” models when embarking upon new analysis techniques and calls technical support when roadblocks are hit.

5. TRAINING OPPORTUNITIES

LS-DYNA Analysis for Structural Mechanics, January 27-30, 2014

Femap and NX Nastran Training, April 21-25, 2015