Beams: a sneaky, simple and complex element

A Seminar for Femap and NX Nastran Users

Kyle Hamilton | Staff Mechanical Engineer
Adrian Jensen, PE | Senior Staff Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com

AppliedCAx

Siemens PLM Software Sales & Support
CAD ι CAM ι CAE ι Teamcenter

www.AppliedCAx.com

Our Focus:

- Ensuring our clients are utilizing FEA tools in the most effective fashion

How We Can Help:

- Superior technical support and ongoing training
- Switching maintenance to us costs you nothing

Upcoming Training:

- Our next hands-on FEMAP training class is October 10-14th in Portland, OR. Join us!

Interested in becoming a client, or in training?

Please contact Erik Barber at:
(503) 962-0287
Erik.Barber@AppliedCAx.com

**Applied CAx**
We do this *every day*

TABLE OF CONTENTS

1. INTRODUCTION TO BEAMS

1.1 THEORY

Beam elements in Femap are powerful elements yet take special care to ensure they are used properly. The NX Nastran Theoretical Manual says it best:

> *"Although it is one of the simplest of structural elements and one that is well known to everybody, the beam has been a troublesome element in the development of NASTRAN, due to difficulty in selecting the properties that it should have. **In retrospect it seems clear that the versatility of the beam concept is the cause of the difficulty.**"*

In practice, only two nodes and a cross section are required to create an element. Beams each have a unique coordinate system— the X-axis is down the length of the beam from End A to End B, and the Y-axis is defined by the user in the beam definition dialog. Each end of a beam has 6 degrees of freedom and any of these 6 can be released at either end.
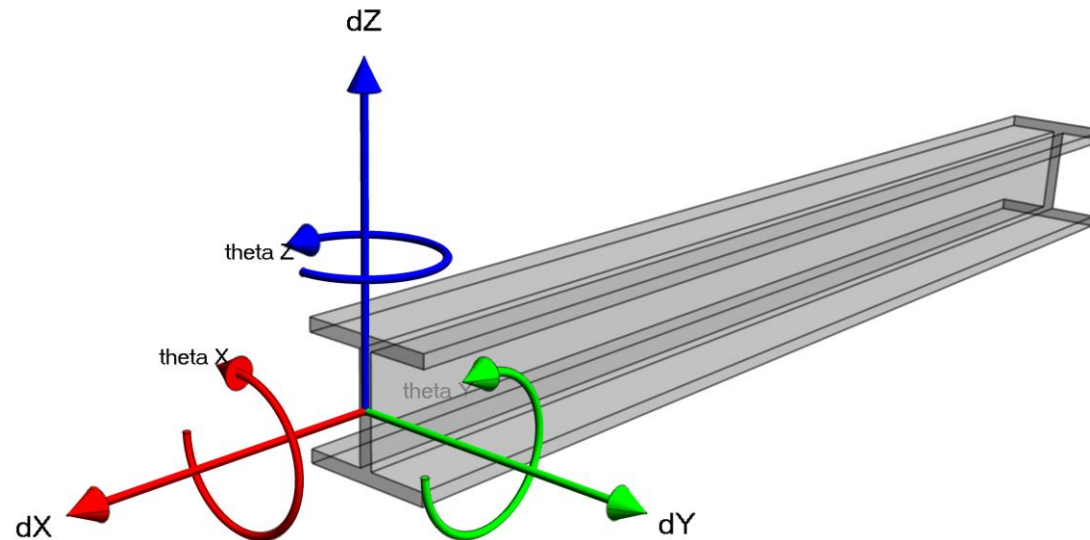
Before jumping into the model it is important to understand the terminology used in Femap, it is easy to get lost in the jargon of beam elements. The diagram below is from the NX Nastran Element Guide and provides some guidance on the gobbledygook of beam terminology.

The important terms to remember today are:

Beam End A: The end of the beam at the first node selected

Beam End B: The end of the beam at the second node selected

Plane 1: Plane through the beam defined by the local Y axis and X axis

Plane 2: Plane through the beam defined by the local Z axis and X axis
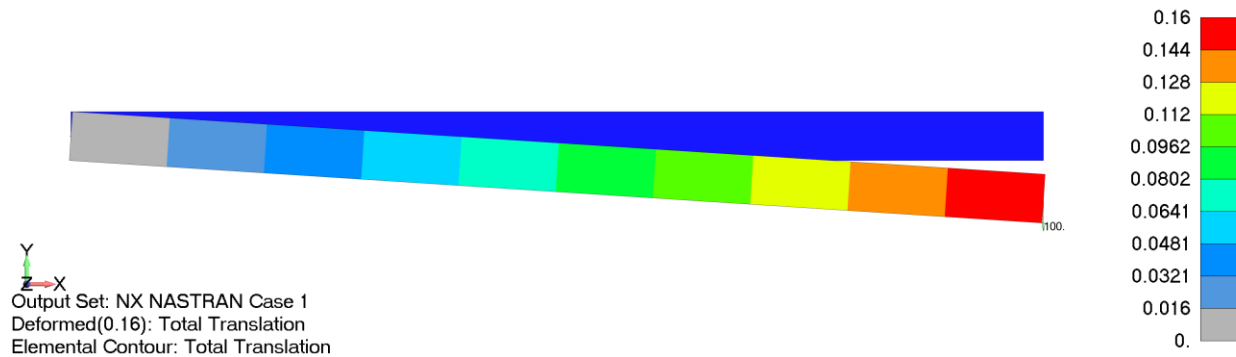
Combined Stress: Sum of the Axial, shear, and bending stress (note: torque is not included)
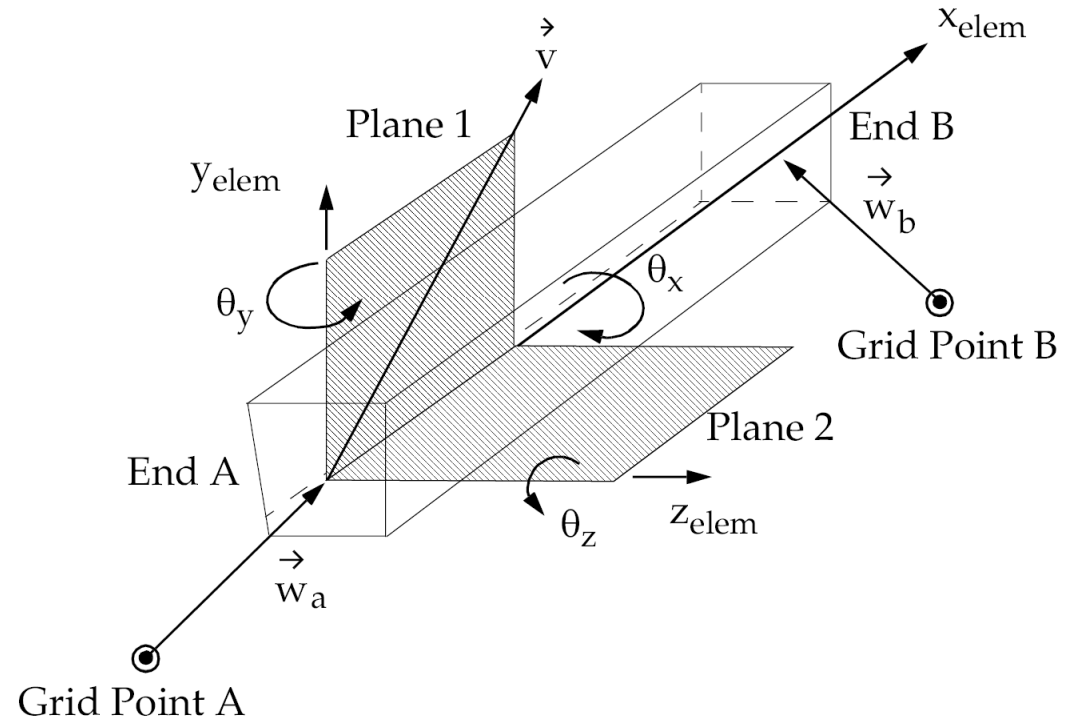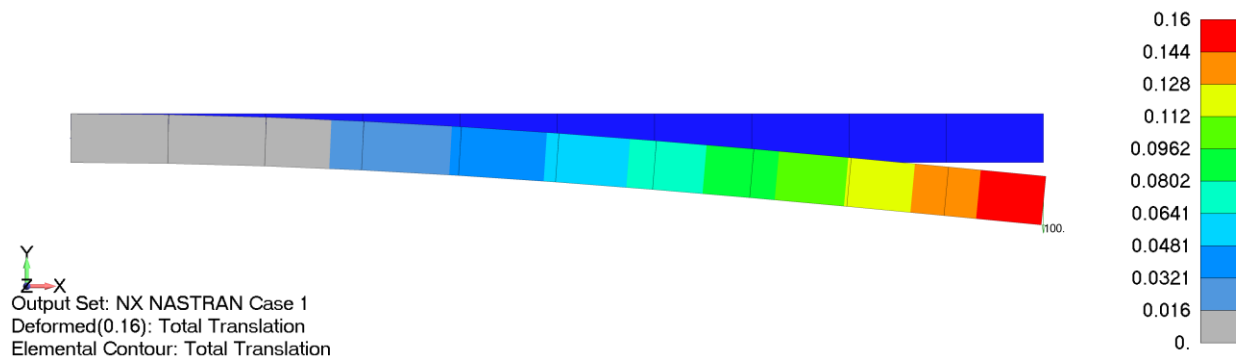
1.2 BEAM MESH

Equations for beam elements are developed from classical beam theory (e.g. Timoshenko Beam Theory). Being built on classical beam theory means a simulation engineer can perform a hand calculation and validate the model fairly easily. It also means that beams are non-isoparametric elements – that is, one doesn't need to consider mesh density in the traditional sense. One beam element will give you accurate stress and deflection results between two nodes in a static analysis. Of course, additional elements can be added if more resolution is desired in the deflection. The image below shows deflection contoured over a simple cantilever beam model. The maximum deflection is the same between the 1 element and 10 element model, but the 10 element model shows the bending contour and intermediate stresses more accurately.
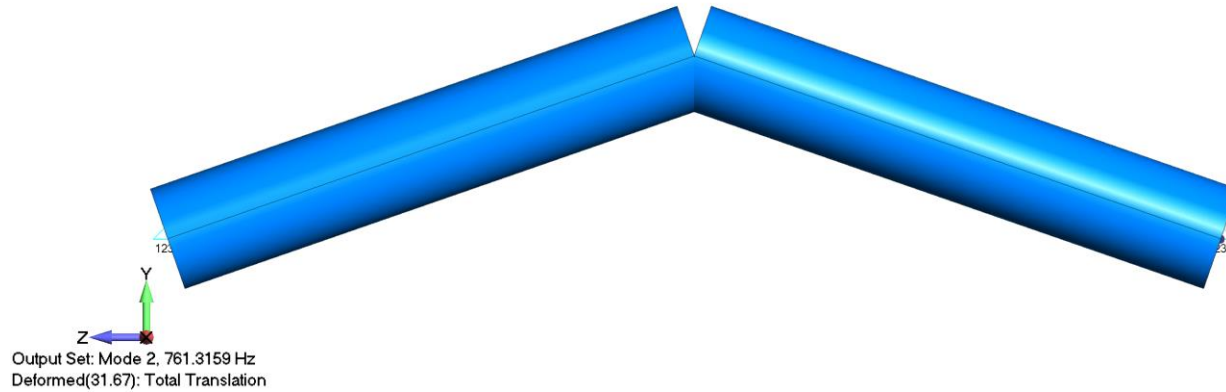
1-element beam:

Y
Z→X
Output Set: NX NASTRAN Case 1
Deformed(0.16): Total Translation
Elemental Contour: Total Translation

10-element beam

Y
Z→X
Output Set: NX NASTRAN Case 1
Deformed(0.16): Total Translation
Elemental Contour: Total Translation

Another situation where more elements should be considered when using beam elements is in vibration analysis. One element may not be enough to capture all modes of vibration of a beam. Let's consider a simply supported beam .

With two elements there are a total of 4 modes, with very little definition in the actual shape of the mode:

Output Set: Mode 2, 761.3159 Hz
Deformed(31.67): Total Translation

If we jump up to ten elements, there is a lot more resolution in the mode shape and we can see the higher order modes which didn't appear in the results for the two element beam model:

Output Set: Mode 7, 6422.453 Hz
Deformed(31.67): Total Translation
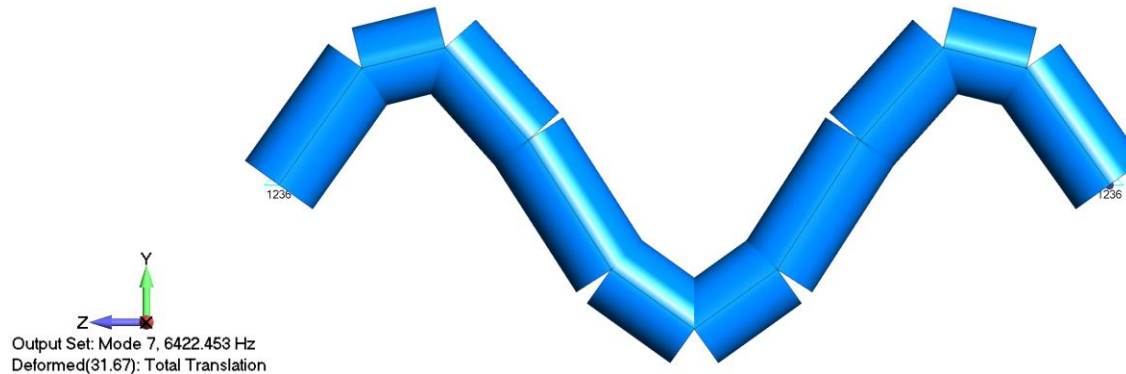
Back to stresses-- what about the Pt1 to Pt4 stresses? Beam stresses are calculated at the stress recovery points defined in the beam cross section definition dialogue. If stress values at different points are desired these can be changed in the beam property definition editor.

If the stresses at a stress recovery point are desired it can be plotted by selecting the combined stress at the selected beam end and stress recovery point. The combined stress includes the bending moment in PL1 and PL2, axial and shear stress in PL1 and PL2. It does not include the torsional stress.
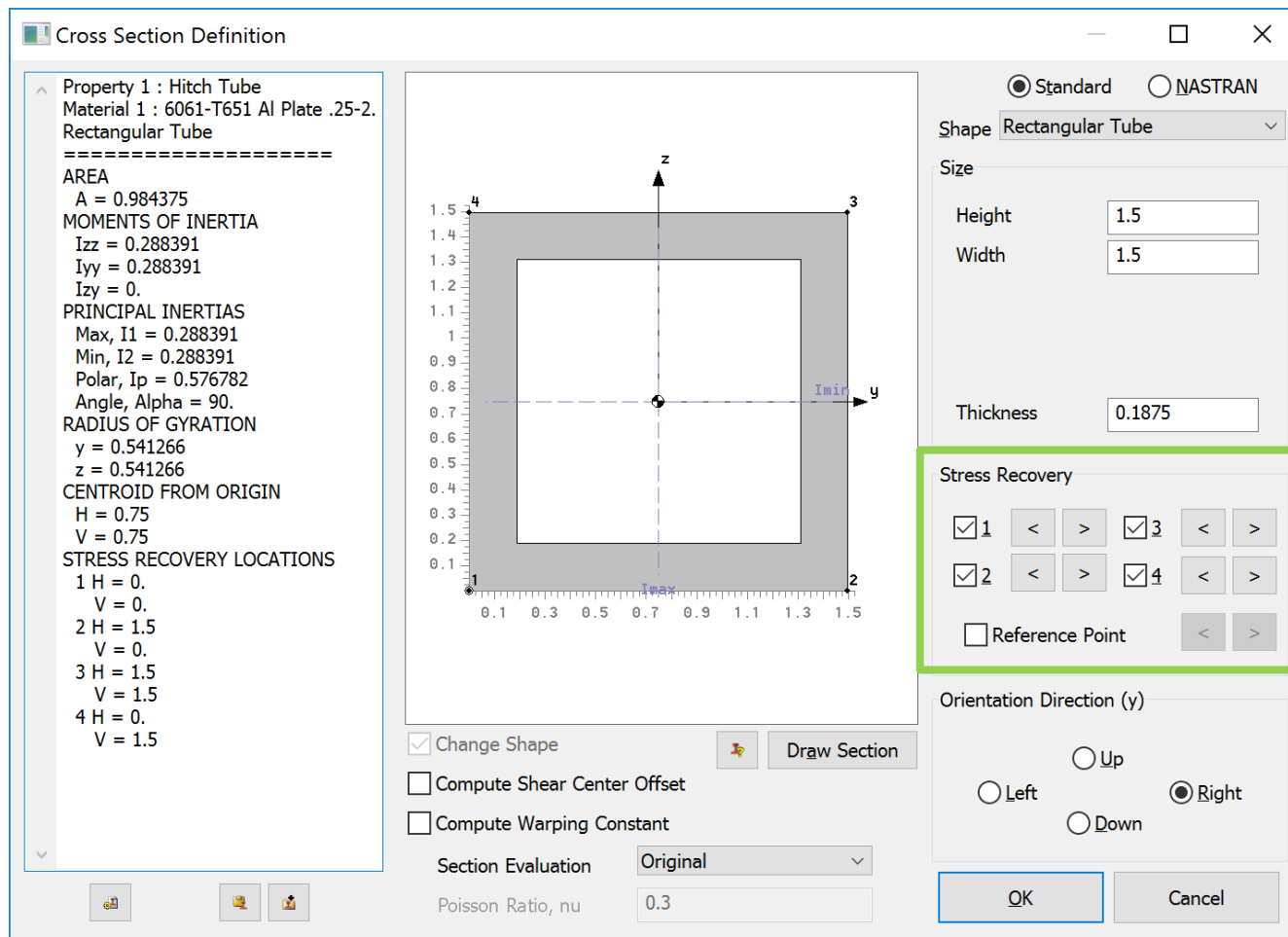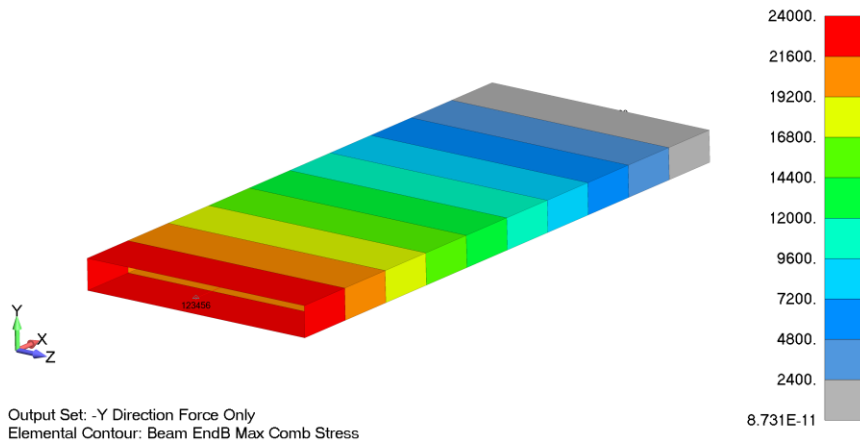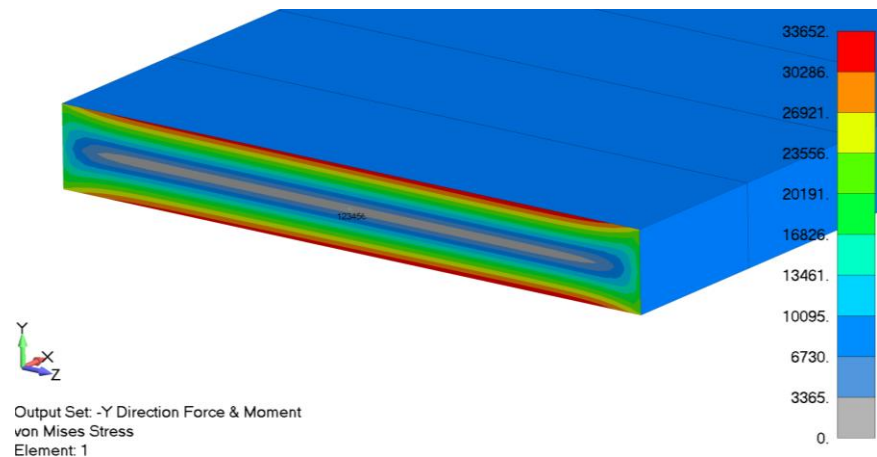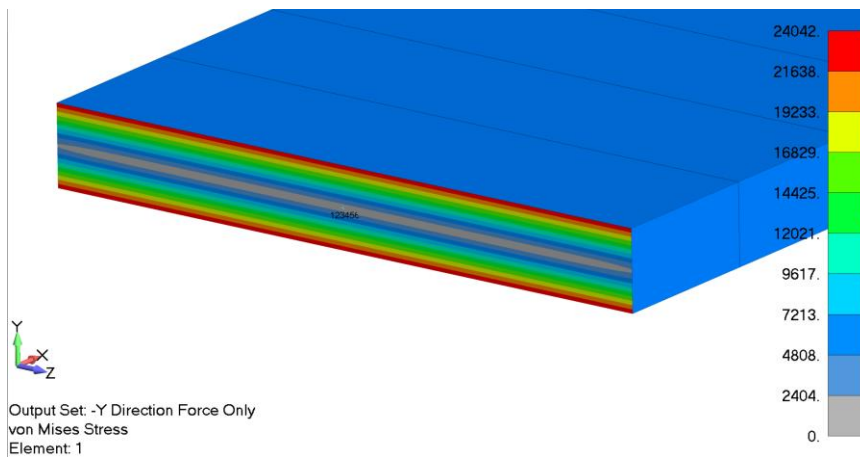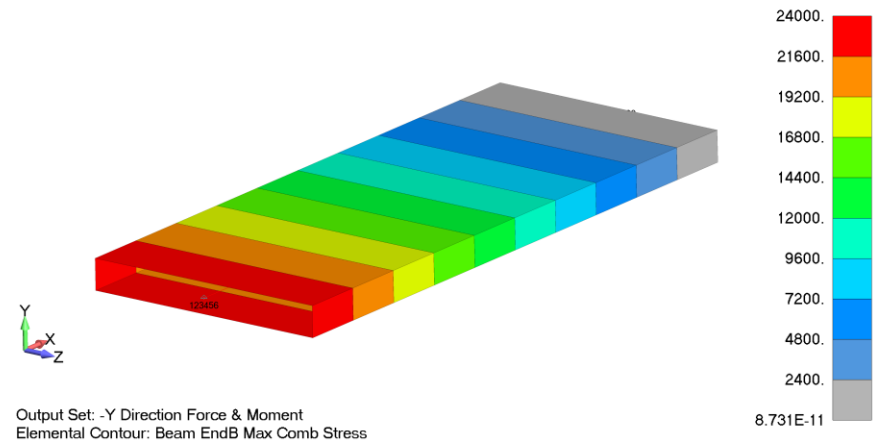
By now you may have noticed that there is no von Mises Stress output vector for beam elements. A lot of the time engineers are used to looking at von Mises stress in their analysis—so what stress result should you look at for guidance in your design? We can take a look at our simple cantilever beam model for answers again!

If there is minimal torsional loads the Beam End Max Combined Stress is equal to the von Mises stress, but if there are torsional forces you could be severely under-predicting your stresses.
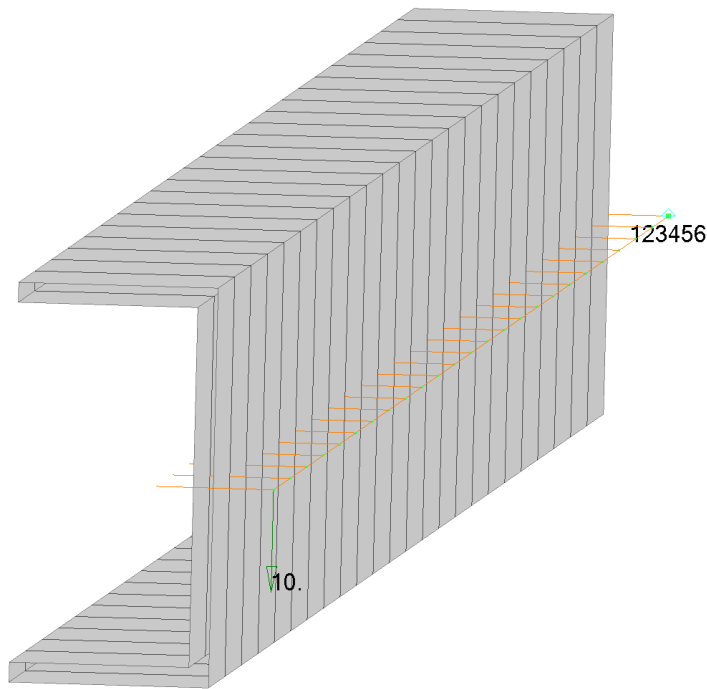
-Y Force Only	-Y Force, and Moment about X-axis

Output Set: -Y Direction Force Only
Elemental Contour: Beam EndB Max Comb Stress

Output Set: -Y Direction Force & Moment
Elemental Contour: Beam EndB Max Comb Stress

Output Set: -Y Direction Force Only
von Mises Stress
Element: 1

Output Set: -Y Direction Force & Moment
von Mises Stress
Element: 1

Sometimes the torsion in beam elements can affect your analysis in unexpected ways. One case to be careful of is thin walled open section beams. In these beams the shear center and centroid are not coincident. If a downward force is applied to the centroid, it would cause torsion in the beam (and we know from the previous discussion that beam elements do not like torsion. To avoid this, Femap applies a shear center offset to the element. This is important-- the load is always applied through the shear center, regardless of what this offset is.

Note: To achieve this offset, numerically a MPC is added to the beam to offset it from the connected nodes. This could cause issues in nonlinear analysis so a shear center offset should be avoided for nonlinear work.

With "Thickness / Cross Section" activated one can see that the beam is offset and the load is applied through the shear center

Shear Center offset boxed in green

1.3   LIMITATIONS

It should be apparent by now that there beams excel in some areas and are limited in others. Some things to keep in mind when considering using beams:

- Structures that are dominated by torsional stresses may be better represented with plate elements
- In non-linear analysis, adding a neutral axis offset may cause unpredictable results.

1.4   APPLICABILITY

Beam elements can be very useful for the following use cases:

- Long, slender structural bodies

- Bolted Connections

- First-pass analysis to determine beam sizing or geometry

- Structures with very low torsion

2. BOLTED CONNECTIONS

Now it's time to put some beams to work. One of the most common uses for beam elements is modeling bolted connections. One approach is to add MPCs to the holes and use a beam element to connect the MPCs.

Our example model is made up of plate elements and is ready for some bolted connections:
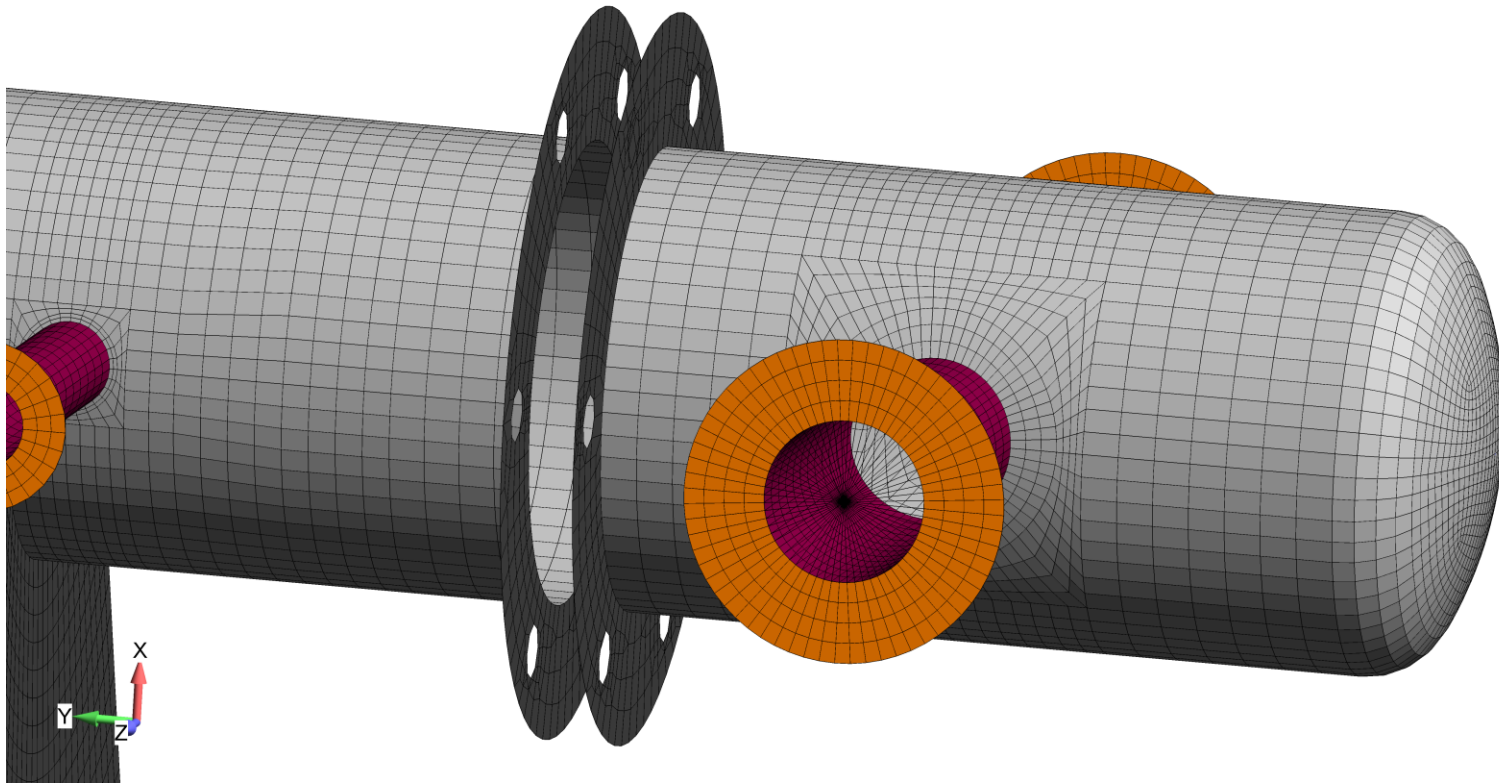
3.1 FASTENER API

Let's use an API to help us out with our bolted connection. This API is useful for connecting two holes in plate elements, and requires a bolt material to be created prior to starting the API. The Hole To Hole Fastener API is accessed in the Custom and User Tools menu, Custom Tools > Meshing > Hole To Hole Fastener.

This brings up a dialog box where one can simply select the curves on the first hole, then select the curves on the second hole, and Femap will create your MPCs and a beam element connecting the two.
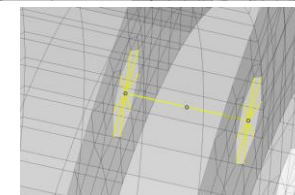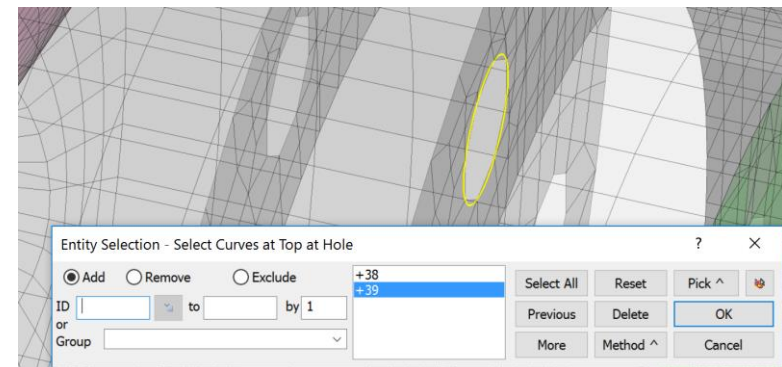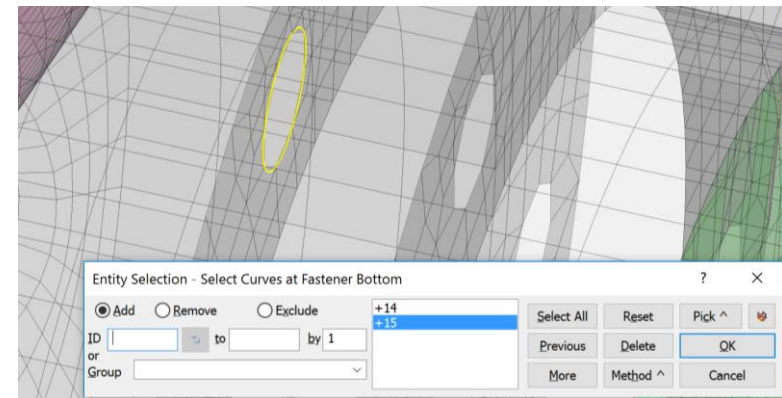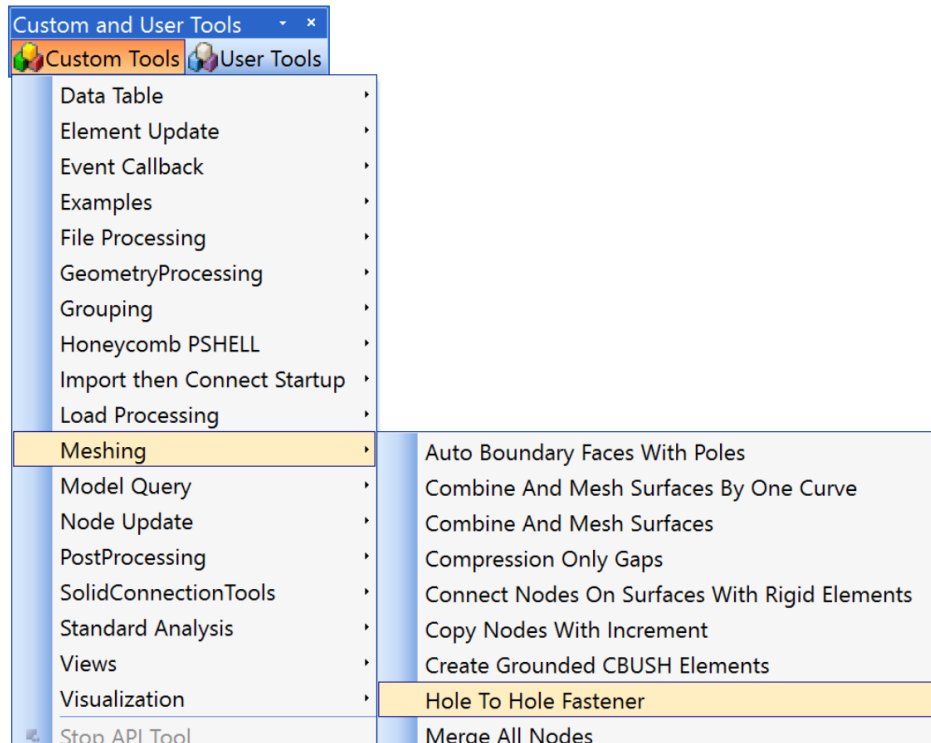
With the bolts created we can turn on Thickness and make sure everything looks good:
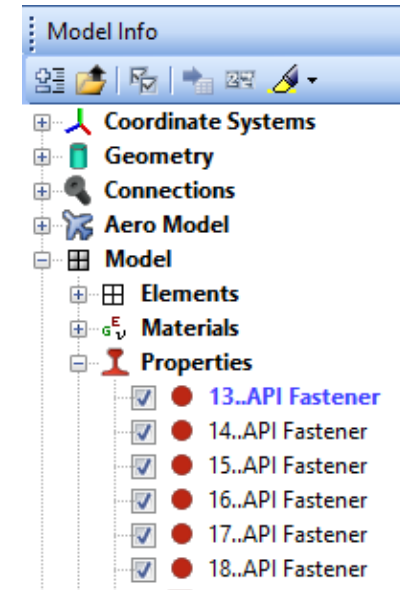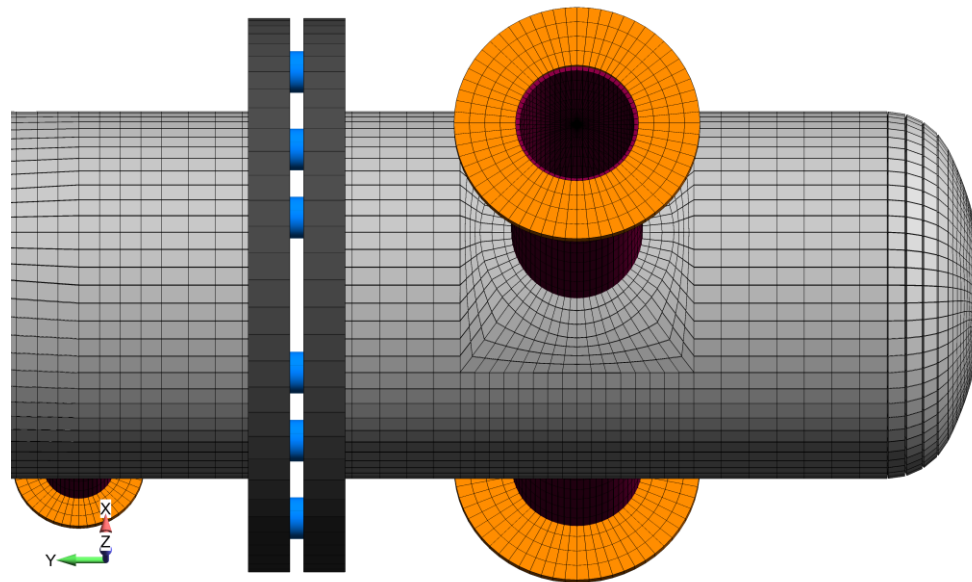

You may have notices we never set a bolt size when creating these bolts. This API measures the diameter of the curves and creates a beam element the same size. If different size is desired, the radius can be changed in the property definition.

The boundary conditions have already been applied, so this is ready to analyze!
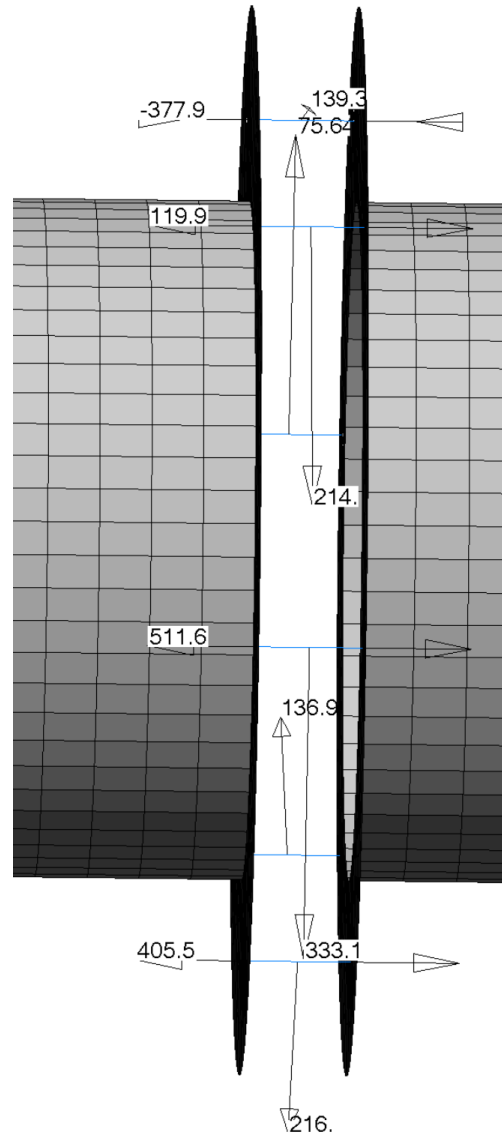
3.2   CONTOUR ARROWS

One way to query forces on the bolts is to use contour arrows. This is selected in the post-processing toolbox.

The setup is shown below on the left. Start at the top and move your way down the options.
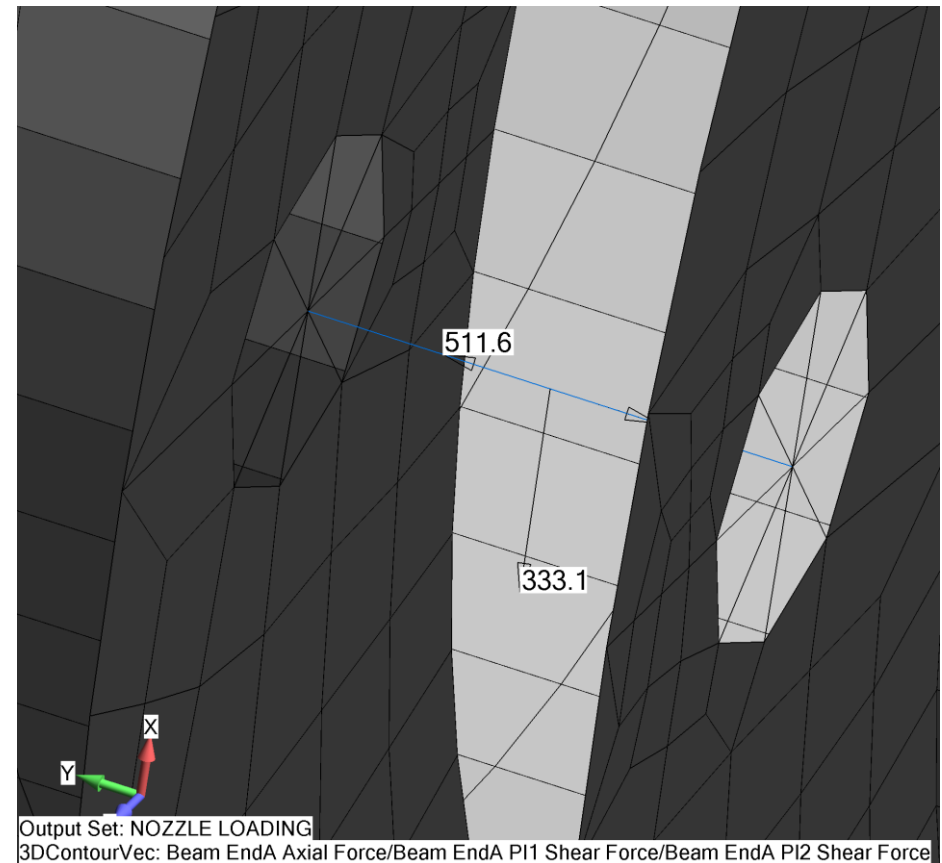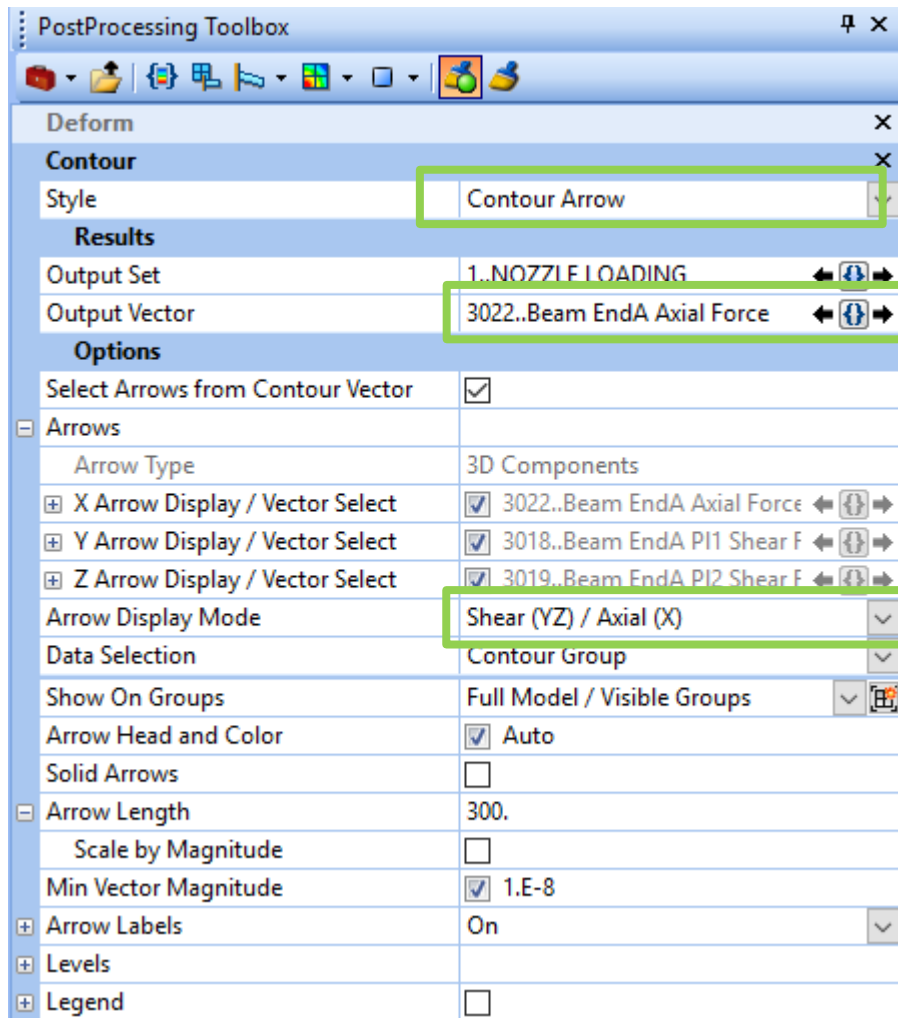
While this is convenient to visualize forces on one or two bolts it can get overwhelming as the number of bolts increases:

3.3   EXPORTING SHEAR AND AXIAL FORCE TO CSV

Most analysts are going to want these bolt forces in a table format. Luckily, Femap provides a straightforward way to drop these results into a .csv file. Prior to the below steps, ensure that the data you want to export to a table is contoured over the model.

- Turn on the Data Table (Tools > Data Table).

- Select List > Output > Contoured Results to Data Table
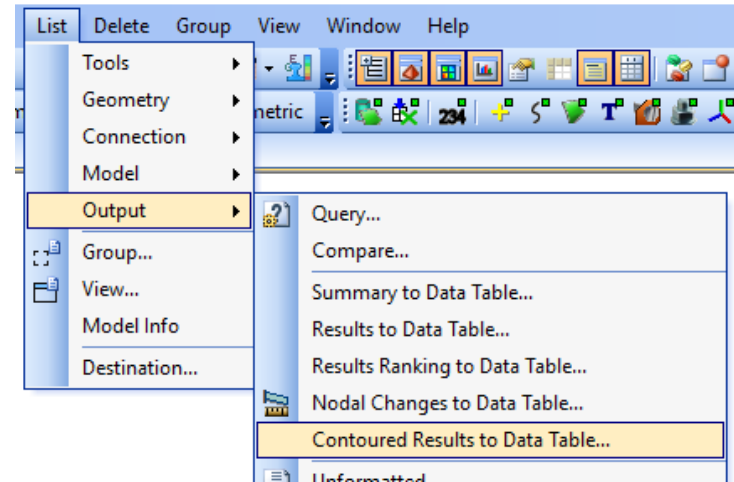
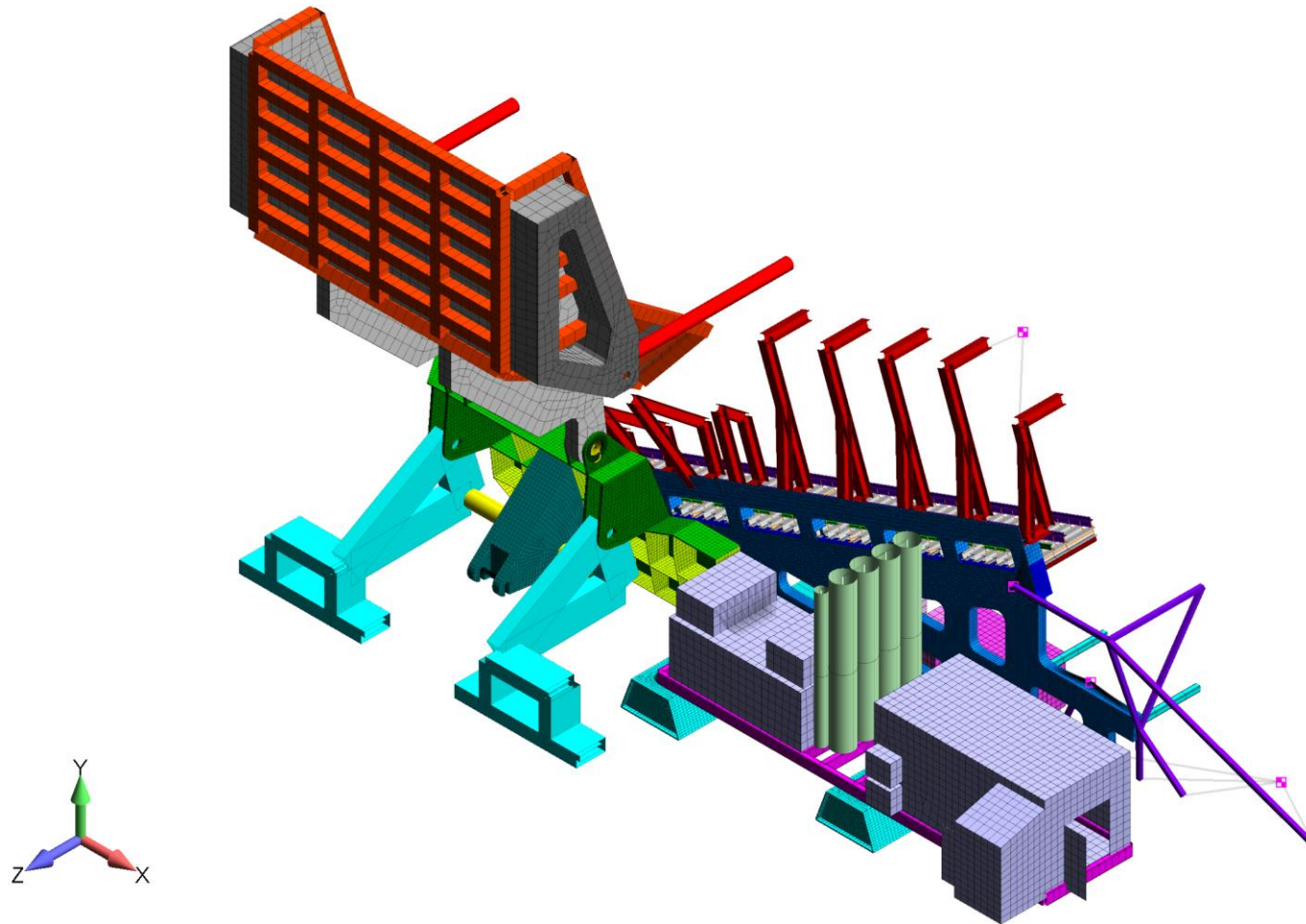- Click the Save button, and save your results to a .csv file

4. LINKAGES WITH BEAMS

Another common use case for beam elements is linkages. Beam elements allow you to release any of the 6 degrees of freedom at the ends, which will allow the beam end to move as desired.

Let's start off with the farm equipment model shown below:

Let's say there is a new intern in the office, and they built this model, analyzed it, and sent it your way. The customer is really interested in the stresses in the yellow beams shown below:
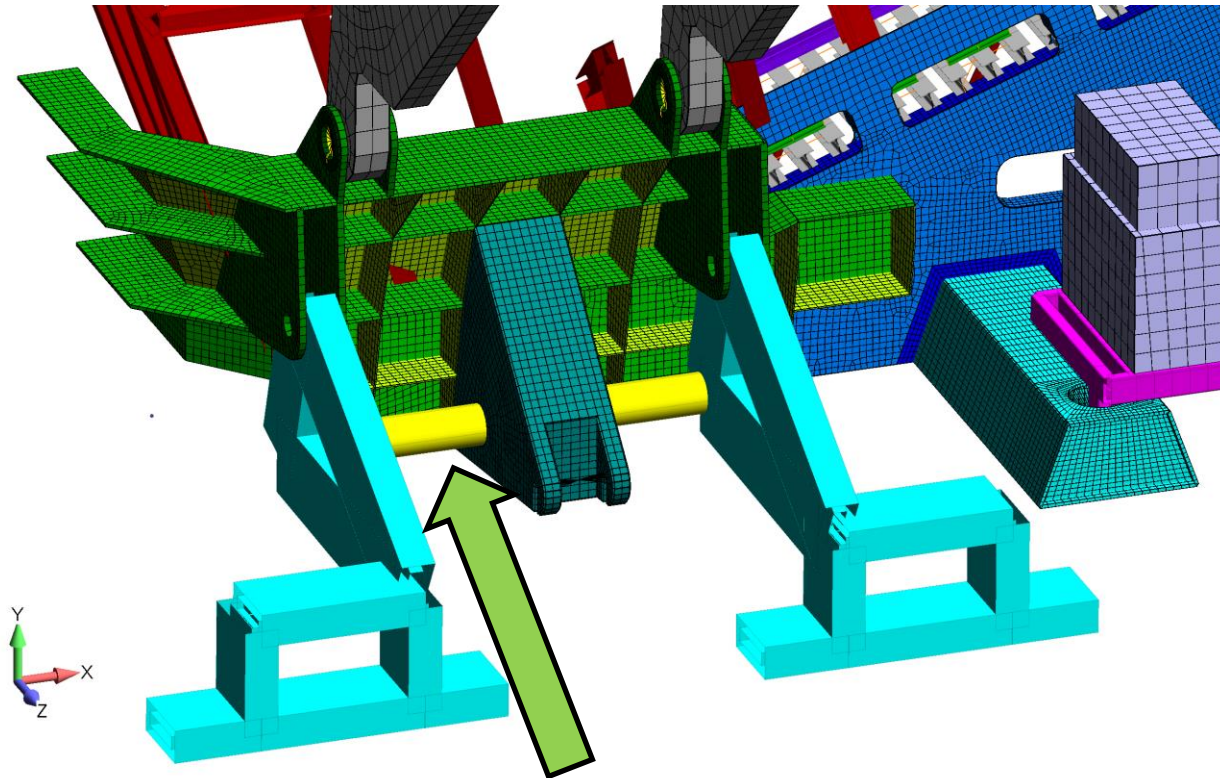
4.1   FREEBODY DIAGRAM ON BEAMS

First, let's take a look at the free body diagram at one end of the yellow beam:

Output Set: NX NASTRAN STATIC ANALYSIS SET - 6dof
Freebody: Yellow Beam Loads
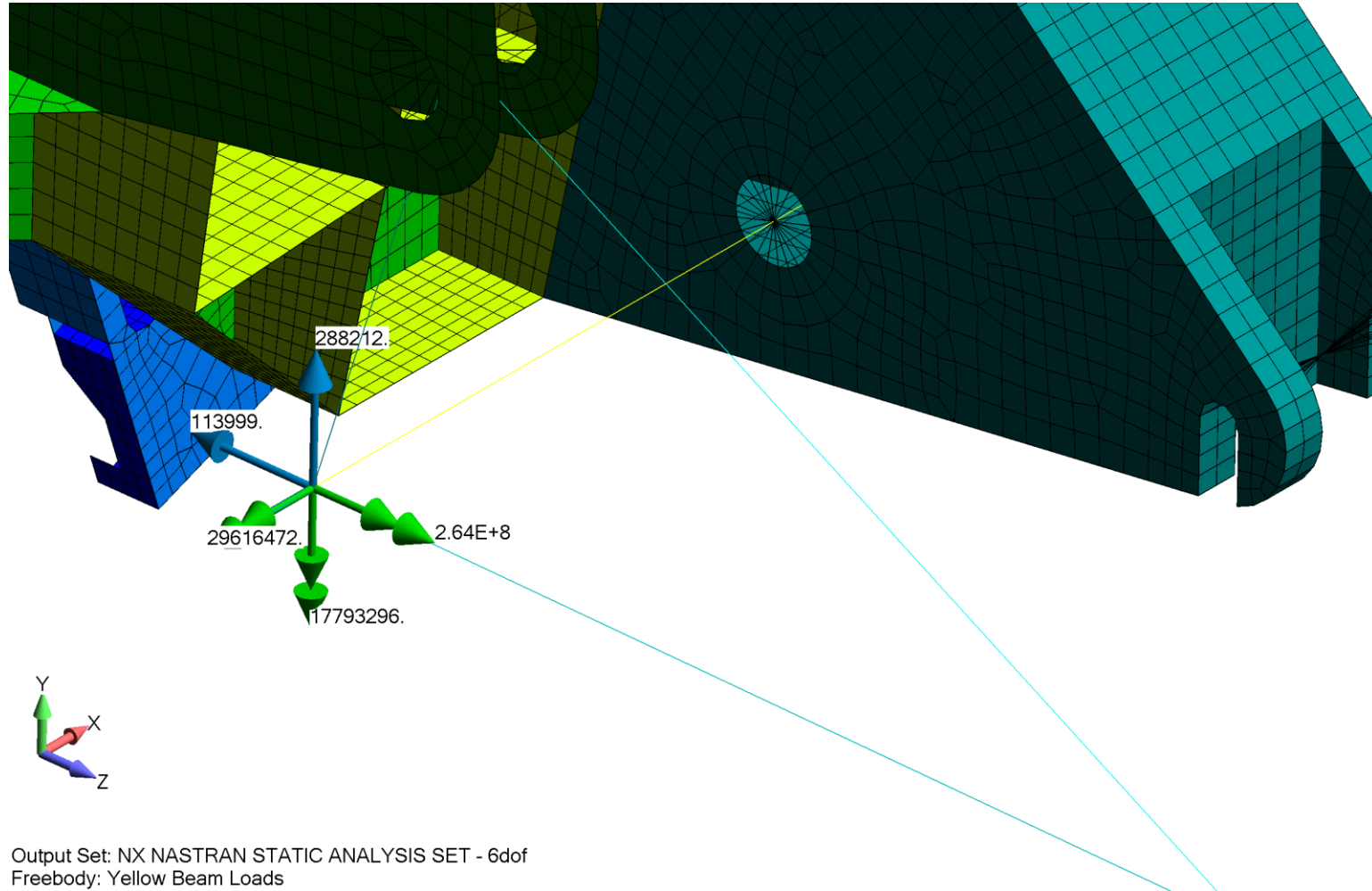
The forces seem reasonable, but notice the large moment about the X-axis. This beam is supposed to rotate freely, so you know that there should be no moment about the rotational axis.

4.2   BEAM ENDS, END RELEASE

In order to let the beam rotate freely along the x-axis, the ends need to be released in the RX direction. Note that the beam releases are with respect to the local beam coordinate system, this beam just happens to be lined up with the global coordinate.

Open up Modify > Update Element > Beam/Bar Releases
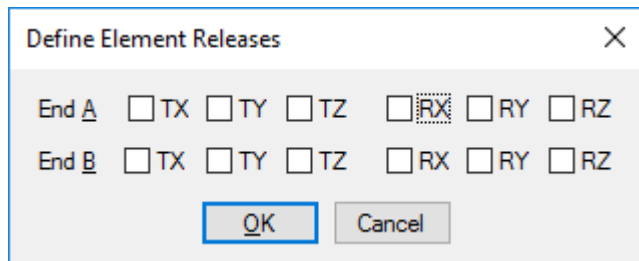
As discussed before, there are 6 dof for each beam end which can be released. For example, selecting the End A, TX box will allow End A of the beam to translate in the beam's X (axial) direction.

You can display beam End A and End B by selecting View > Options, and select Element – Directions, then "Show Direction" in the upper right corner.
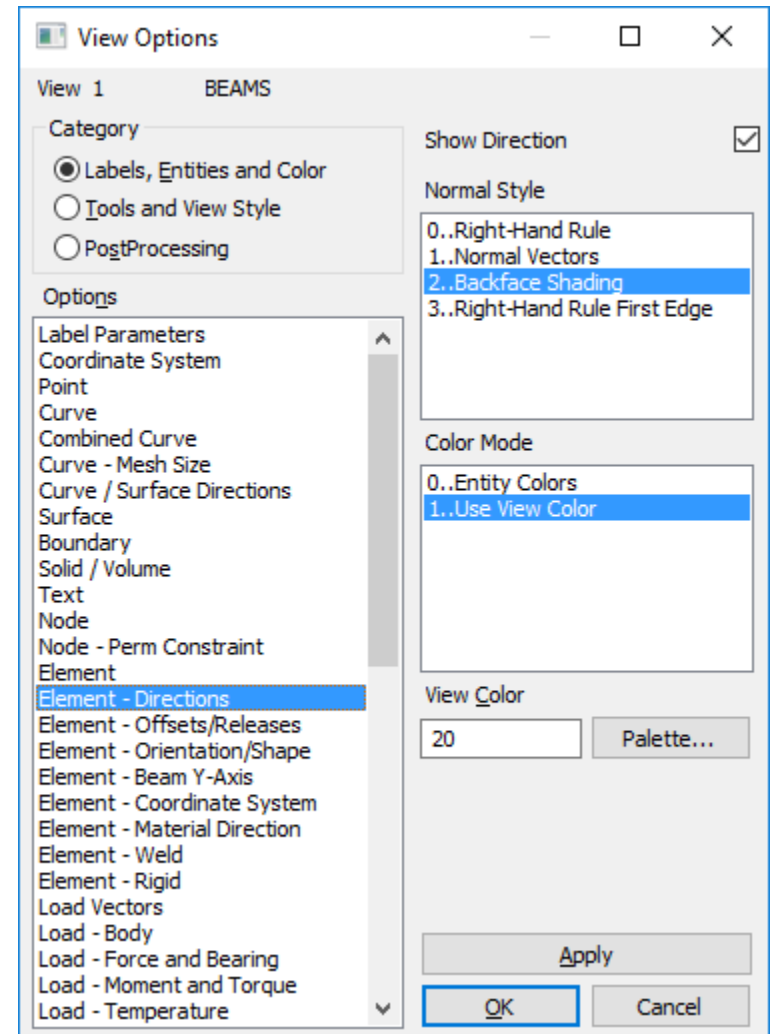
With element directions turned on you can see an arrow head on each element. The arrow head is located on End B, and pointing from beam End A to End B.

We want the outer end of the beam to be able to rotate (the end on the left side of the image), so we will release beam End A
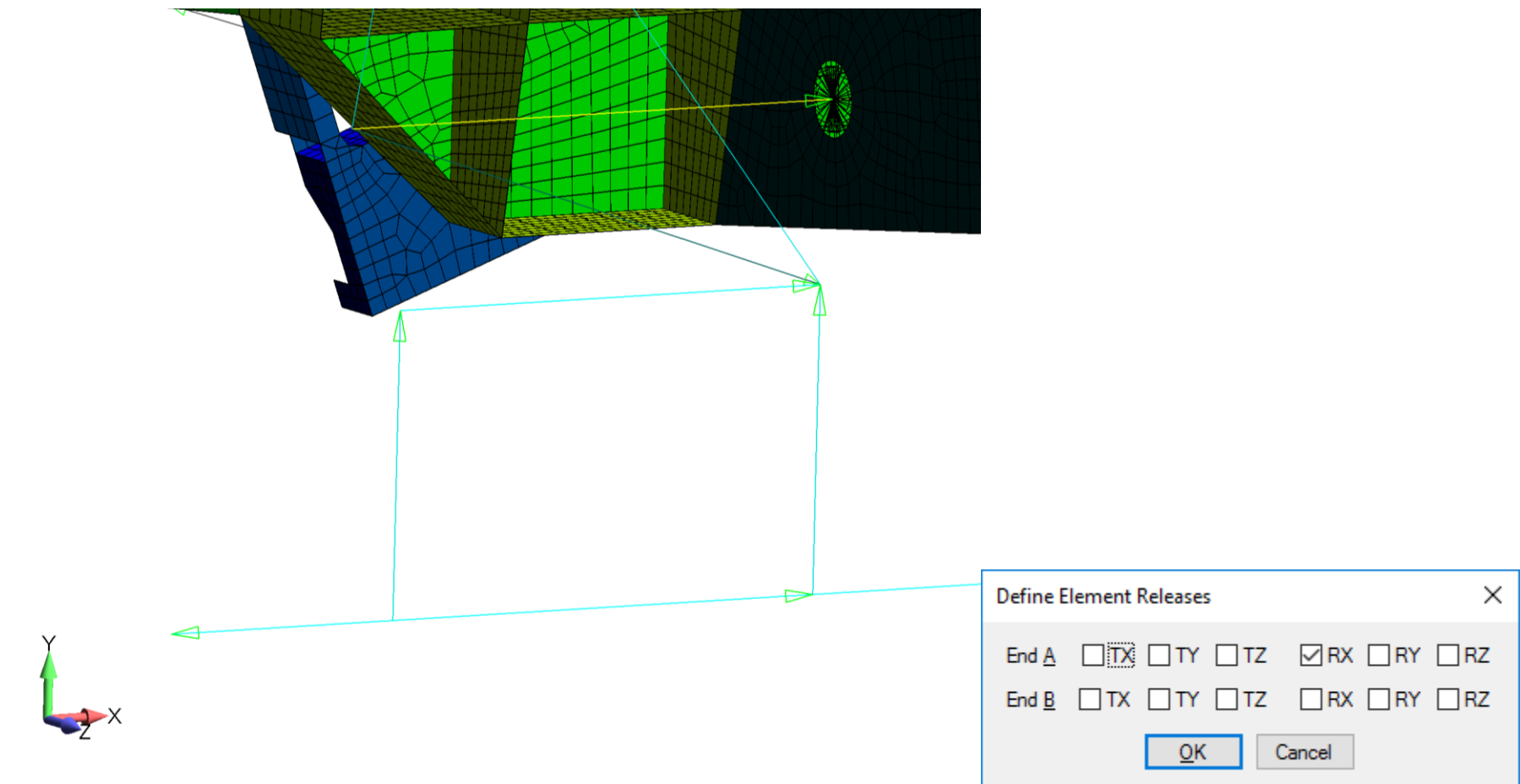
On the other side the beam is flipped around, with the arrow head on the right side of the beam in the image.
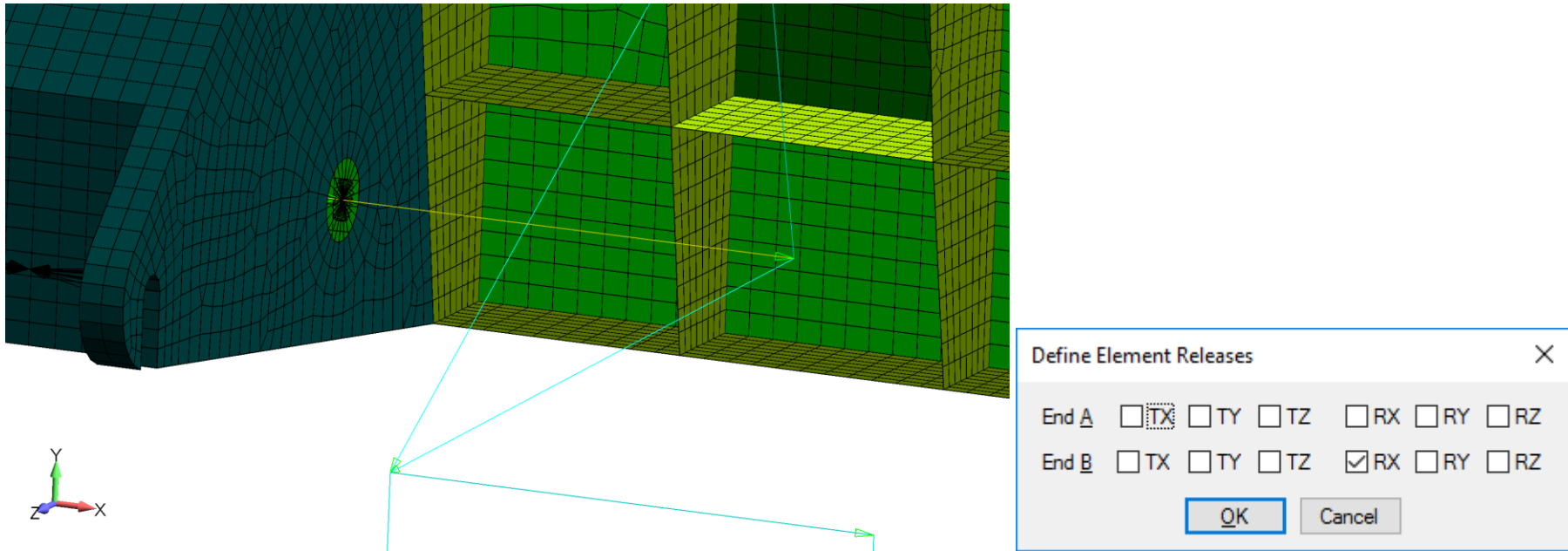
Here we will release End B in RX to allow it to rotate on the end on the right side of the image.

With the ends properly released we can go back and take a look at the free body diagram to ensure the loads make sense.

Output Set: NX NASTRAN STATIC ANALYSIS SET - RX Released
Freebody: Yellow Beam Loads
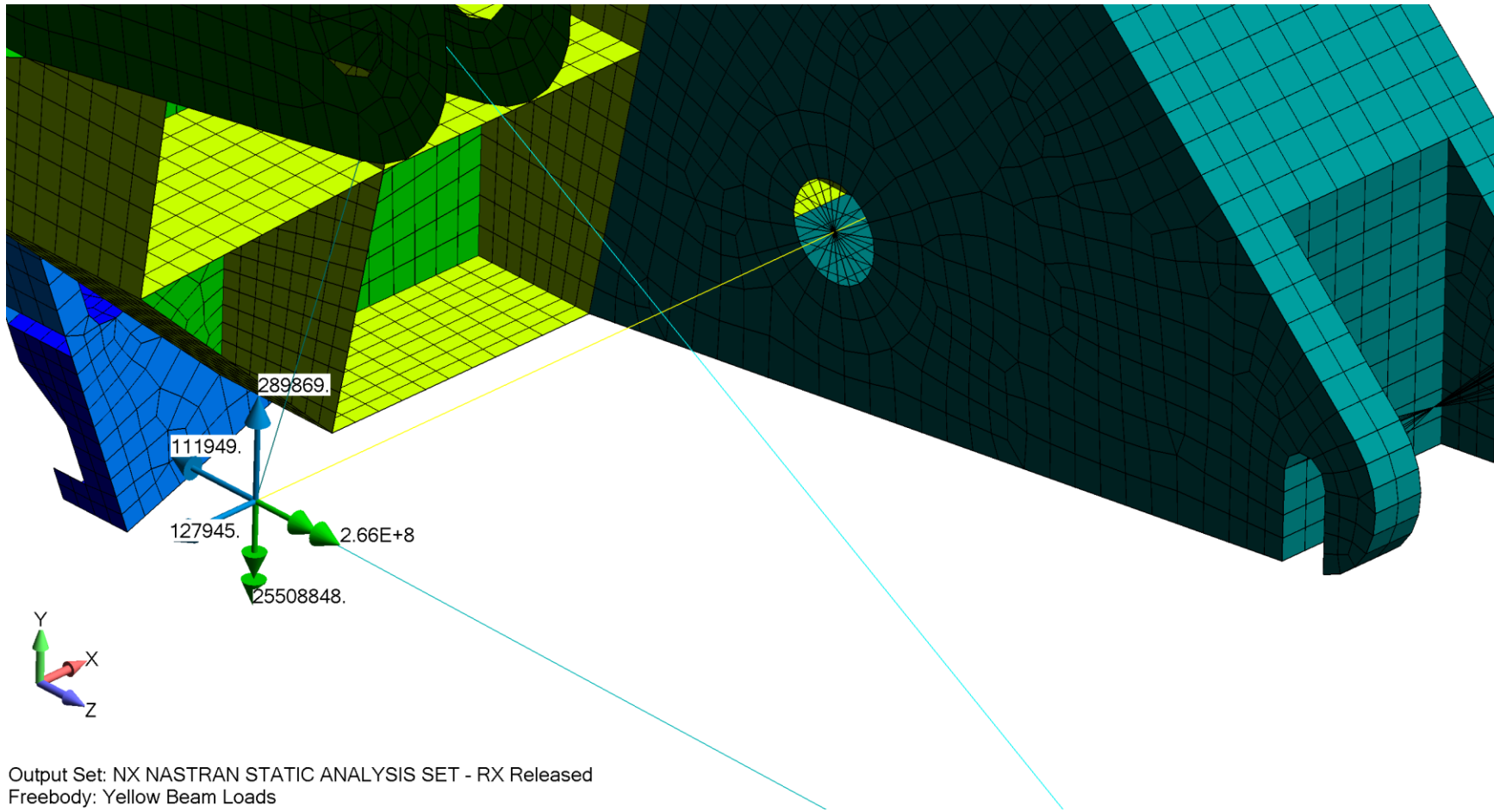
With the beam ends released there is no moment about the x-axis of the beam, so it looks like the problem has been corrected.

4.3   ADVANCED POST FOR FULL FIELD STRESSES
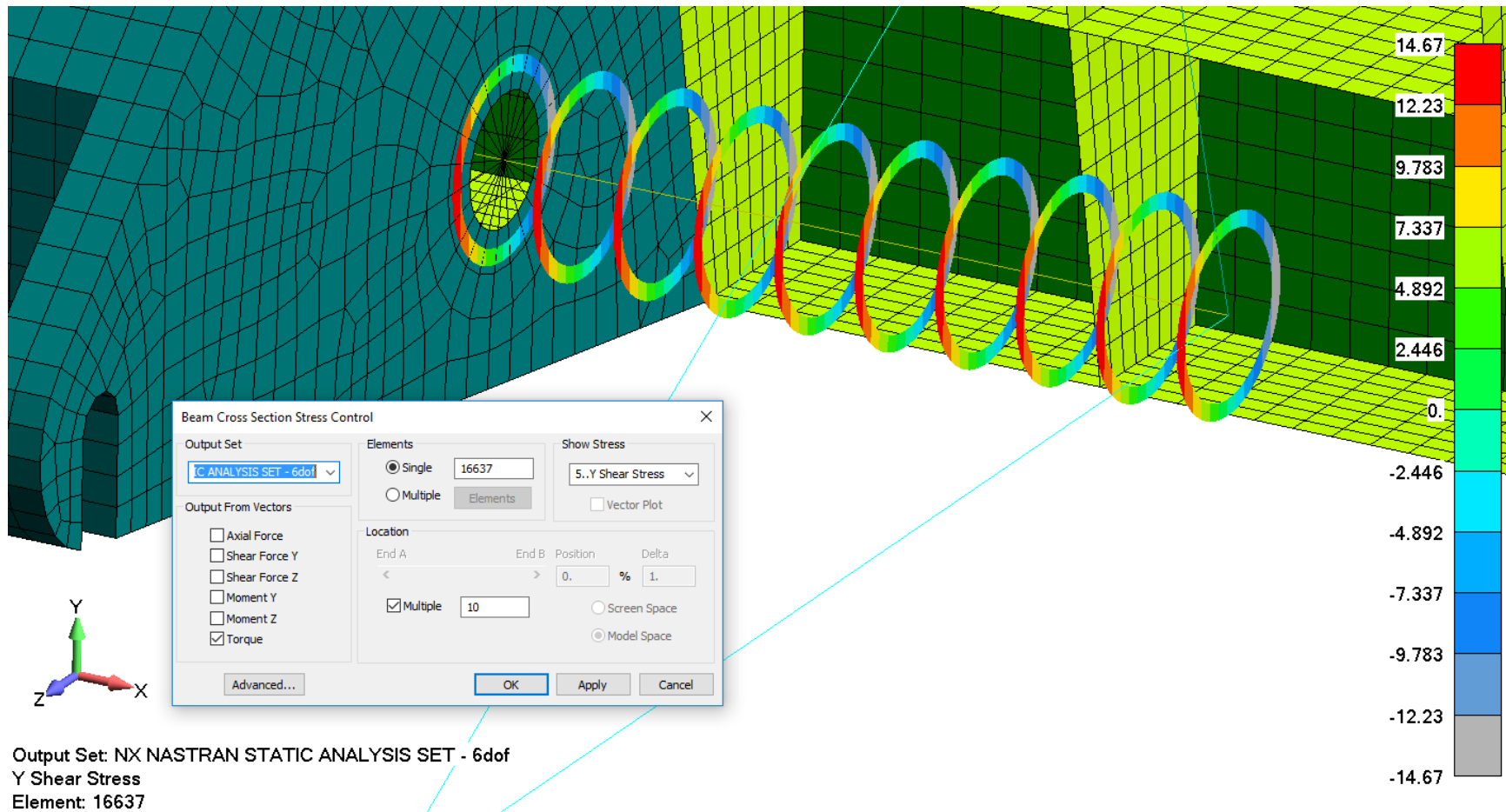
For further investigation we can go back to the Advanced Post tool.

The image below shows the Y Direction Shear Stress due to torque when the beam end is not released

The image below shows the Y Direction Shear Stress due to torque when the beam end is released, and as expected there is zero stress due to torque.

Output Set: NX NASTRAN STATIC ANALYSIS SET - RX Released
Y Shear Stress
Element: 16637
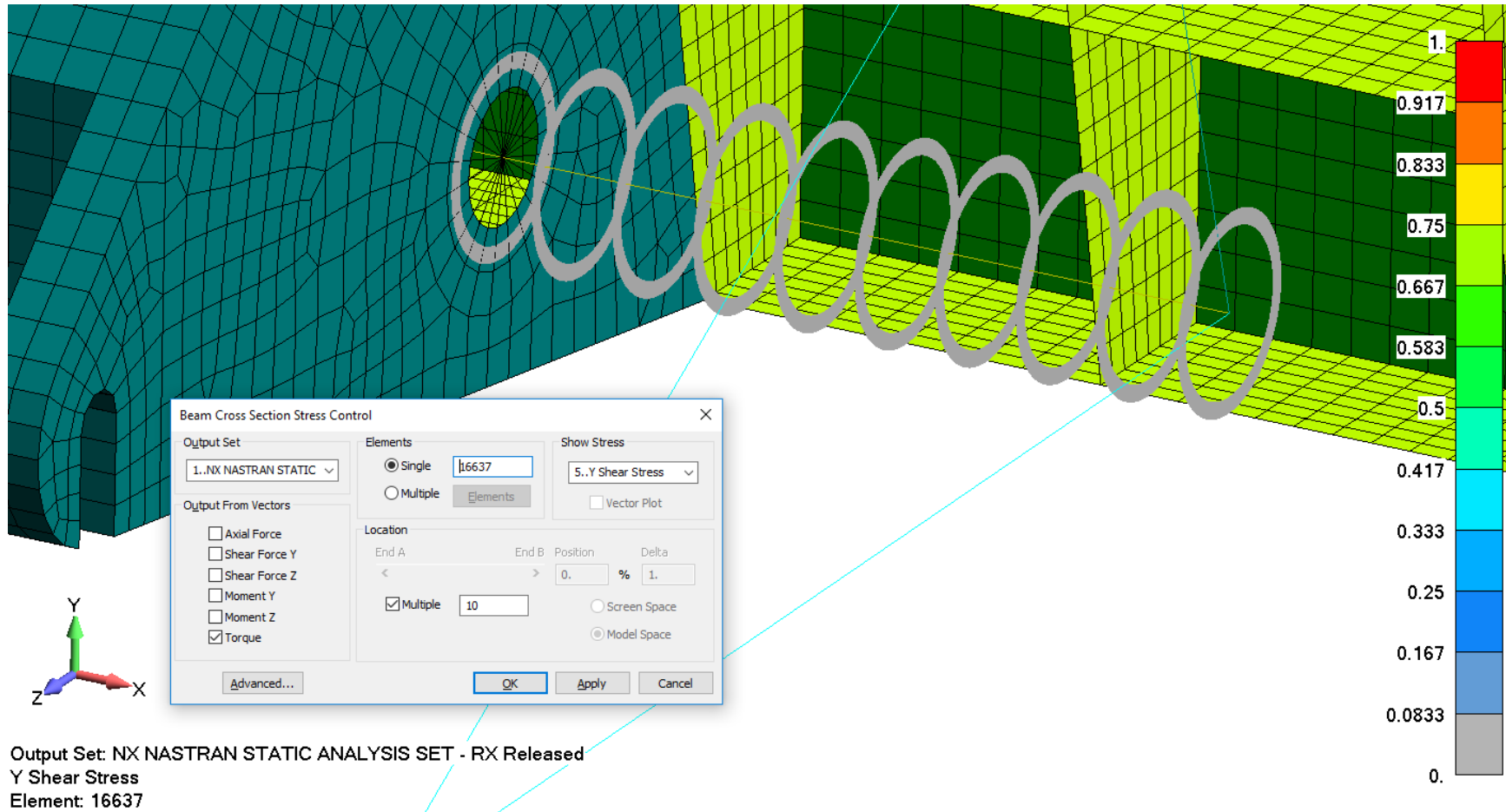
5. SUMMARY

The takeaway here is to keep track of what you are asking Femap to do. If a load is applied directly to a beam element, one must keep in mind that there will be no torsional bending because the load is always applied through the shear center. If another load condition is required, one can manually apply a load to the centroid via rigid element, but any deflections will not be shown visually in the post processing. If an open section beam is to be loaded away from the shear center, a more transparent model can be created by using plate elements instead of beam elements.

6. SEMINAR Q & A

Q: Why not release both End A and End B for a beam that sits in bearings on both ends?

While this does make intuitive sense, it doesn't work out numerically in the FEA solver. If both ends are released there is no stiffness in the local RX direction, and the beam will be under-constrained.

Q: Is there a way to make the API use the same bolt property each time instead of creating a new property for each beam?

Yes! There are many bolt APIs like this with different options. See www.appliedcax.com for more APIs. Check out "Automatic Spider and Beam Creator" for an example of a bolt API with more options.

Q: What is the difference between Max Combined Stress and von Mises Stress?

Max combined stress is calculated by combining all beam stresses except torsion. Von Mises includes all of those stress components and adds torsion. That is why the example with a force only showed that Max Combined stress matched up with the Von Mises stress, while when a torque was applied to the beam, Von Mises stress was much higher than Max Combined stress.