# NSTX-U

# TF Outer Leg Clamp Assembly

**NSTXU-CALC-132-12**

**Rev 0**

**March 2012**

**Prepared By:**


Peter Rogoff Digitally signed by Peter Rogoff
DN: cn=Peter Rogoff, o=MED, ou=PPPL,
email=progoff@pppl.gov, c=US
Date: 2012.03.13 12:22:02 -04'00'

Peter Rogoff, PPPL Mechanical Engineering


**Reviewed By:**


Irving Zatz Digitally signed by Irving Zatz
DN: cn=Irving Zatz, o=PPPL, ou,
email=zatz@pppl.gov, c=US
Date: 2012.03.13 12:39:50 -04'00'

Irv Zatz, PPPL Mechanical Engineering


**Reviewed By:**


Mark Smith Digitally signed by Mark Smith
DN: cn=Mark Smith, o=PPPL, ou,
email=msmith@pppl.gov, c=US
Date: 2014.11.14 14:56:40 -05'00'

Mark Smith, PPPL Mechanical Project Engineer

# PPPL Calculation Form

Calculation # **NSTXU-132-12-00** Revision #**00** WP # **1677**
(ENG-032)

## *Purpose of Calculation: (Define why the calculation is being performed.)*

The NSTX principal magnetic coil outer legs are restrained by the specially designed clamps which are in turn connected to form a solid ring like support structure. However, this assembly is not sufficient to fully restrain the rotational motions which is created by the PF forces. Therefore, connections between the clamp assemblies to the vacuum vessel (VV) are necessary to effectively restrain deformations which are created by these forces. This was accomplished by a two strut assembly at each coil outer leg to the existing VV support pads. These assemblies are designed in such a way that struts will be subjected to constant axial forces only in tension or compression. There are no moments carried through the strut ends. Aurora designed end inserts, which are connected to the strut end pins, are used in order to permit universally complete free rotation at each end.
This analysis is performed in order to check and verify the design integrity by selecting and specifying the necessary pins and support brackets. The actual assembly geometry and FEA simulations are presented in the subsequent calculation section.

## *References (List any source of design information including computer program titles and revision levels.)*

The complete assembly FEA simulations are executed via FEA 2010.1r.2"Nastran code".
Required analysis forces were extracted from various ANSYS code Global Models simulations.
Results are checked using classical stress equations and are listed in the calculation sections as they are used.

## *Assumptions (Identify all assumptions made as part of this calculation.)*

For this analysis the following assumptions are important: Lap joints of the main rings are holding (this is a subject of the NSTXU-132-11-00), surfaces between the actual coil conductor assemblies are not slipping, and the clamp bracket bolts (four for each bracket) are properly installed to provide the best load path for the forces generated in this complete assembly. If any of these locations lose contact, slip or change orientation, the contacts between the "Aurora inserts" and the end pins, on both ends of the struts, will change modifying the orientation of the applied forces used in the FEA simulations. In this case, the struts will retain axial forces carried as before, but the supporting pins will be subjected to different force and moments distributions.

## *Calculation (Calculation is either documented here or attached*

All the existing calculations are provided by the following Power Point files:

## 1) Ring_Coil_Joint2.ppt

This presentation describes the original NASTRAN simulation of the TF coil supports by the "Coil-Clamp-Ring" assembly. This depicts a section of the actual joint which is restrained by ANSYS global models calculated displacements and forces represented as boundary conditions. Therefore, NASTRAN capabilities to simulate 30-degrees cyclic symmetry through the MPCs, SPCDs (enforced displacements) and concentrated forces at regular intervals are used (see the model pictures).
So, using this model, it demonstrated the necessity to redesign the link/struts and general clevis supports for the pins which transmit forces between the VV and the Coil-Clamp-Ring assembly. In this case, the links carry the axial force, only, and moments are not created.
On the other hand, because of the segment boundary conditions and the TF and PF forces which are applied in the coil simulation (see model pictures), the ring structure can be evaluated effectively. Therefore, general ring strains and stresses are examined and are considered very conservative since the simulation doesn't include the additional welding materials. This is an important consideration since maximum strains and stresses are in the potential weld areas. So, the simulation is considered very conservative since all the developed stresses and strains are within the SS316 material allowable.

For SS316, Yield = 42,000. Psi.
Calculated Max Principal = 22,050.psi
Calculated Max Strain = .002 (for one anomalous nodal point at an artificially high stress concentration whose result was deemed exaggerated). The max strain for the rest of the structure ~ 0.0011)

Note: as of this writing, the design for the complete assembly was not fully completed. Additional analyses evaluation may be required when everything is finalized.

## 2) PinA-PinB-Beam_Data.ppt

This presentation fully describes the TF-Coil-Clamp-Pin assembly. This simulation models the 1.25 inch diameter pin which is supported by the upper and the lower brackets. The brackets are connected to the main assembly by eight bolts. The simulation was performed in steps by changing the geometry of the pin, as well as the brackets, to justify the final design using link/strut forces obtained from the various global models. The final brackets/pin configuration is depicted by the simulation in Case#3 (see the appropriate slide).
Applied forces simulations on the pin are: for the tensile link/strut F = 27,000. Lbs, and for the compressive link/strut F = -15,000. Lbs. These forces should be considered conservative.

## 3) New_TF-Clevis-Pin-9-28-2011PR.ppt

This presentation describes the possible link/strut force changing development as the function of the TF and PF application. The interesting finding through the simulation, was to show that if the TF and PF forces are of similar magnitude, the link/strut in the direction of the PF force supports no load (developed forces in this link/strut are close to zero). Only as the PF increases relative to the TF force, will this link/strut pick up the compressive loads.

## Conclusion *(Specify whether or not the purpose of the calculation was accomplished.)*

Based on the above described simulation it was determined to recommend the following final design parameters for the TF-Coil-Clamp-Pin assembly:

- Increase the pin diameter from 1.0 to 1.25 inches. Material, Inconel 718, Yield = 150,000. Psi.
- Increase the insert thickness from 0.1 to 0.2 inches. Inconel 718 with minimum Yield = 105.000. Psi.
- Keep the brackets 316 Stainless, Yield = 42,000. Psi.
- Update the geometry of the brackets as per Case#3 configuration.

Preparing Engineer's printed name, signature, and date

Peter Rogoff Peter Rogoff

Digitally signed by Peter Rogoff
DN: cn=Peter Rogoff, o=MED, ou=PPPL, email=progoff@pppl.gov, c=US
Date: 2012.03.13 12:23:02 -04'00'

**I have reviewed this calculation and, to my professional satisfaction, it is properly performed and correct.**

Checker's printed name, signature, and date

Irving Zatz Irving Zatz

Digitally signed by Irving Zatz
DN: cn=Irving Zatz, o=PPPL, ou, email=zatz@pppl.gov, c=US
Date: 2012.03.13 12:40:11 -04'00'

PinA-PinB-Beam_Data.ppt

P.R. 12/2011

## Explanation of the calculations presented with this study!

This presentation, is an attempt to explain the possible creation/variation of the reactive forces and stresses, in the VV to Coil clamp strut/link assembly as a function of the possible TF and PF load scenarios.

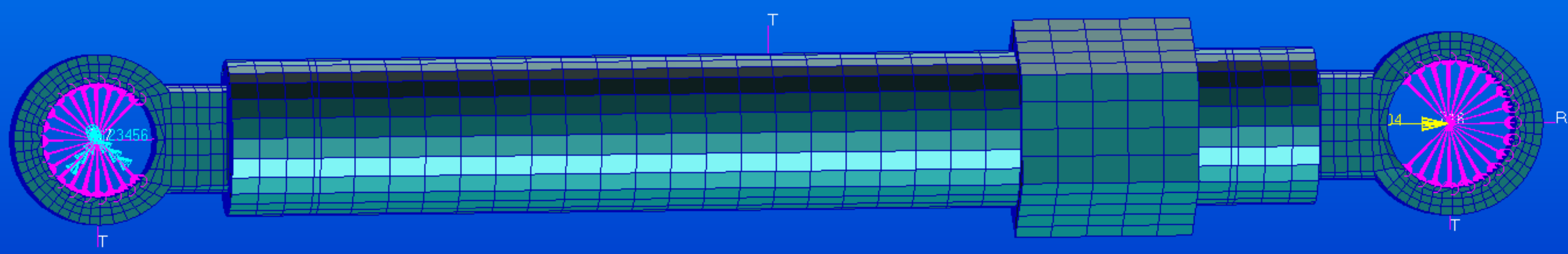
Because of the design shapes of the main linkages/struts and the assembly of the Aurora pivot assemblies, which connect to the supporting pins, the linkages produce different stiffness in tension and compression. As the structure rotation in the PF force direction increases, the link/strut in that plane goes in the increasing compressive mode. Therefore, this whole coil-clamp-ring assembly is affected as well as the reactive forces on the VV clevis.

The analyses procedure is as follows:

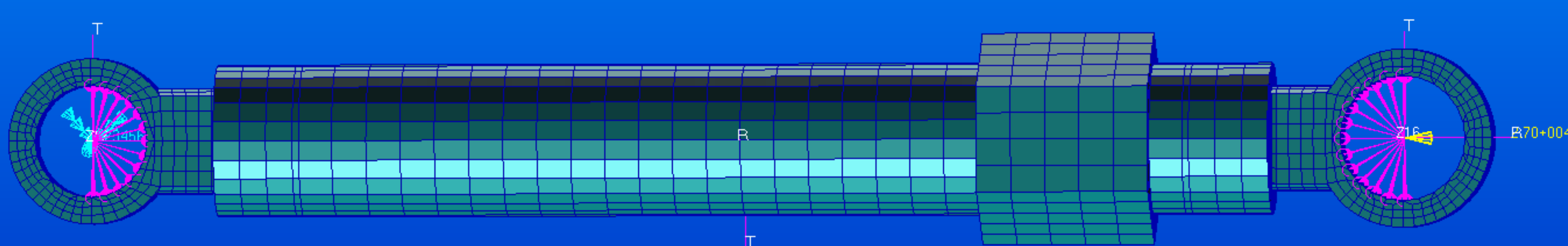
* Using a fine FEA model of the Link/Strut, calculate the spring rates for the tension and compression modes. A complete explanation is included in the "NSTXU- CALC-132-09". For clarity, some of that data is included here.
* Using the calculated FEA Spring Rates, displacements for tension and compression, calculate the <u>equivalent bar element geometry</u> which can be used in the simple FEA simulation using beam/bar elements.
* Create a simple stick model for link-strut-pin model.
* Apply loads in sequence by changing the model as one link/strut goes into compression.
* Use superposition to calculate the final forces, displacements and stresses.

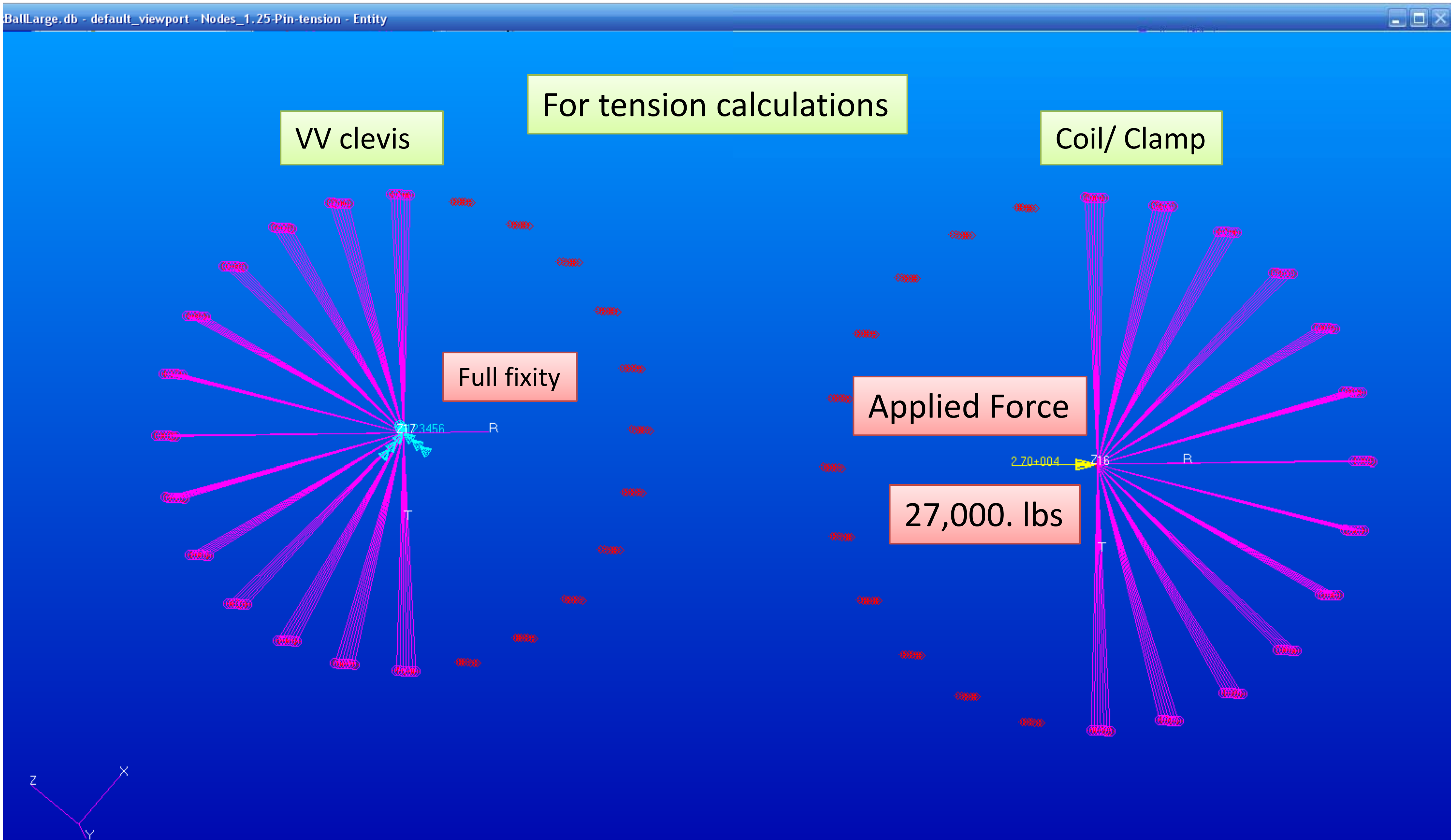
Note: All this is described in the following slides.
Please see the slides which demonstrate these geometric change/variation simulations.

Loads, fixity, and supports for Link in tension calculation

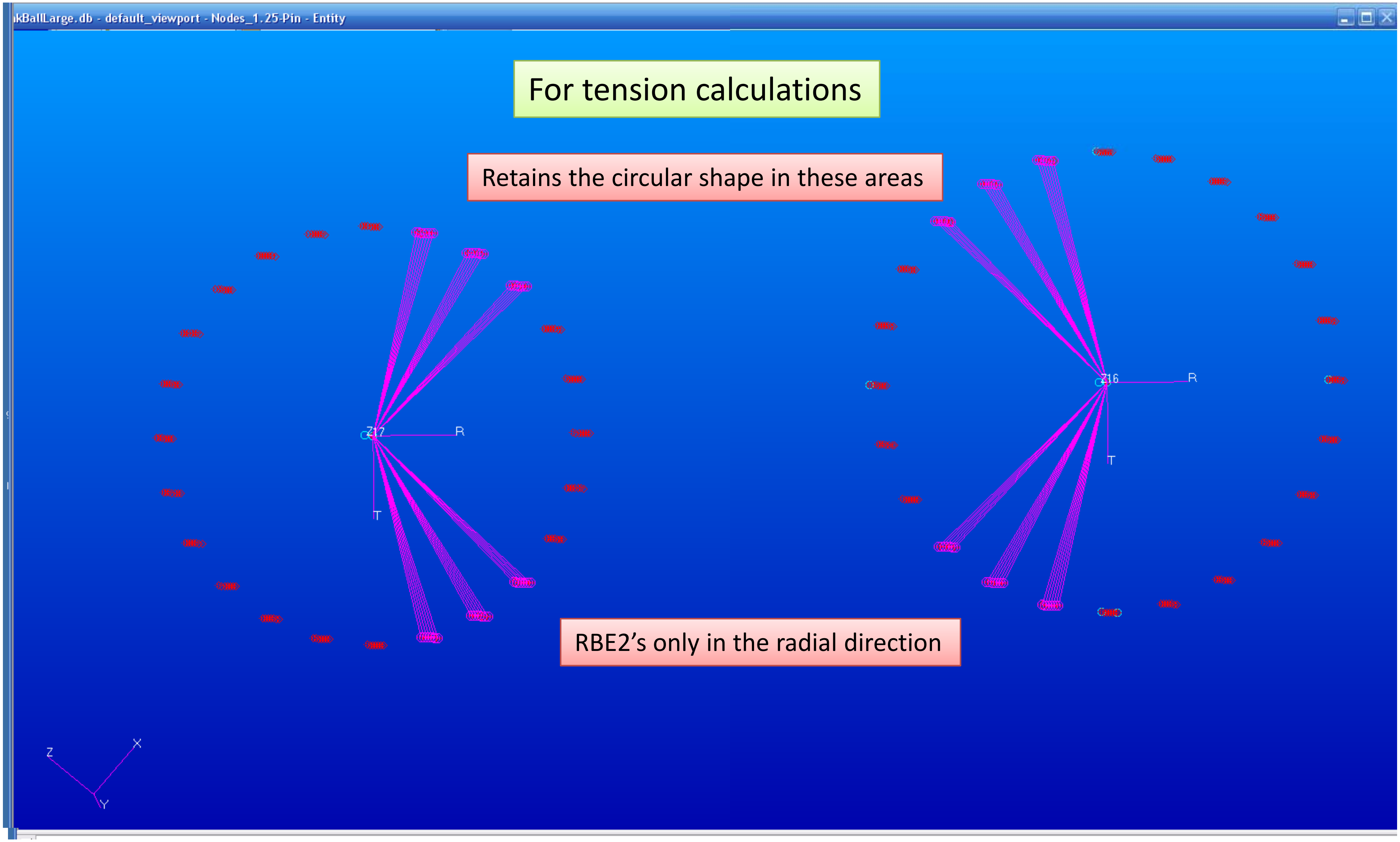


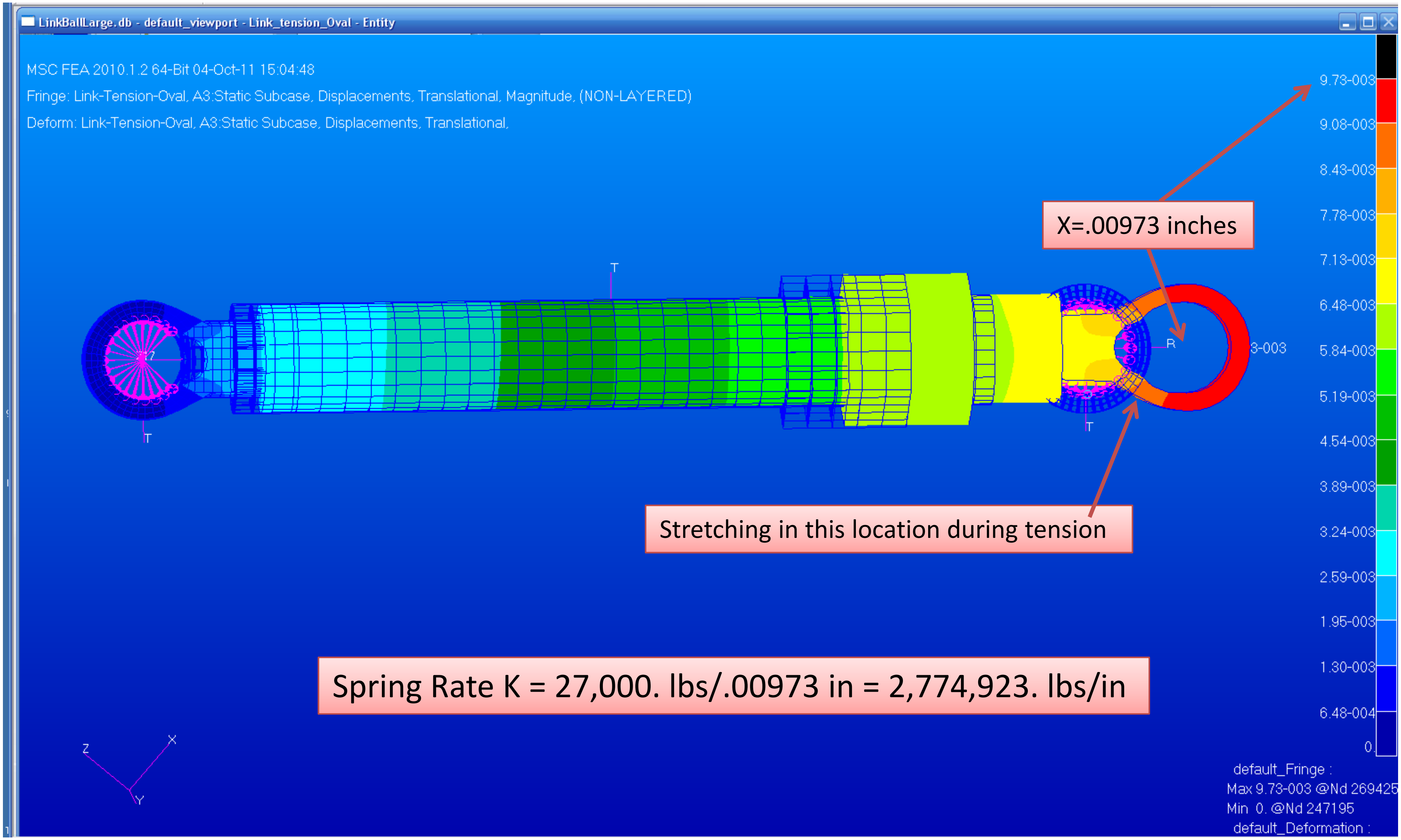
Loads, fixity, and supports for Link in Compression calculation

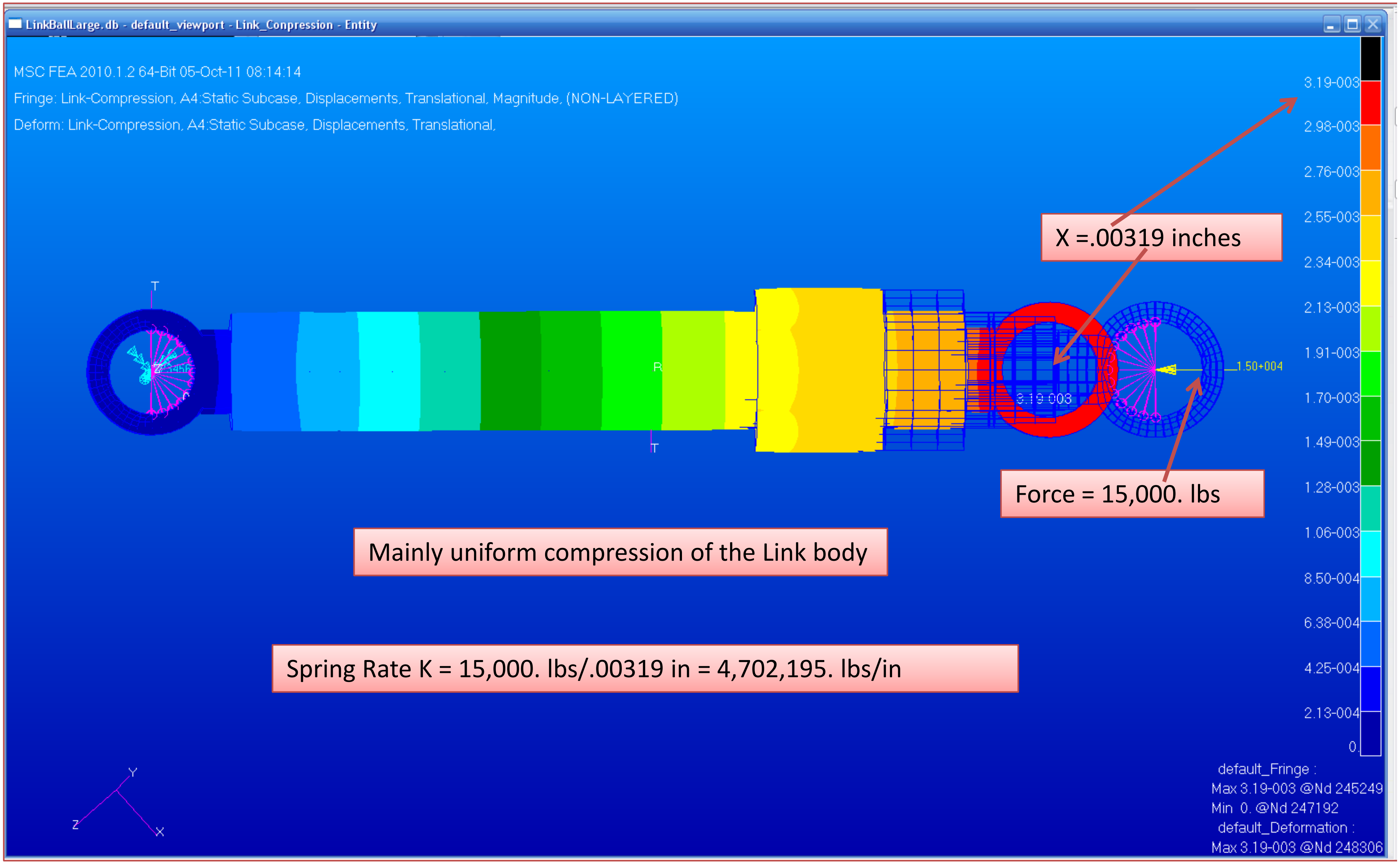





RBE2 elements, which simulate contacts to the Pins located at coil/clamp supports and the VV clevis locations, are considered fully fixed rigid links.
Note: For the link in compression calculations, RBE2 spider is reversed at the Coil/clamp location (pin diameter = 1.25 inch)


ıkBallLarge.db - default_viewport - Nodes_1.25-Pin - Entity
For tension calculations
Retains the circular shape in these areas
Z17
R
T
Z16
R
T
RBE2's only in the radial direction
Z
X
Y


LinkBallLarge.db - default_viewport - Link_tension_Oval - Entity
MSC FEA 2010.1.2 64-Bit 04-Oct-11 15:04:48
Fringe: Link-Tension-Oval, A3:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Link-Tension-Oval, A3:Static Subcase, Displacements, Translational,
X=.00973 inches
Stretching in this location during tension
Spring Rate K = 27,000. lbs/.00973 in = 2,774,923. lbs/in
9.73-003
9.08-003
8.43-003
7.78-003
7.13-003
6.48-003
5.84-003
5.19-003
4.54-003
3.89-003
3.24-003
2.59-003
1.95-003
1.30-003
6.48-004
0
default_Fringe :
Max 9.73-003 @Nd 269425
Min 0. @Nd 247195
default_Deformation :


LinkBallLarge.db - default_viewport - Link_Compression - Entity
MSC FEA 2010.1.2 64-Bit 05-Oct-11 08:14:14
Fringe: Link-Compression, A4:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Link-Compression, A4:Static Subcase, Displacements, Translational,
X =.00319 inches
Force = 15,000. lbs
Mainly uniform compression of the Link body
Spring Rate K = 15,000. lbs/.00319 in = 4,702,195. lbs/in
3.19-003
2.98-003
2.76-003
2.55-003
2.34-003
2.13-003
1.91-003
1.70-003
1.49-003
1.28-003
1.06-003
8.50-004
6.38-004
4.25-004
2.13-004
0.
1.50+004
3.19-003
default_Fringe :
Max 3.19-003 @Nd 245249
Min 0. @Nd 247192
default_Deformation :
Max 3.19-003 @Nd 248306

Calculation of the equivalent areas based on the actual link/strut Spring Rate "K"

Basic elongation equation is : e = FL/AE, F/e=AE/L=K, A=KL/E

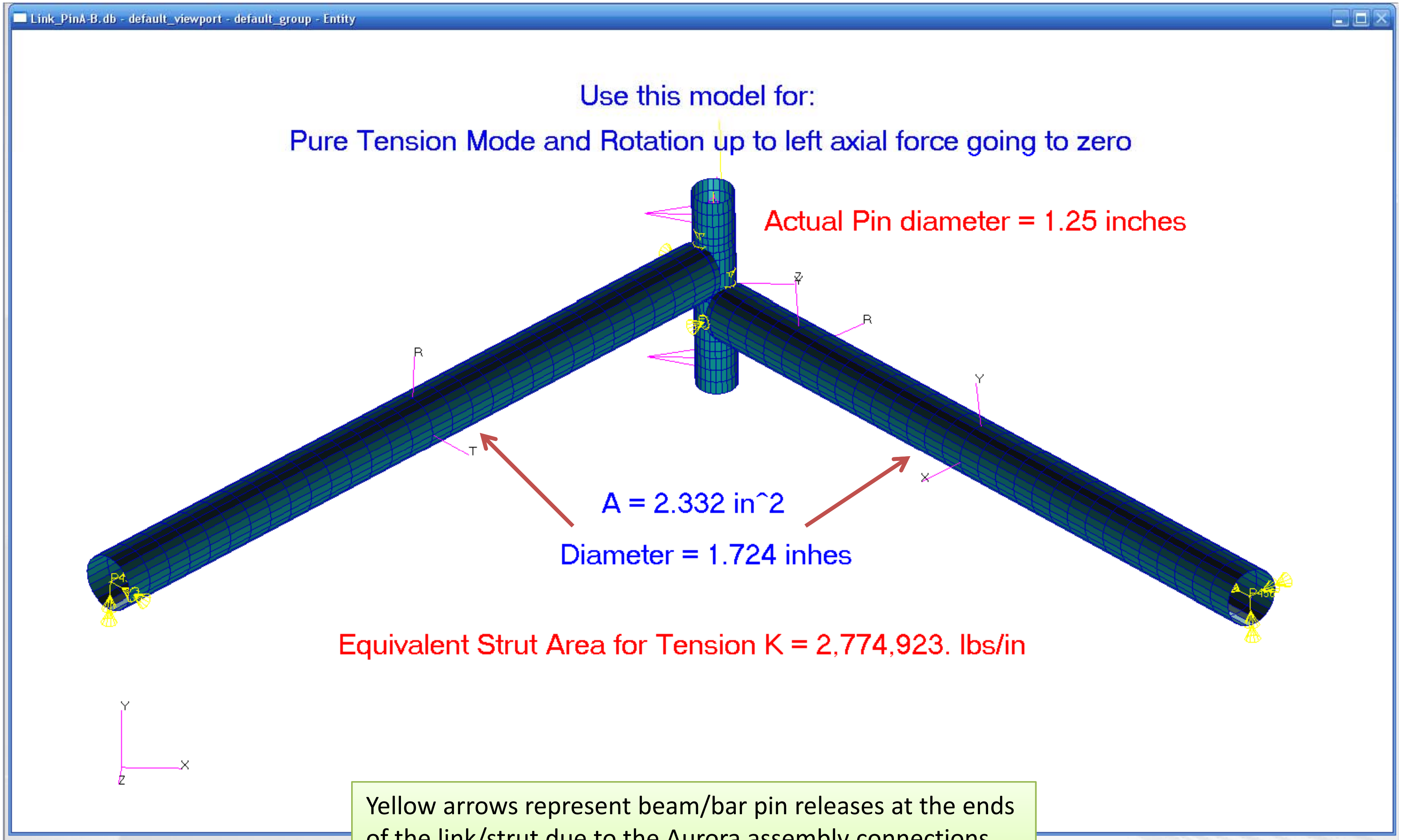
Data for struts: L = 24.875 inches
E = 29,000,000.
Tension K = 2,774,923. lbs/inch
Compression K = 4,702,195. lbs/inch

A (tension) = (2,774,923. x 24.875) / 29,000,000. = 2.332 in^2

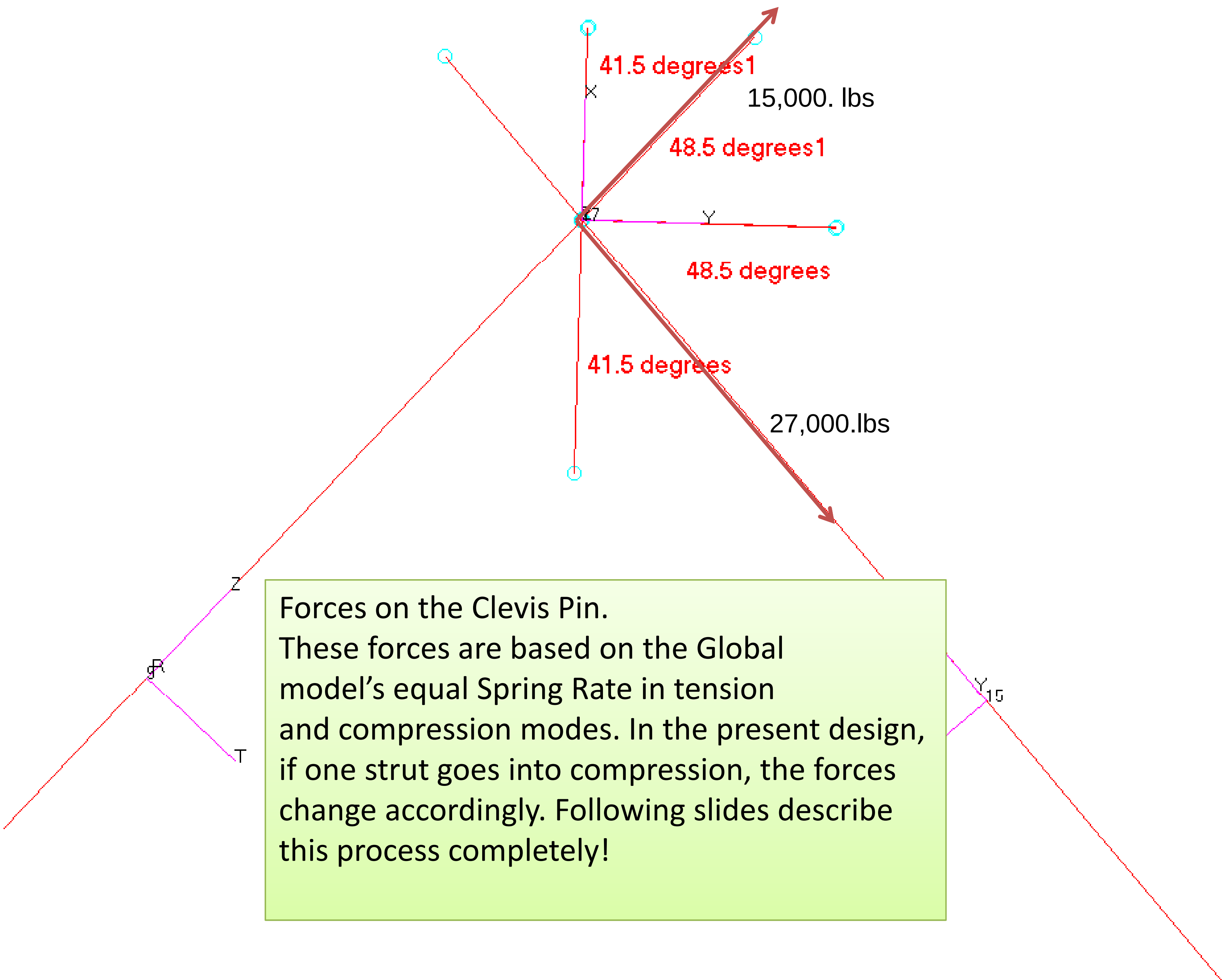
- A=pi r^2 ; therefore r = .8618 inches

A (compression) = (4,702,195. x 24.785) / 29,000,000. =3.952 in ^2

- A=pi r^2 ; therefore r = 1.122 inches




Yellow arrows represent beam/bar pin releases at the ends of the link/strut due to the Aurora assembly connections with the pin surfaces.



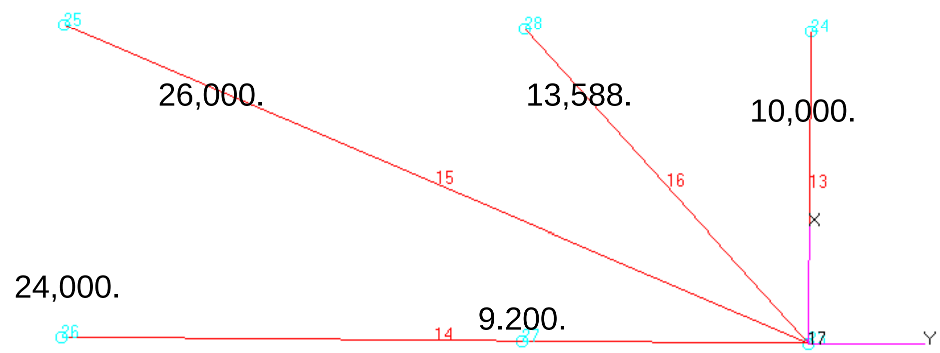

Forces on the Clevis Pin.
These forces are based on the Global model's equal Spring Rate in tension and compression modes. In the present design, if one strut goes into compression, the forces change accordingly. Following slides describe this process completely!

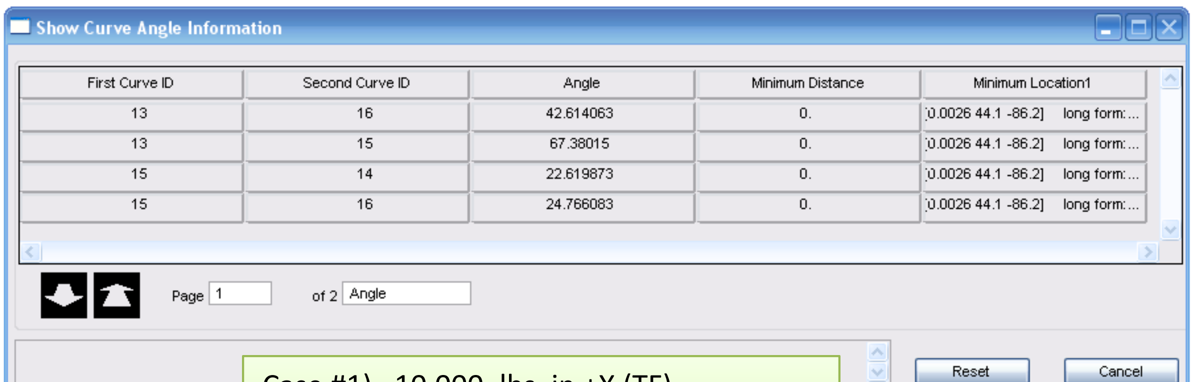
# History of the applied Force sequence

Vector Forces are in Lbs.

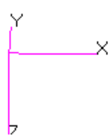
TF

PF



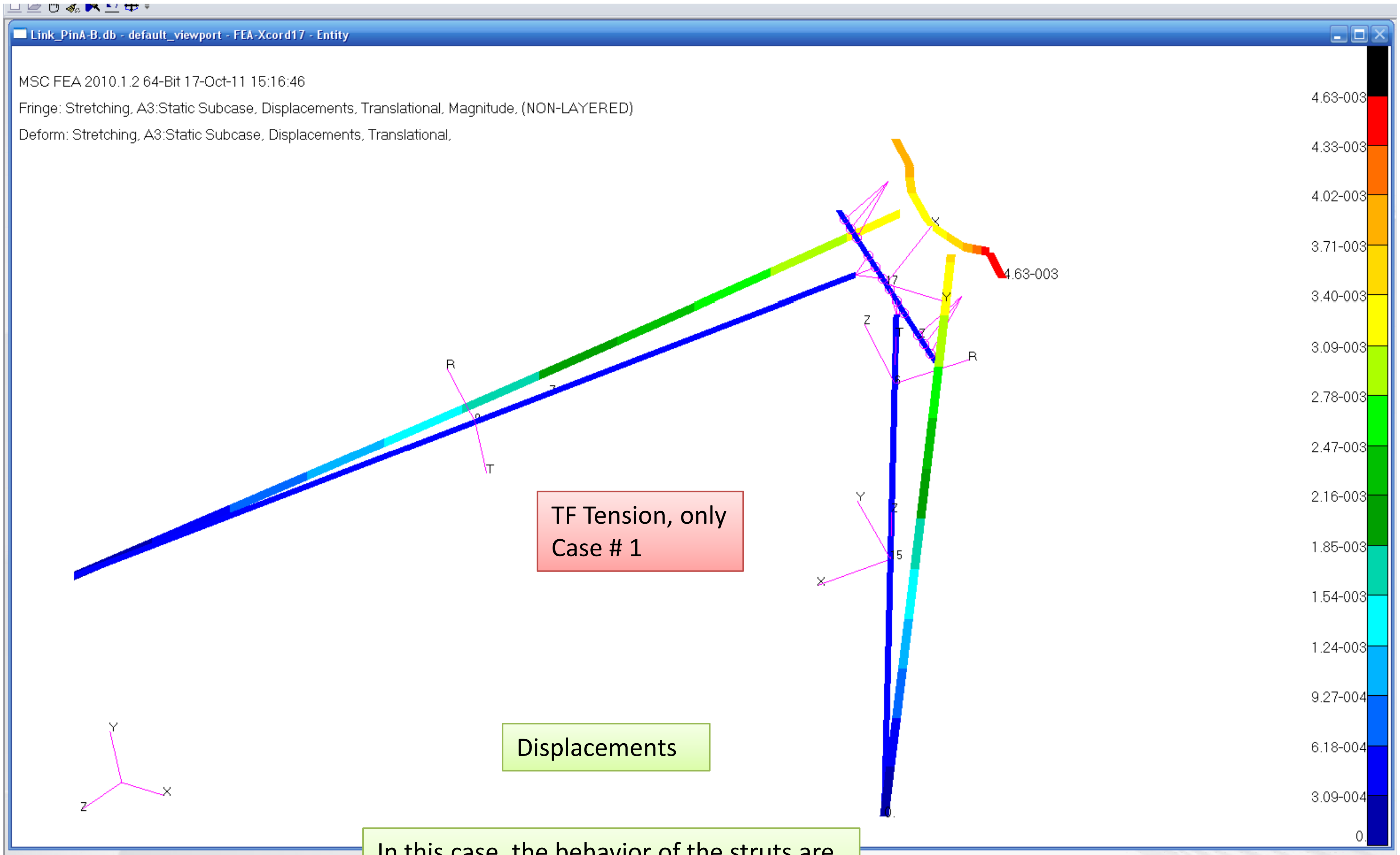




| First Curve ID | Second Curve ID | Angle | Minimum Distance | Minimum Location1 |
|---|---|---|---|---|
| 13 | 16 | 42.614063 | 0. | [0.0026 44.1 -86.2] long form: ... |
| 13 | 15 | 67.38015 | 0. | [0.0026 44.1 -86.2] long form: ... |
| 15 | 14 | 22.619873 | 0. | [0.0026 44.1 -86.2] long form: ... |
| 15 | 16 | 24.766083 | 0. | [0.0026 44.1 -86.2] long form: ... |




Case #1) 10,000. lbs. in +X (TF)
Case #2) 9,200. lbs. in -Y (PF)
Case #3 ) 14,800. lbs. in -Y (Additional PF)

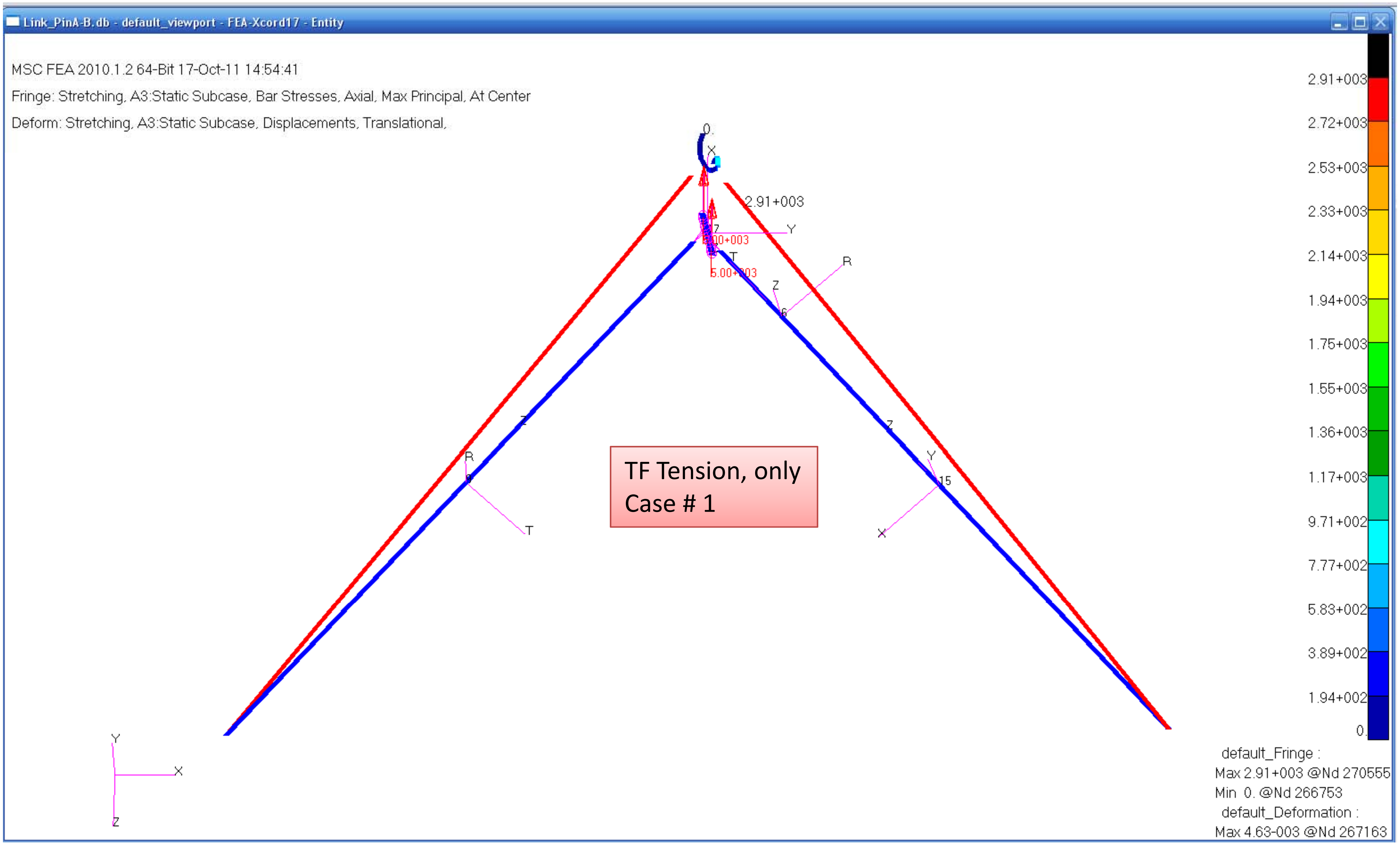
Results from each case are superimposed for the combined final results

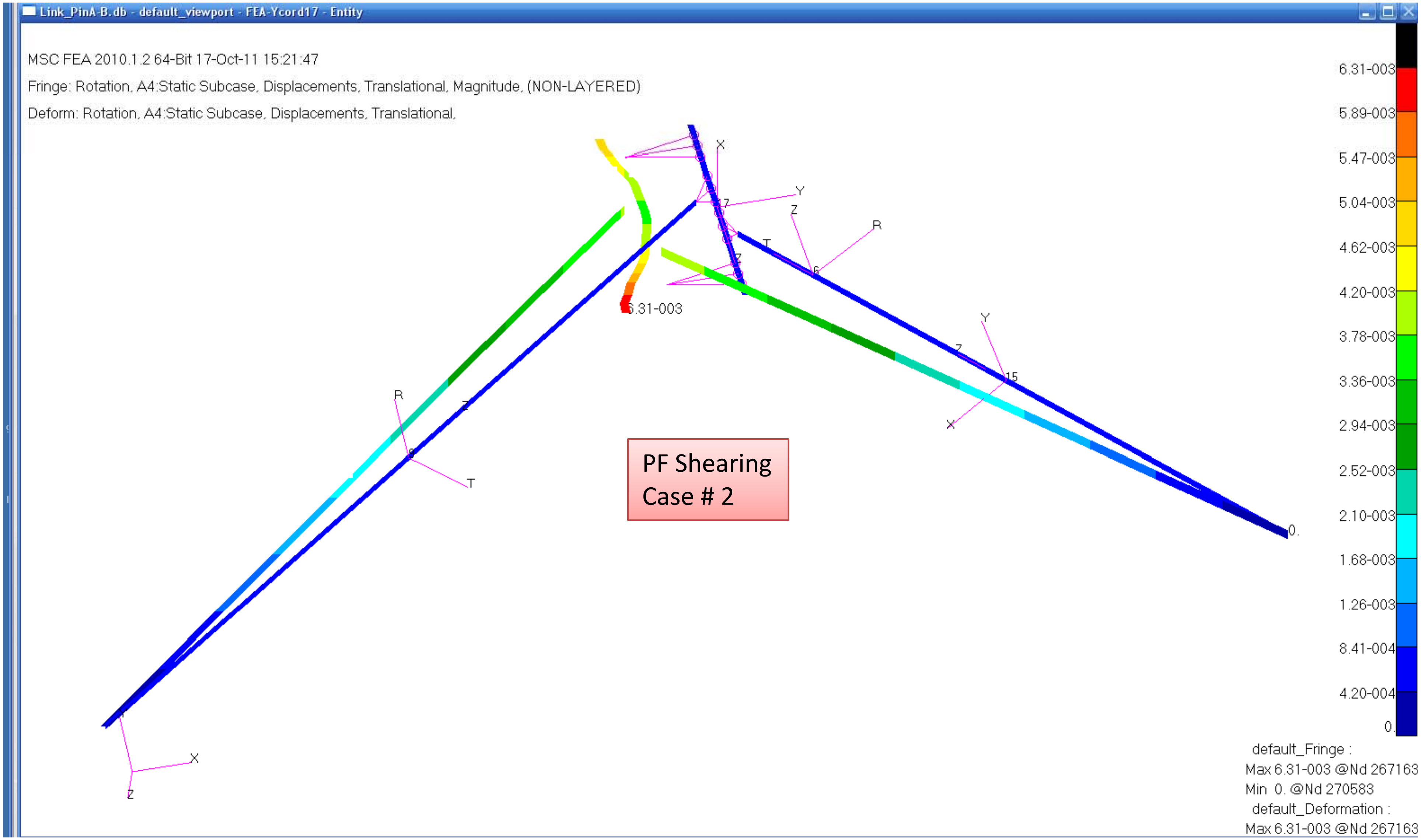


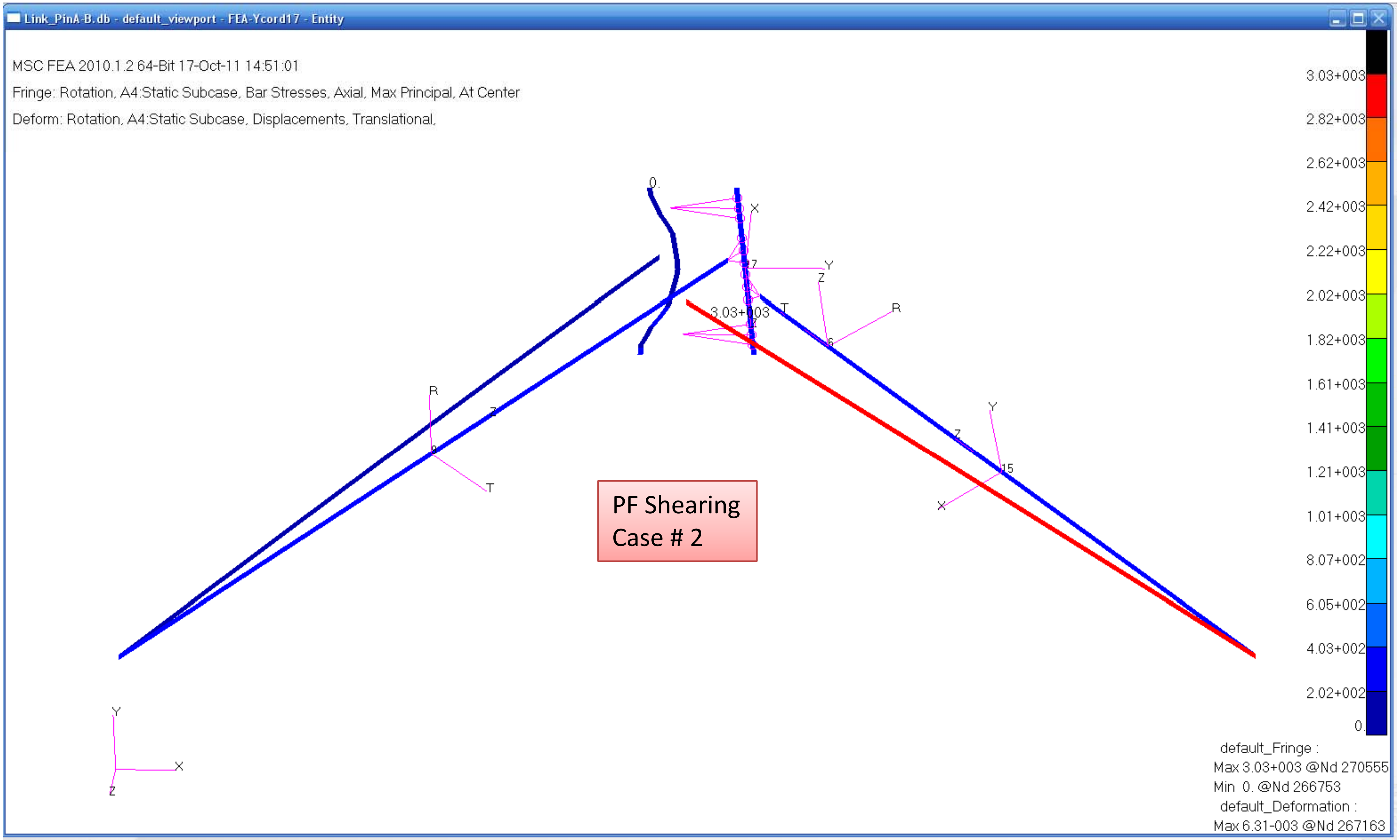

TF Tension, only Case # 1

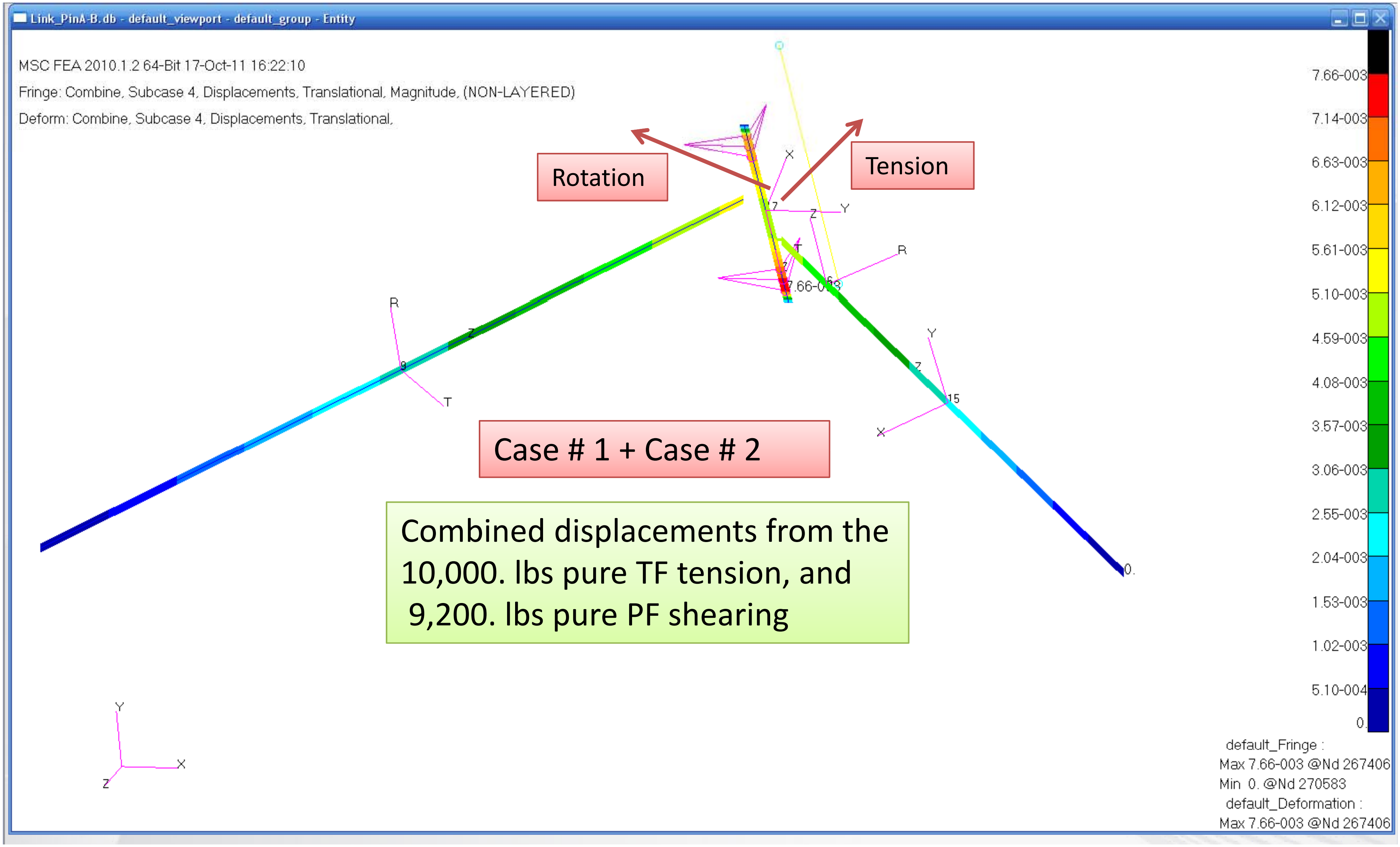
Displacements

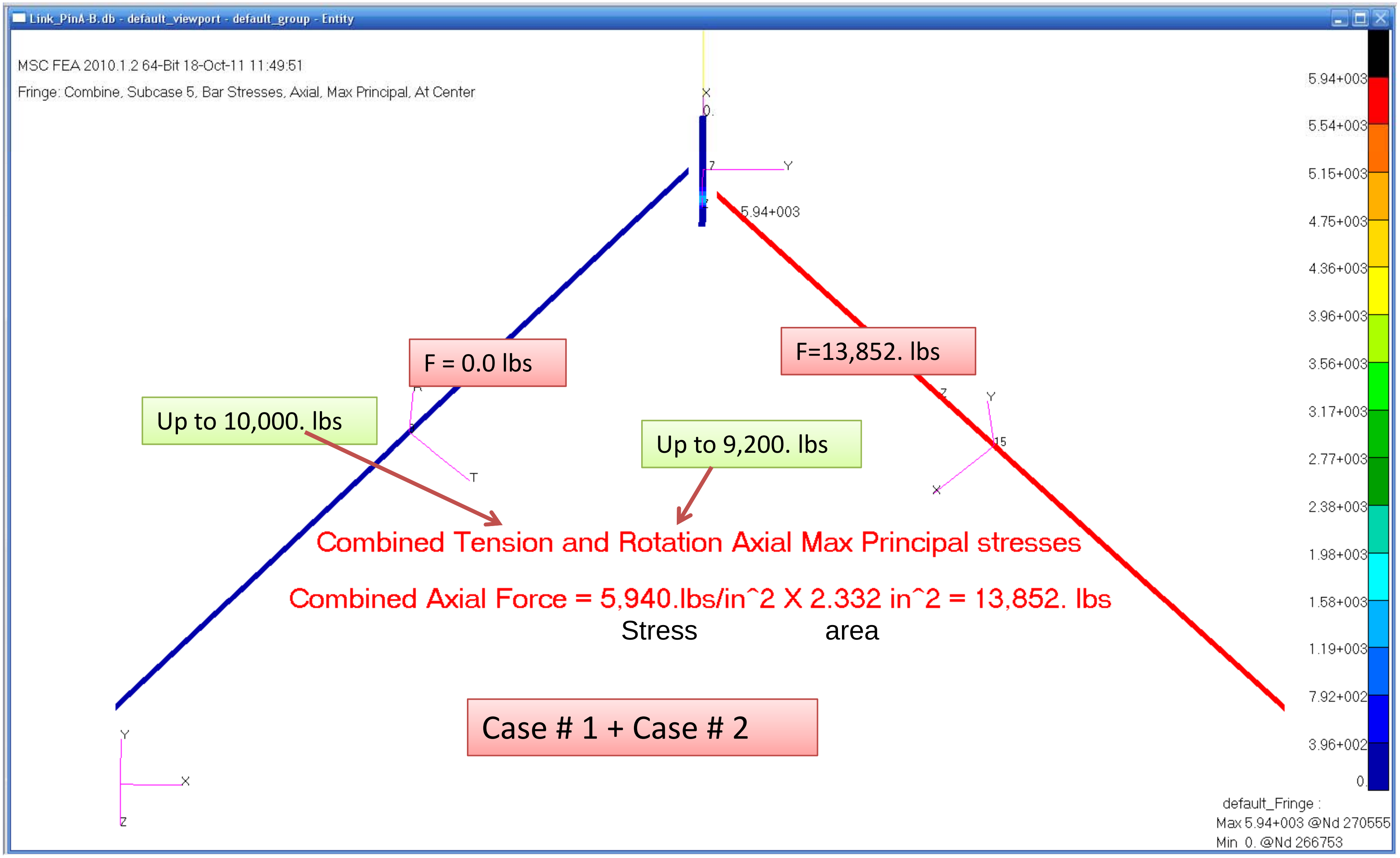
In this case, the behavior of the struts are about the same! This is expected .


Link_PinA-B.db - default_viewport - FEA-Xcord17 - Entity
MSC FEA 2010.1.2 64-Bit 17-Oct-11 14:54:41
Fringe: Stretching, A3:Static Subcase, Bar Stresses, Axial, Max Principal, At Center
Deform: Stretching, A3:Static Subcase, Displacements, Translational,
2.91+003
TF Tension, only
Case # 1
2.91+003
2.72+003
2.53+003
2.33+003
2.14+003
1.94+003
1.75+003
1.55+003
1.36+003
1.17+003
9.71+002
7.77+002
5.83+002
3.89+002
1.94+002
0.
default_Fringe :
Max 2.91+003 @Nd 270555
Min 0. @Nd 266753
default_Deformation :
Max 4.63-003 @Nd 267163


Link_PinA-B.db - default_viewport - FEA-Ycord17 - Entity
MSC FEA 2010.1.2 64-Bit 17-Oct-11 15:21:47
Fringe: Rotation, A4:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Rotation, A4:Static Subcase, Displacements, Translational,
6.31-003
5.89-003
5.47-003
5.04-003
4.62-003
4.20-003
3.78-003
3.36-003
2.94-003
2.52-003
2.10-003
1.68-003
1.26-003
8.41-004
4.20-004
0.
PF Shearing
Case # 2
default_Fringe :
Max 6.31-003 @Nd 267163
Min 0. @Nd 270583
default_Deformation :
Max 6.31-003 @Nd 267163

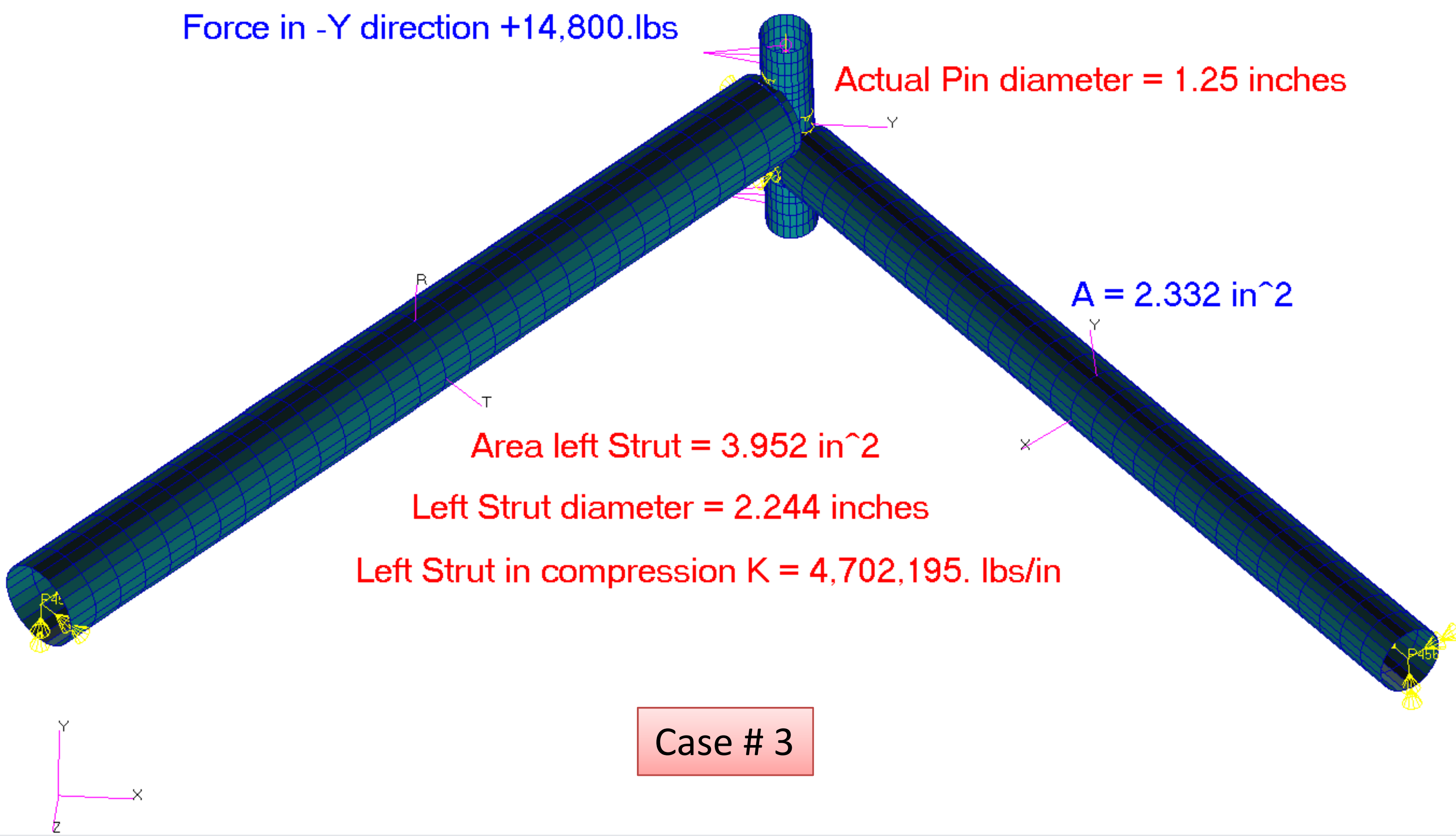

Link_PinA-B.db - default_viewport - FEA-Ycord17 - Entity
MSC FEA 2010.1.2 64-Bit 17-Oct-11 14:51:01
Fringe: Rotation, A4:Static Subcase, Bar Stresses, Axial, Max Principal, At Center
Deform: Rotation, A4:Static Subcase, Displacements, Translational,
PF Shearing
Case # 2
3.03+003
3.03+003
2.82+003
2.62+003
2.42+003
2.22+003
2.02+003
1.82+003
1.61+003
1.41+003
1.21+003
1.01+003
8.07+002
6.05+002
4.03+002
2.02+002
0.
default_Fringe :
Max 3.03+003 @Nd 270555
Min 0. @Nd 266753
default_Deformation :
Max 6.31-003 @Nd 267163

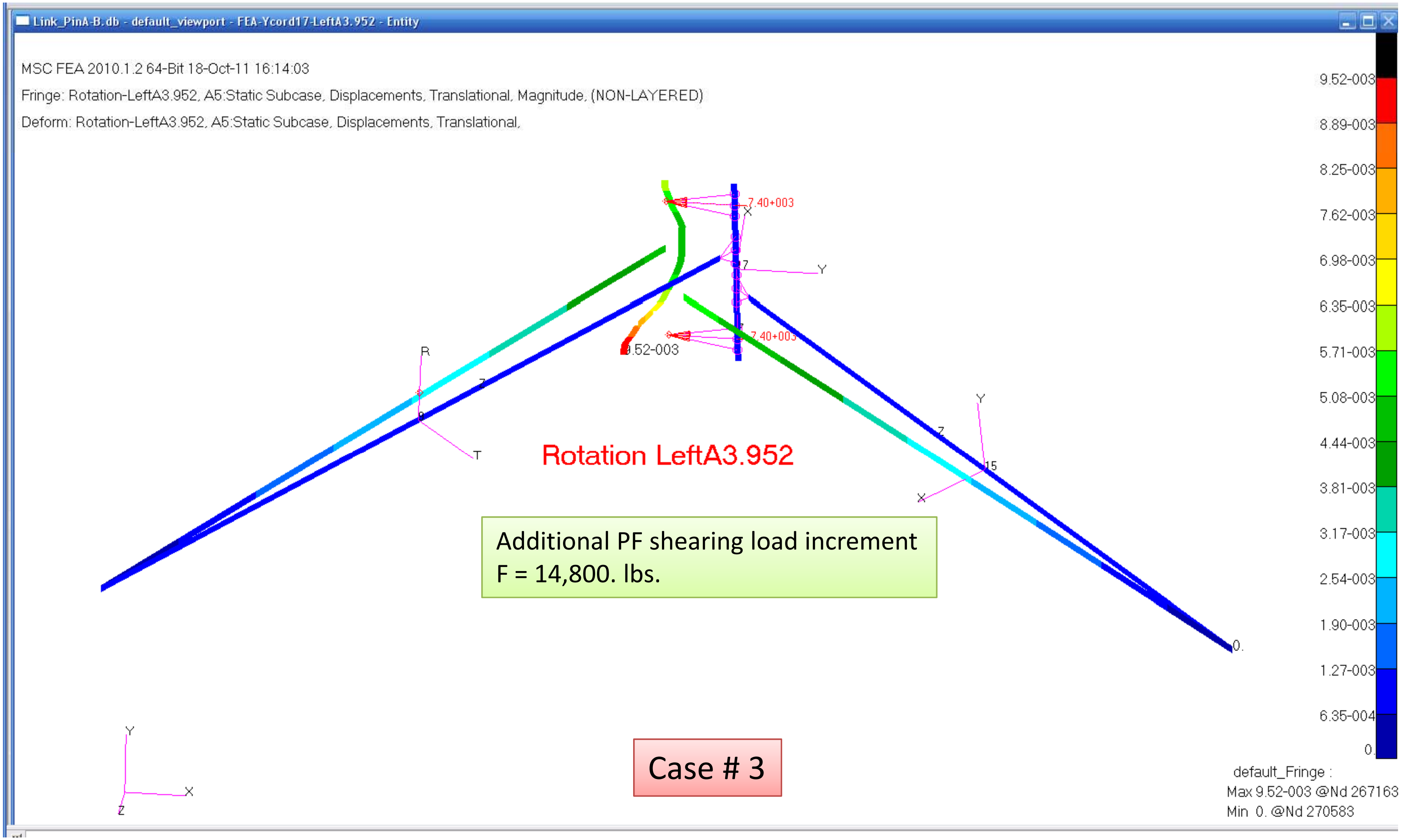

Link_PinA-B.db - default_viewport - default_group - Entity
MSC FEA 2010.1.2 64-Bit 17-Oct-11 16:22:10
Fringe: Combine, Subcase 4, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Combine, Subcase 4, Displacements, Translational,
Rotation
Tension
Case # 1 + Case # 2
Combined displacements from the
10,000. lbs pure TF tension, and
9,200. lbs pure PF shearing
7.66-003
7.14-003
6.63-003
6.12-003
5.61-003
5.10-003
4.59-003
4.08-003
3.57-003
3.06-003
2.55-003
2.04-003
1.53-003
1.02-003
5.10-004
0.
default_Fringe :
Max 7.66-003 @Nd 267406
Min 0. @Nd 270583
default_Deformation :
Max 7.66-003 @Nd 267406

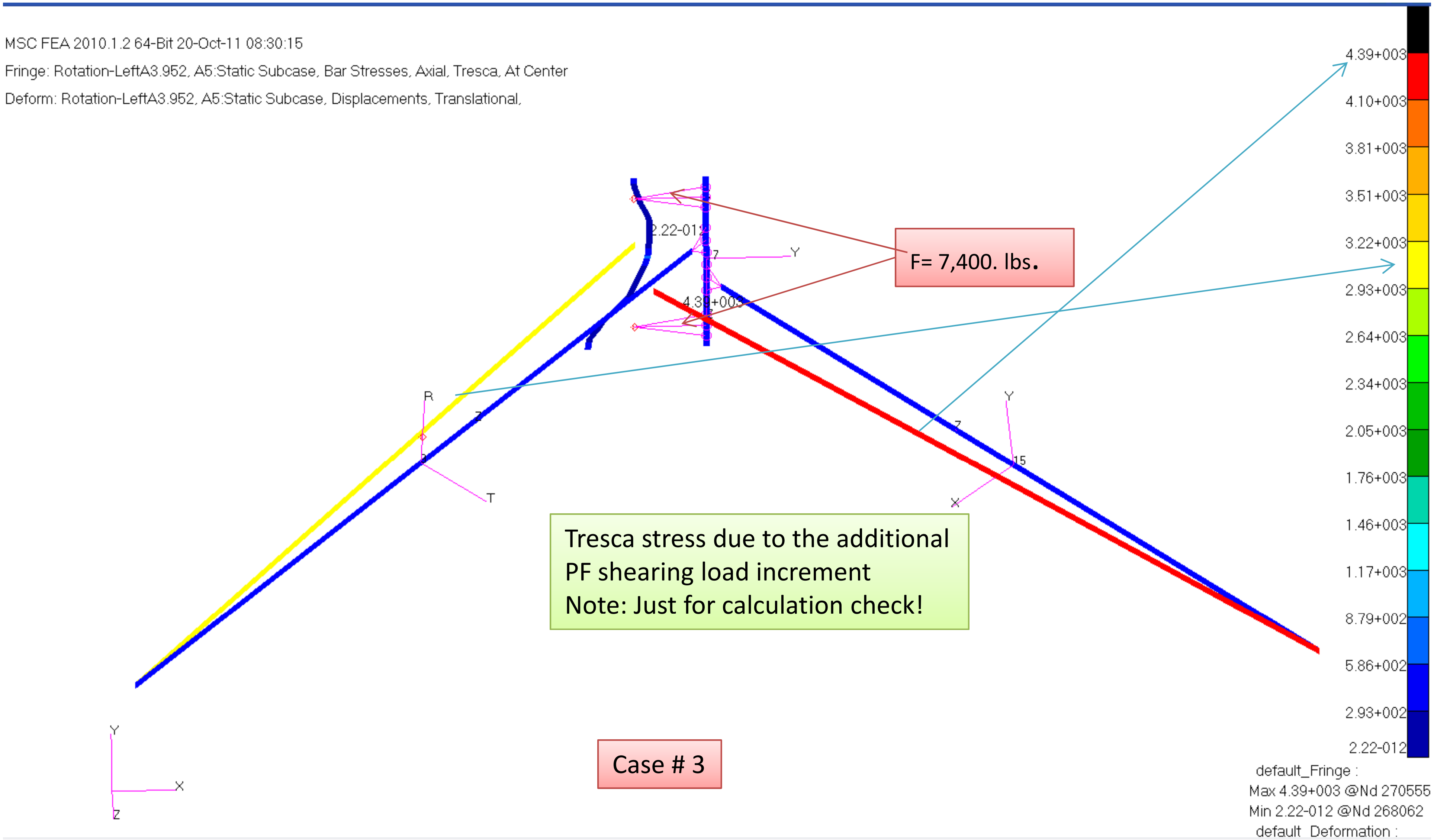


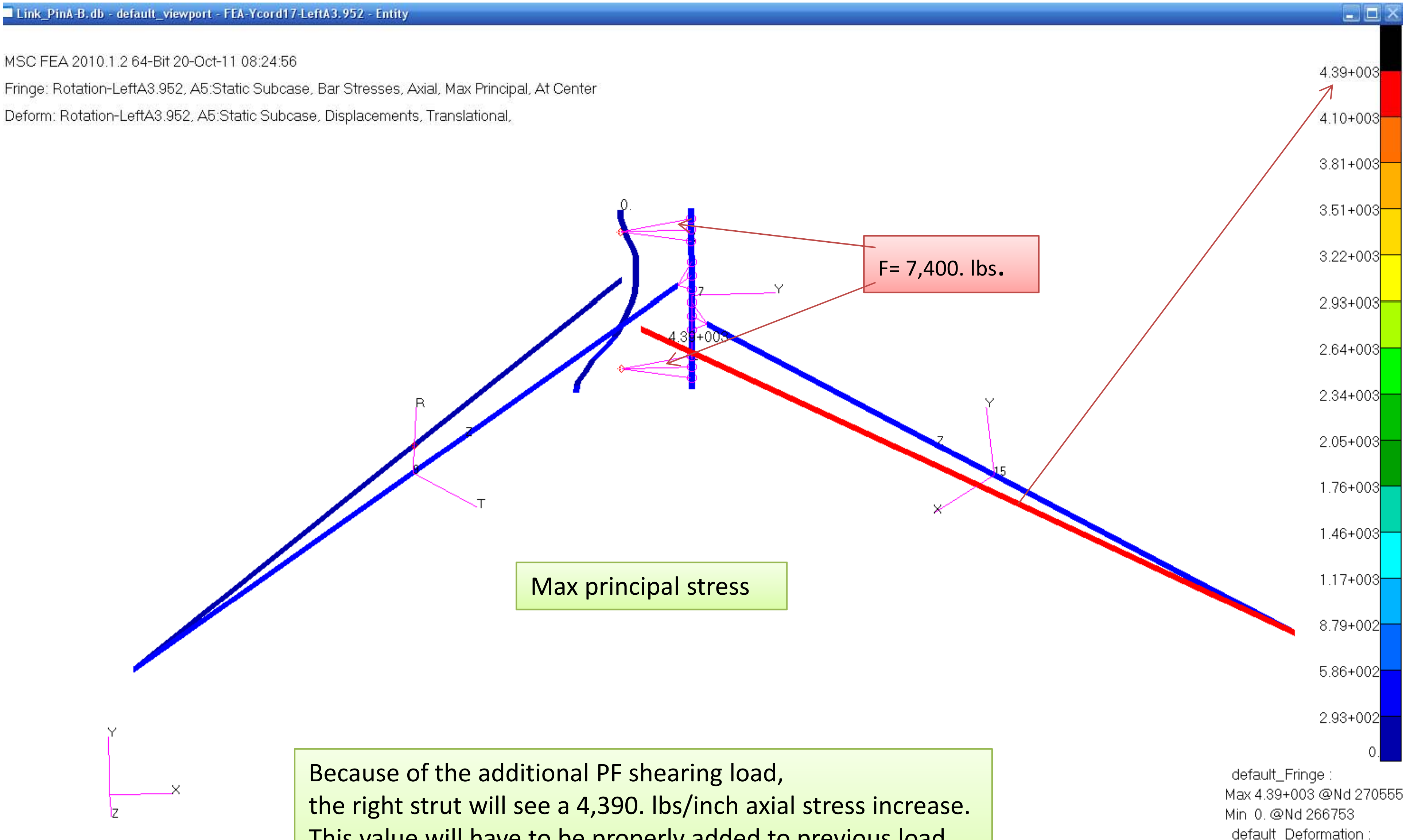
Link_PinA-B.db - default_viewport - default_group - Entity
MSC FEA 2010.1.2 64-Bit 18-Oct-11 11:49:51
Fringe: Combine, Subcase 5, Bar Stresses, Axial, Max Principal, At Center
5.94+003
F = 0.0 lbs
F=13,852. lbs
Up to 10,000. lbs
Up to 9,200. lbs
Combined Tension and Rotation Axial Max Principal stresses
Combined Axial Force = 5,940.lbs/in^2 X 2.332 in^2 = 13,852. lbs
Stress
area
Case # 1 + Case # 2
5.94+003
5.54+003
5.15+003
4.75+003
4.36+003
3.96+003
3.56+003
3.17+003
2.77+003
2.38+003
1.98+003
1.58+003
1.19+003
7.92+002
3.96+002
0.
default_Fringe :
Max 5.94+003 @Nd 270555
Min 0. @Nd 266753

## Continuation of increase of the rotational force with left Strut in compression



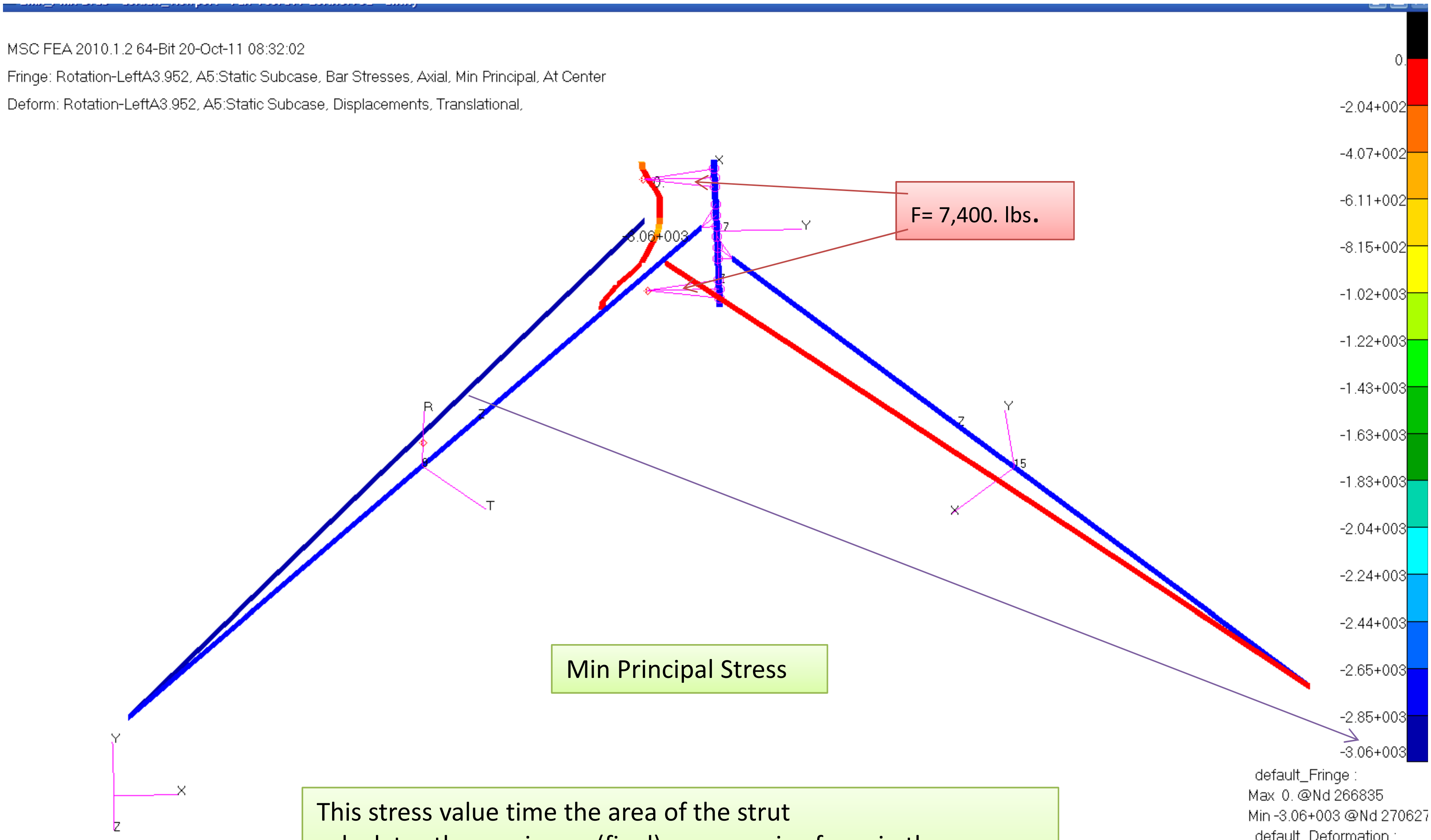

Link_PinA-B.db - default_viewport - FEA-Ycord17-LeftA3.952 - Entity
MSC FEA 2010.1.2 64-Bit 18-Oct-11 16:14:03
Fringe: Rotation-LeftA3.952, A5:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Rotation-LeftA3.952, A5:Static Subcase, Displacements, Translational,
Rotation LeftA3.952
Additional PF shearing load increment
F = 14,800. lbs.
Case # 3
9.52-003
8.89-003
8.25-003
7.62-003
6.98-003
6.35-003
5.71-003
5.08-003
4.44-003
3.81-003
3.17-003
2.54-003
1.90-003
1.27-003
6.35-004
0.
default_Fringe :
Max 9.52-003 @Nd 267163
Min 0. @Nd 270583


MSC FEA 2010.1.2 64-Bit 20-Oct-11 08:30:15
Fringe: Rotation-LeftA3.952, A5:Static Subcase, Bar Stresses, Axial, Tresca, At Center
Deform: Rotation-LeftA3.952, A5:Static Subcase, Displacements, Translational,
F= 7,400. lbs.
Tresca stress due to the additional PF shearing load increment
Note: Just for calculation check!
Case # 3
4.39+003
4.10+003
3.81+003
3.51+003
3.22+003
2.93+003
2.64+003
2.34+003
2.05+003
1.76+003
1.46+003
1.17+003
8.79+002
5.86+002
2.93+002
2.22-012
default_Fringe :
Max 4.39+003 @Nd 270555
Min 2.22-012 @Nd 268062
default_Deformation :




Because of the additional PF shearing load,
the right strut will see a 4,390. lbs/inch axial stress increase.
This value will have to be properly added to previous load condition to calculate the total force in the strut.

Case # 3




This stress value time the area of the strut calculates the maximum (final) compressive force in the left strut = Equiv. A x Axial Stress = 3.952 x -3,060. = - 12,093. lbs

Case # 3

# Results Summary

Case #1
Force Lbs.

| Left | Strut | Right |
|---|---|---|
| 6,786. | | 6,786. |

Case #1 + Case #2
Force Lbs.

| Left | Strut | Right |
|---|---|---|
| 0.0. | | 13,892. |

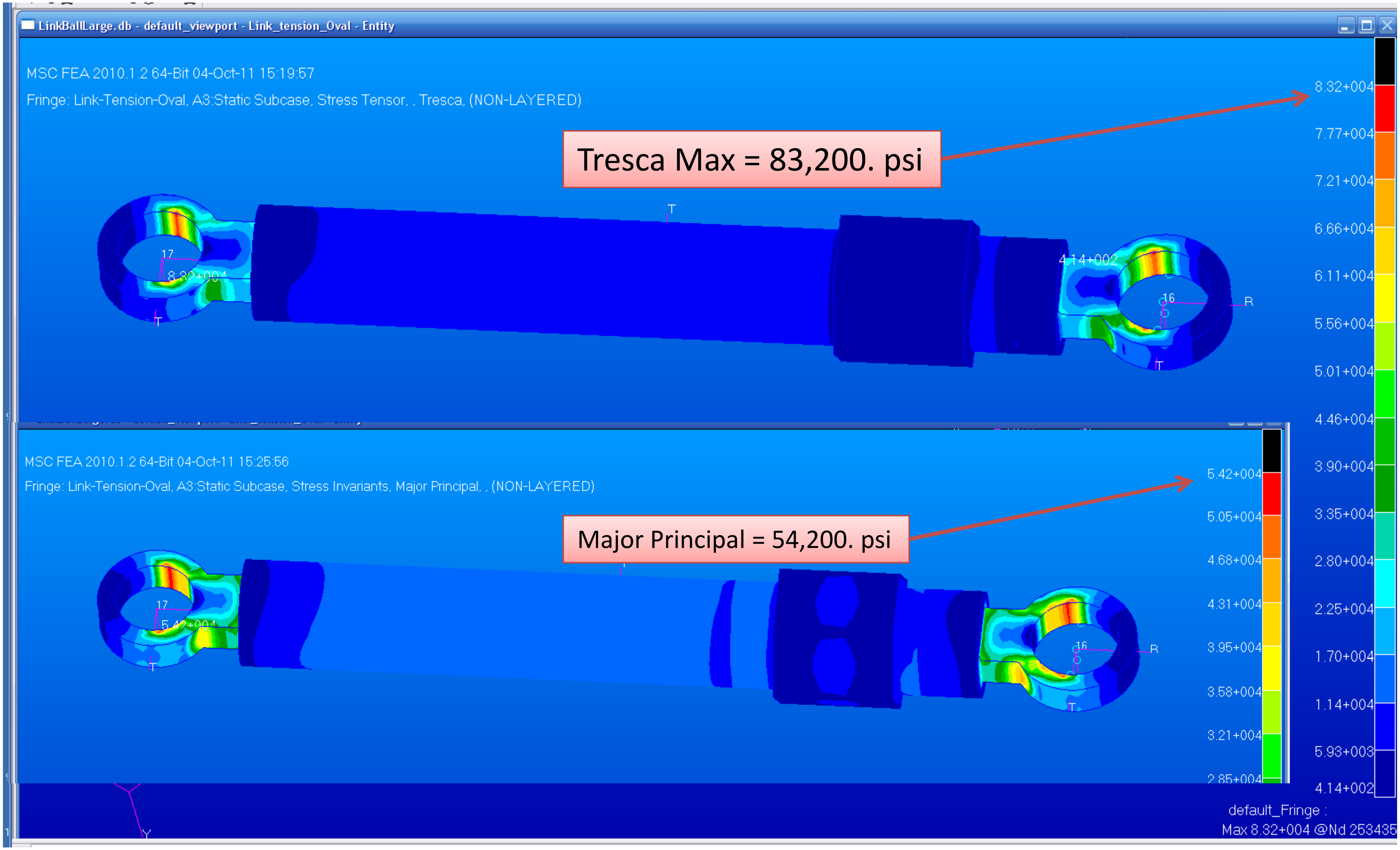
Case #3
Force Lbs.

| Left | Strut | Right |
|---|---|---|
| -12,093. | | 10,375. |

Total superimposed Case#1 + Case#2 + Case#3
Left Force = -12,093. lbs, Right Force = 24,270. lbs
These are the forces reacting at the coil clamp pin.

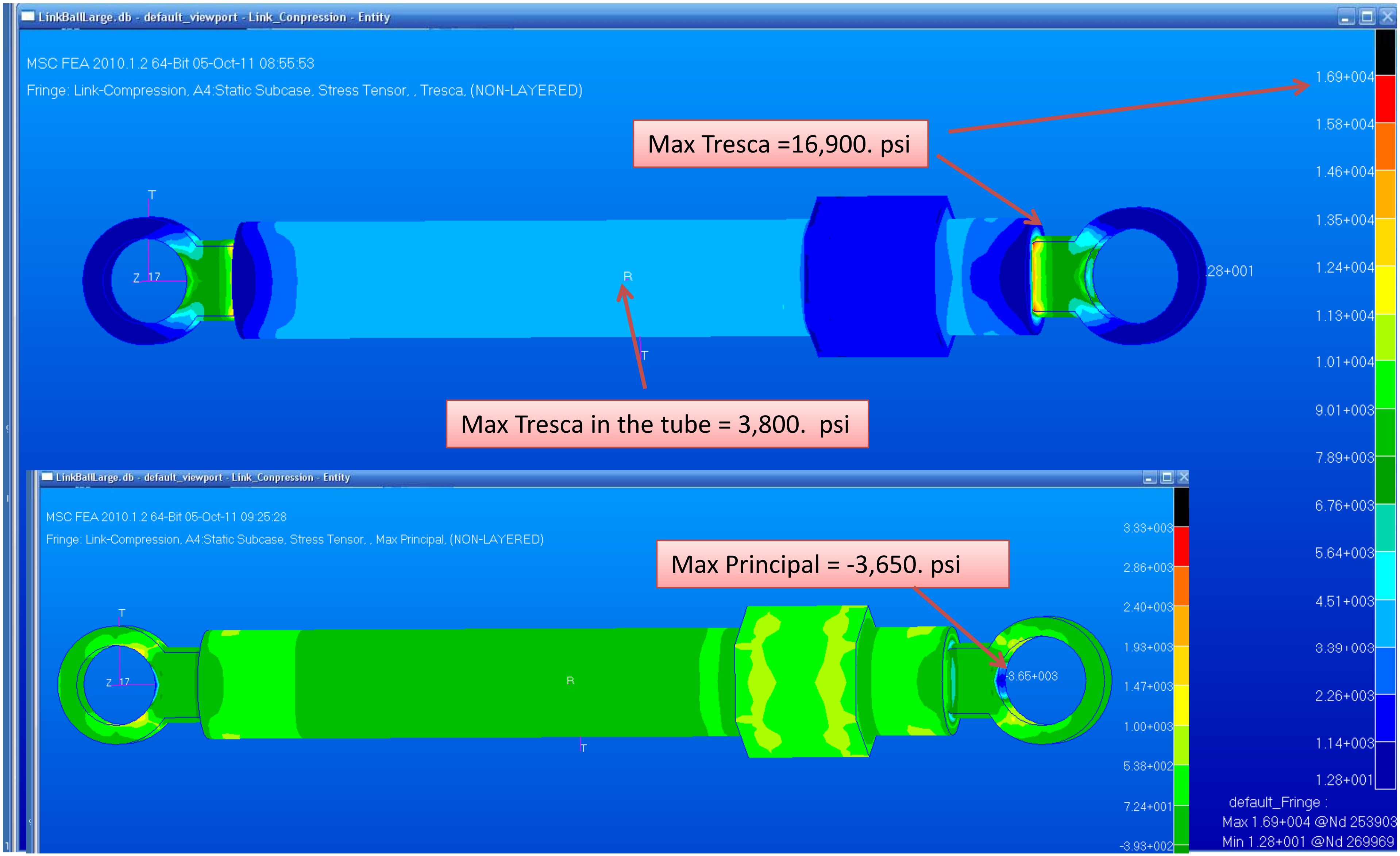
Important Finding: For this strut configuration, if the TF And PF forces are about equal, the strut in the direction of the PF force will have zero force reaction. Please see above data.

Note: Calculated Stresses in this simulation, are not the real stresses developed in the actual Link/Struts.
For real stress explanations, please check "NSTXU-132-09-00" data. (Also see Appendix A1 and A2)
For this simulation, stress values are used only to calculate the average force developed in the links.




Strut in tension

Appendix A 2




Strut in compression

# Appendix 2
# Nastran data base files

On “ASALEHZ-64PC”

G:\Nastran P.R\LinksBallClevis|LinkPinA-B.db

G:\Nastran P.R\LinksBallClevis\LinkBallLarge.db

OR:

P:\public\Snap-srv\progoff\Nastran P.R\LinksBallClevis

# Ring_Coil_Joint.ppt

The presented Nastran Model (simulation) was initiated in early 2011 to study potential changes in the actual design of the main ring and connections to the VV surface. In the end , it demonstrated the necessity to change the clevis, pin, and link/strut geometry. The loads and imposed displacements (SPCDs) were obtained from the various ANSYS global simulations which were executed at that time.

```
                        C A S E    C O N T R O L    E C H O
COMMAND
COUNT
 1      $ DIRECT TEXT INPUT FOR GLOBAL CASE CONTROL DATA
 2      TITLE = MSC.NASTRAN JOB CREATED ON 23-JAN-12 AT 09:01:04
 3      ECHO = NONE
 4      MPC = 20133
 5      SUBCASE 1
 6         SPC = 15
 7         LOAD = 16
 8         DISPLACEMENT(SORT1,REAL)=ALL
 9         SPCFORCES(SORT1,REAL)=ALL
10         STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=ALL
11         GPFORCE=ALL
12      $ DIRECT TEXT INPUT FOR THIS SUBCASE
13      BEGIN BULK
             INPUT BULK DATA ENTRY COUNT =  293840
       TOTAL COUNT=    293782

                        M O D E L   S U M M A R Y

                  NUMBER OF GRID     POINTS   =   109973

                  NUMBER OF CBEAM    ELEMENTS =       40
                  NUMBER OF CHEXA    ELEMENTS =    91380
                  NUMBER OF CPENTA   ELEMENTS =      417

                  NUMBER OF RBE2     ELEMENTS =        4
```

## General loads and fixities (for reference)

```
$ Loads for Load Case : Untitled.SC1
SPCADD   15      2       4       1       3
$ Enforced Displacements for Load Set : spcd.2
SPCD     16      274199  1       -.002   274199  3       -.001
LOAD     16     1.      1.       5      1.       6      1.       7
        1.       8      1.       4      1.       10     1.       11
        1.       12     1.       13     1.       14     1.       3
        1.       9      1.       1
$ Displacement Constraints of Load Set : spcd.2
SPC1     2       13      274199
$ Displacement Constraints of Load Set : spc1.4
SPC1     4       123456  275203
$ Displacement Constraints of Load Set : spc1.1
SPC1     1       123456  275224  275245
$ Displacement Constraints of Load Set : spc1.3
SPC1     3       123456  274199
$ Nodal Forces of Load Set : force.1.cid23
FORCE    1       153878  23      4989.75 .851532 .422446 .310537
$ Nodal Forces of Load Set : force.3.cid23
FORCE    3       51623   23      7280.38 .851385 .415267 .320464
$ Nodal Forces of Load Set : force.4.cid23
FORCE    4       7004    23      9404.85 .88415  .395902 .248074
$ Nodal Forces of Load Set : force.5.cid23
FORCE    5       2423    23      9149.43 .913631 .342218 .219467
$ Nodal Forces of Load Set : force.6.cid23
FORCE    6       24972   23      8843.69 .946437 .26152  .189378
$ Nodal Forces of Load Set : force.7.cid23
FORCE    7       155374  23      4919.01 .857815 .42165  .29388
$ Nodal Forces of Load Set : force.8.cid23
FORCE    8       152382  23      4971.92 .842713 .427984 .326594
$ Nodal Forces of Load Set : force.9.cid23
FORCE    9       61127   23      8432.66 .82637  .44164  .34938
$ Nodal Forces of Load Set : force.10.cid23
FORCE    10      60192   23      12171.7 .81715  .434007 .379347
$ Nodal Forces of Load Set : force.11.cid23
FORCE    11      65165   23      12379.7 .805984 .416417 .420698
$ Nodal Forces of Load Set : force.12.cid23
FORCE    12      26322   23      4332.06 .966352 .194896 .167865
$ Nodal Forces of Load Set : force.13.cid23
FORCE    13      71421   23      12533.2 .79387  .394242 .462972
$ Nodal Forces of Load Set : force.14.cid23
FORCE    14      71795   23      6317.33 .785491 .376172 .491426
$ Referenced Coordinate Frames
CORD2C   23              0.      0.      0.      0.      1.      0.
         0.      0.      -1.
ENDDATA 8d9abeeb
```

# Explanation for the included analyses!

Model simulations depicted below are identical for the finite element representations using the the same geometry and formulation. However, some differences are as follows: Boundary conditions, loads and MPC symmetry conditions are identical, except at the link/strut/VV interface for simulation “A7”. Displacement equal to 0.5 mm (estimate), in the negative radial direction, was applied to account for any VV external pressure influenced deformations. So, the results are given for two cases, A4 and A7. The assumption for A7 is, that if the VV shrinks somehow, then the main rings and the connecting links would go into some pre load condition. However, for both load conditions, the obtained stresses in the main parts of the coil/clamp assembly (excluding the clevis and the link pin) are within the material requirements for 316 stainless steel (Sy=42,000.psi).

Forces were calculated from the existing global ANSYS models and applied as shown. Enforced displacements at the coil cuts were also estimated from the same simulations. NASTRAN MPCs (long pink lines) simulate the symmetry conditions of the main ring at 30 degree intervals.

In both models, the clevis-pin interfaces show very high stresses requiring the redesign of these members. However, the general forces and moments going through the main assembly probably remain similar.
The fully changed design and FEA calculations of the links, pin, and pin supports are documented in the “NXTSU-CALC-132-12-00”.

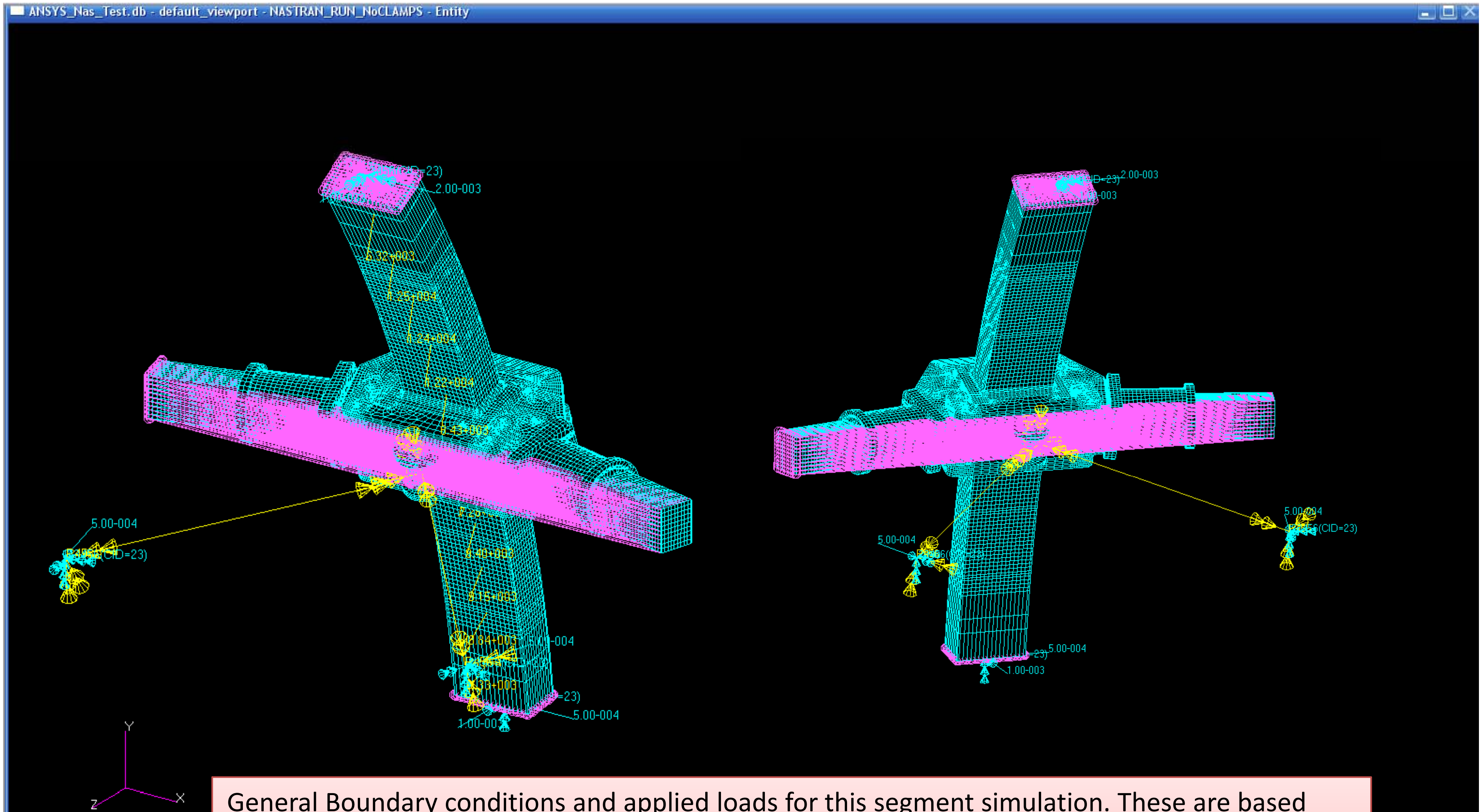
# <u>Where to find the information for this presentation</u>

Data for this simulation is in the "ANSYS_Nas_Test.db" which contains:

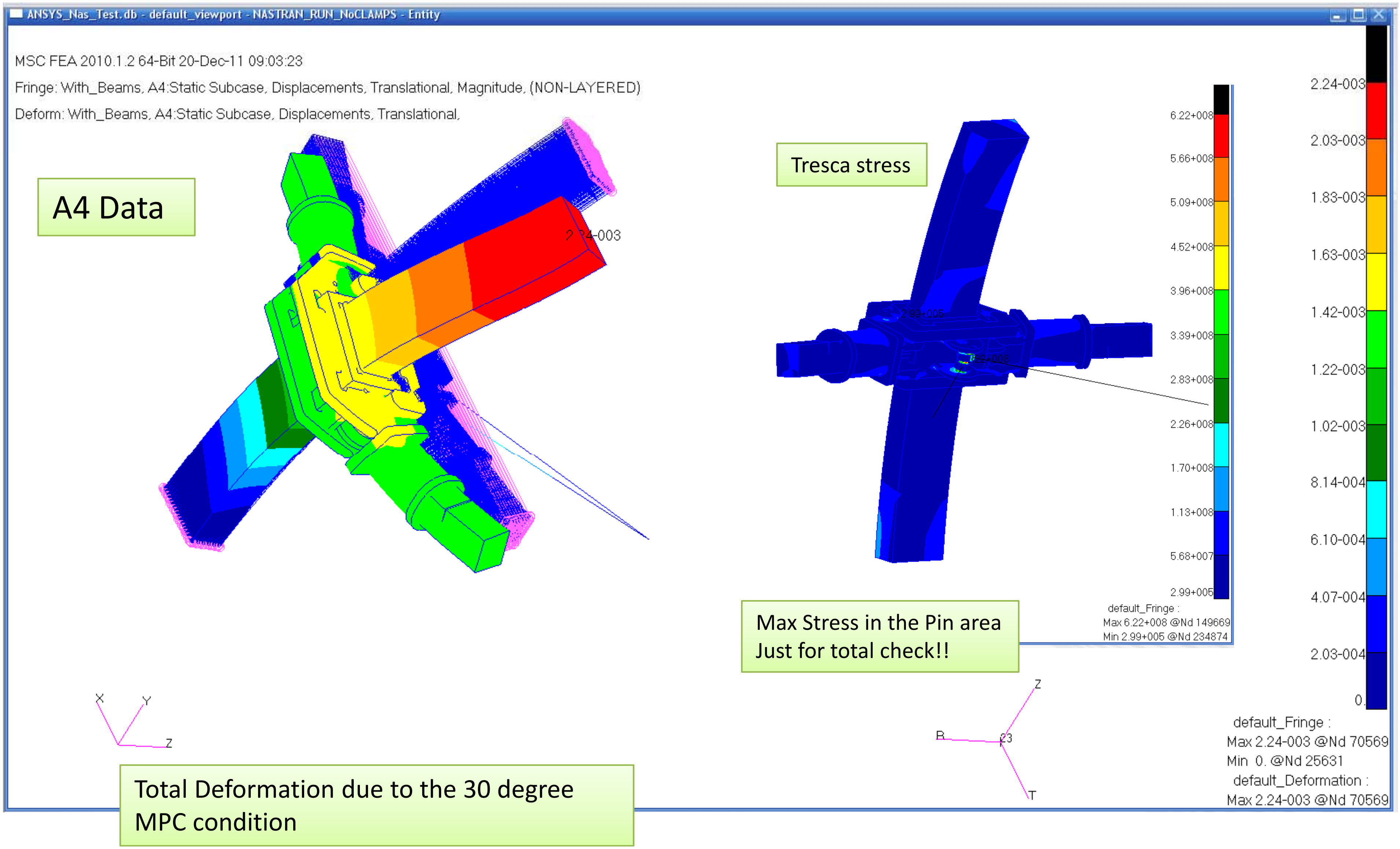
* Complete Nastran simulation,
* Input files (.bdf) for A4 and A7 loads and Boundary conditions,
* Attached results files (.xdb) for both loads.

P:\public\Snap-srv\progoff\Nastran P.R
Complete stress data simulation is located here.

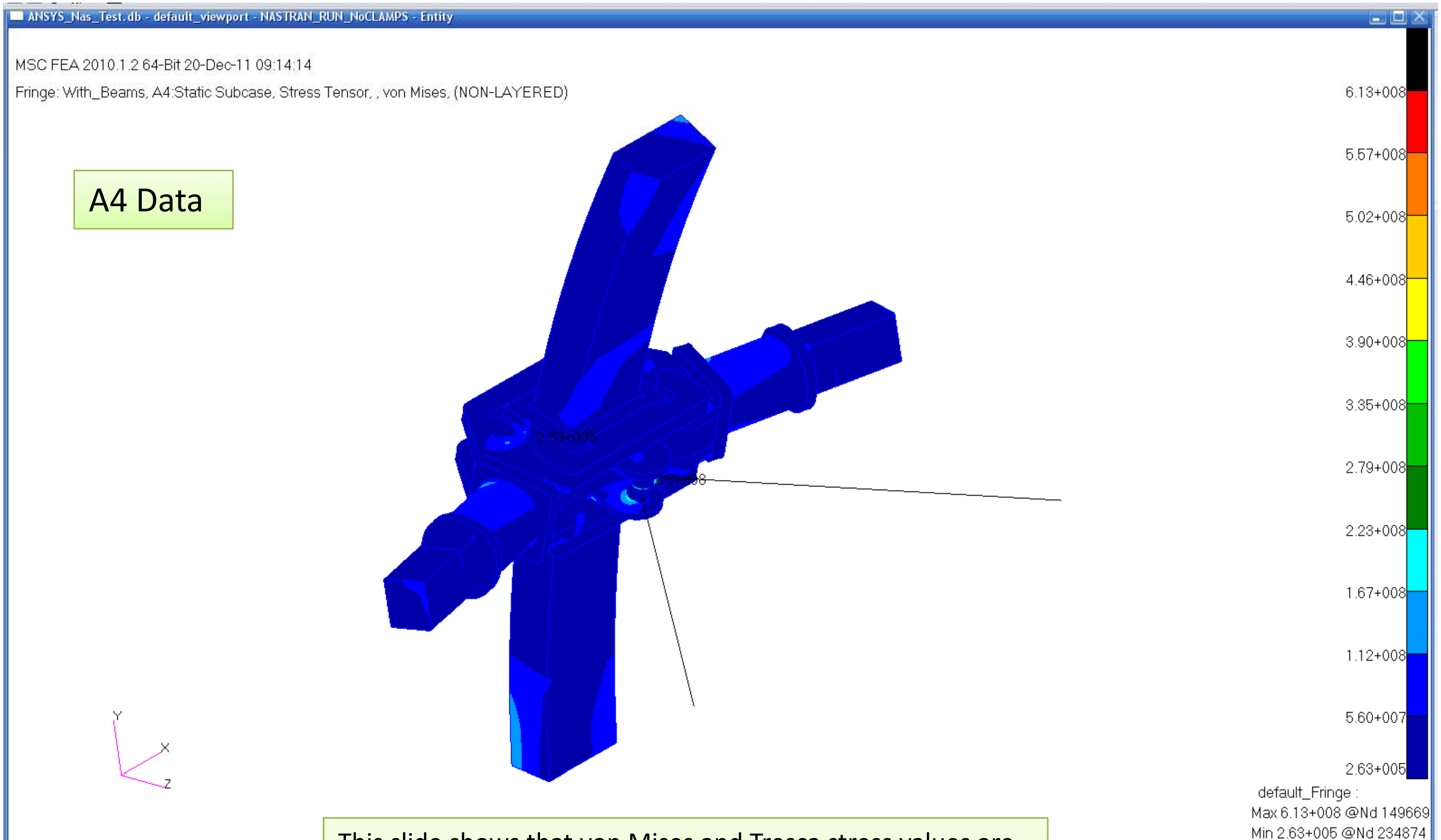
P:\public\Snap-srv\progoff\Nastran P.R\Coil_Clamp_Strain_Data
Complete calculation for the model strain is located here.




General Boundary conditions and applied loads for this segment simulation. These are based on the appropriate ANSYS global models. There are two enforced displacement scenarios at the VV clevis connections: For the A4 Static case, nodes connecting the link/struts are fully fixed in all six degrees of freedom, for the A7 Static case, the radial degree is displaced 0.5 mm to simulate an estimate of the VV compression due to the pressure profile. Based on these two loading conditions, stresses in the supporting pin could change.


ANSYS_Nas_Test.db - default_viewport - NASTRAN_RUN_NoCLAMPS - Entity
MSC FEA 2010.1.2 64-Bit 20-Dec-11 09:03:23
Fringe: With_Beams, A4:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: With_Beams, A4:Static Subcase, Displacements, Translational,
A4 Data
Tresca stress
6.22+008
5.66+008
5.09+008
4.52+008
3.96+008
3.39+008
2.83+008
2.26+008
1.70+008
1.13+008
5.68+007
2.99+005
default_Fringe :
Max 6.22+008 @Nd 149669
Min 2.99+005 @Nd 234874
Max Stress in the Pin area
Just for total check!!
2.24-003
2.03-003
1.83-003
1.63-003
1.42-003
1.22-003
1.02-003
8.14-004
6.10-004
4.07-004
2.03-004
0.
default_Fringe :
Max 2.24-003 @Nd 70569
Min 0. @Nd 25631
default_Deformation :
Max 2.24-003 @Nd 70569
Total Deformation due to the 30 degree
MPC condition

# A4 Data

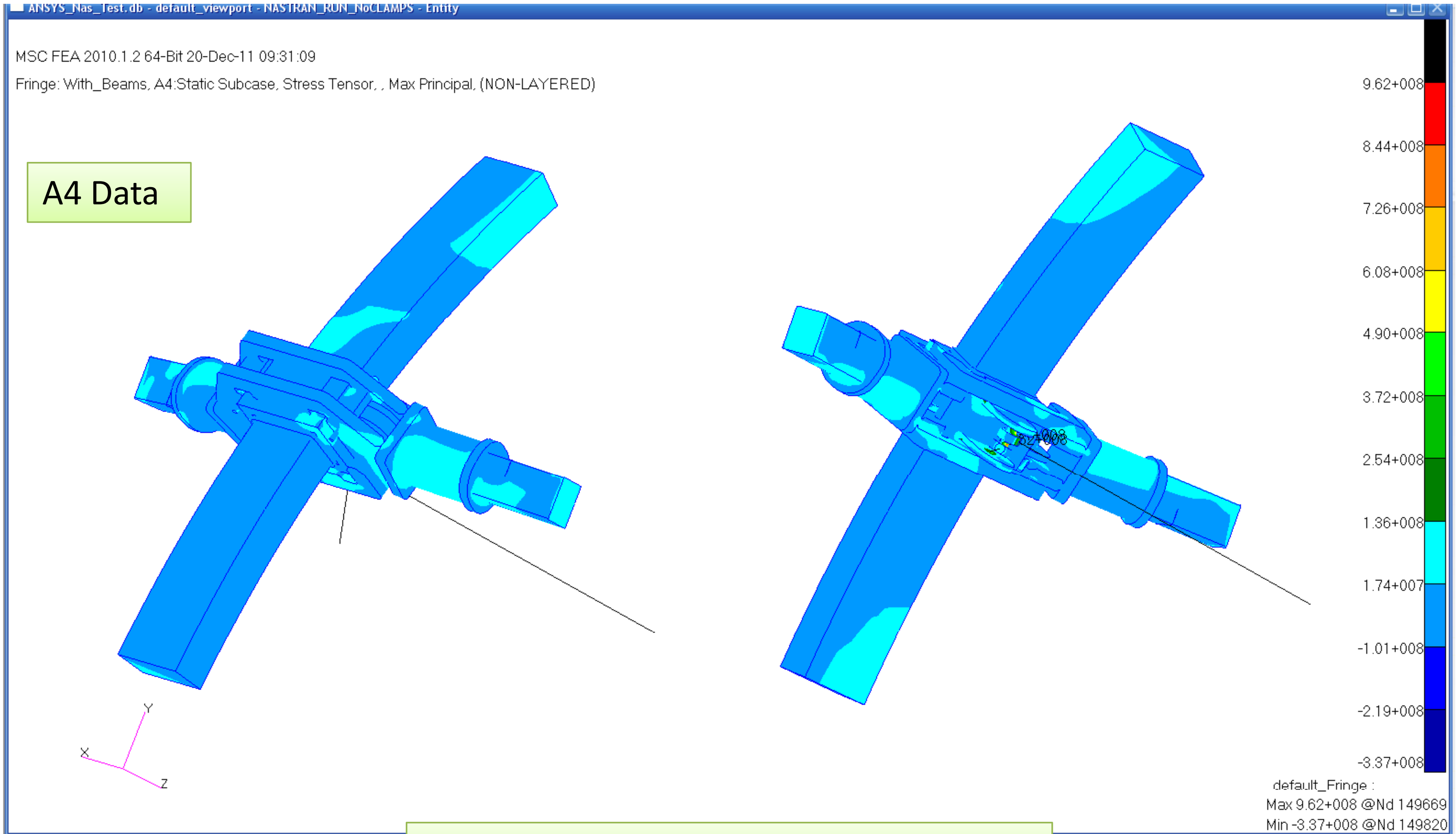



This slide shows that von Mises and Tresca stress values are comparable in the overall contour mode. Max at pin location.

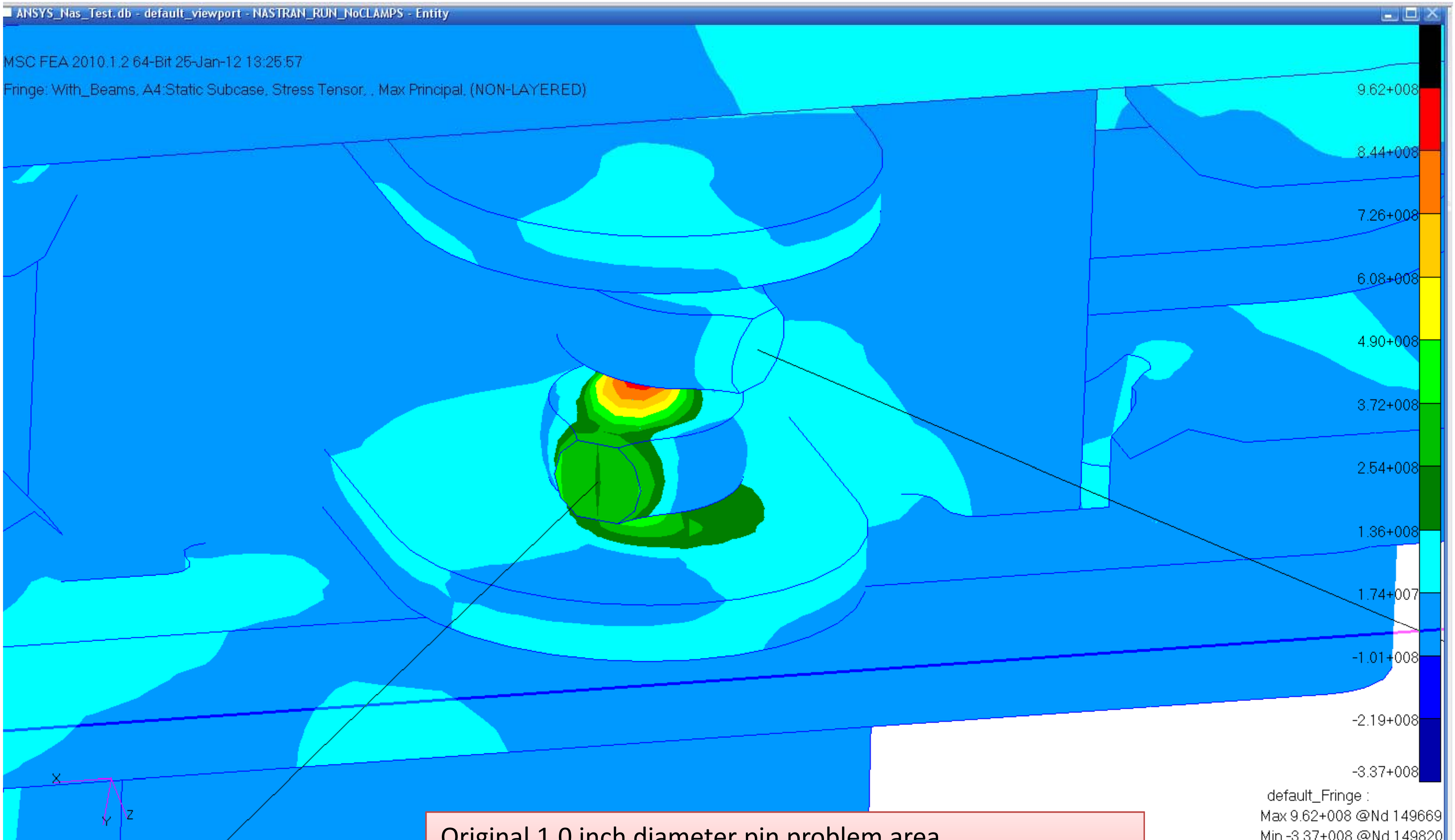
Tresca = 622Mpa

von Mises = 613Mpa

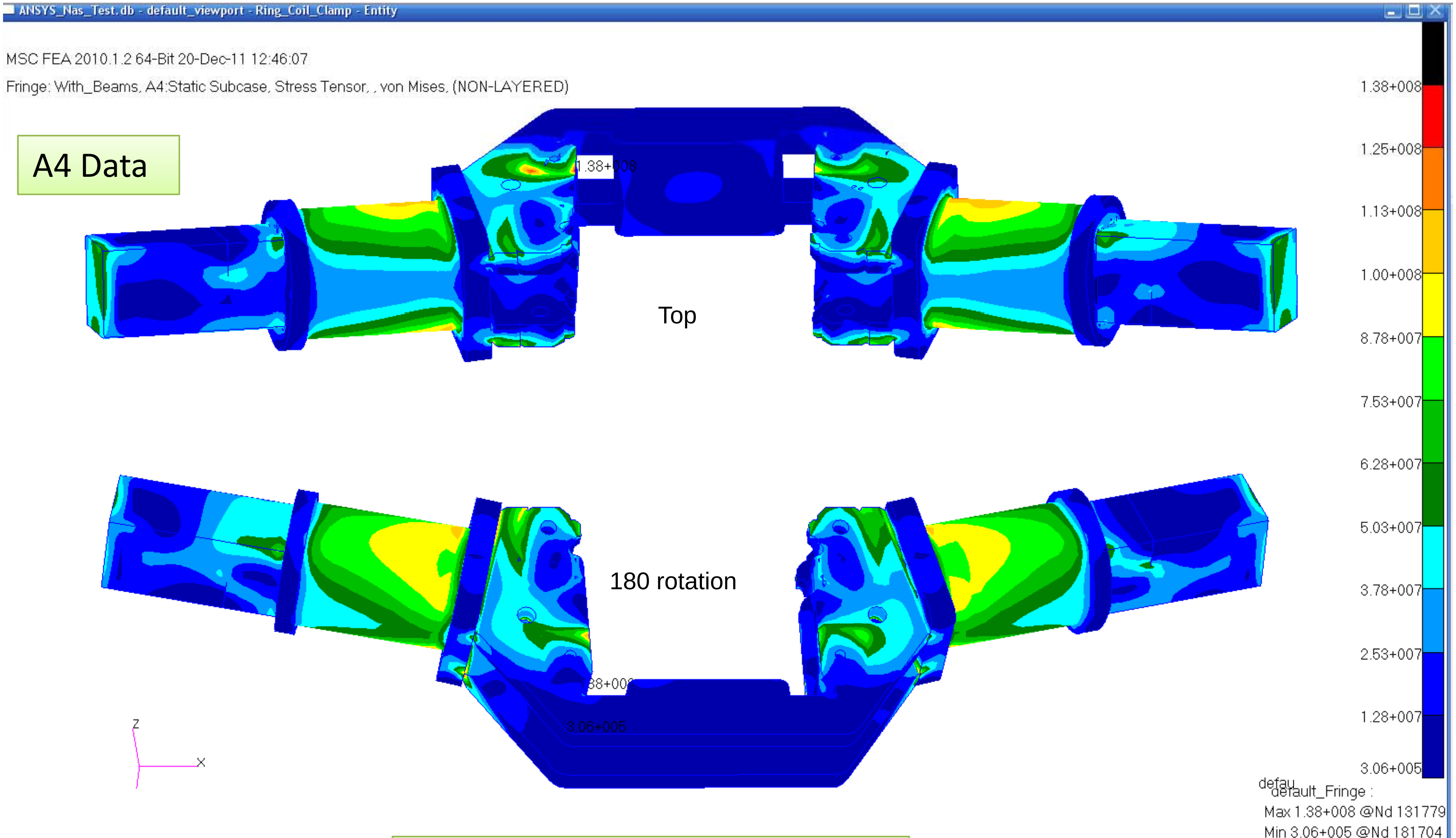
# A4 Data



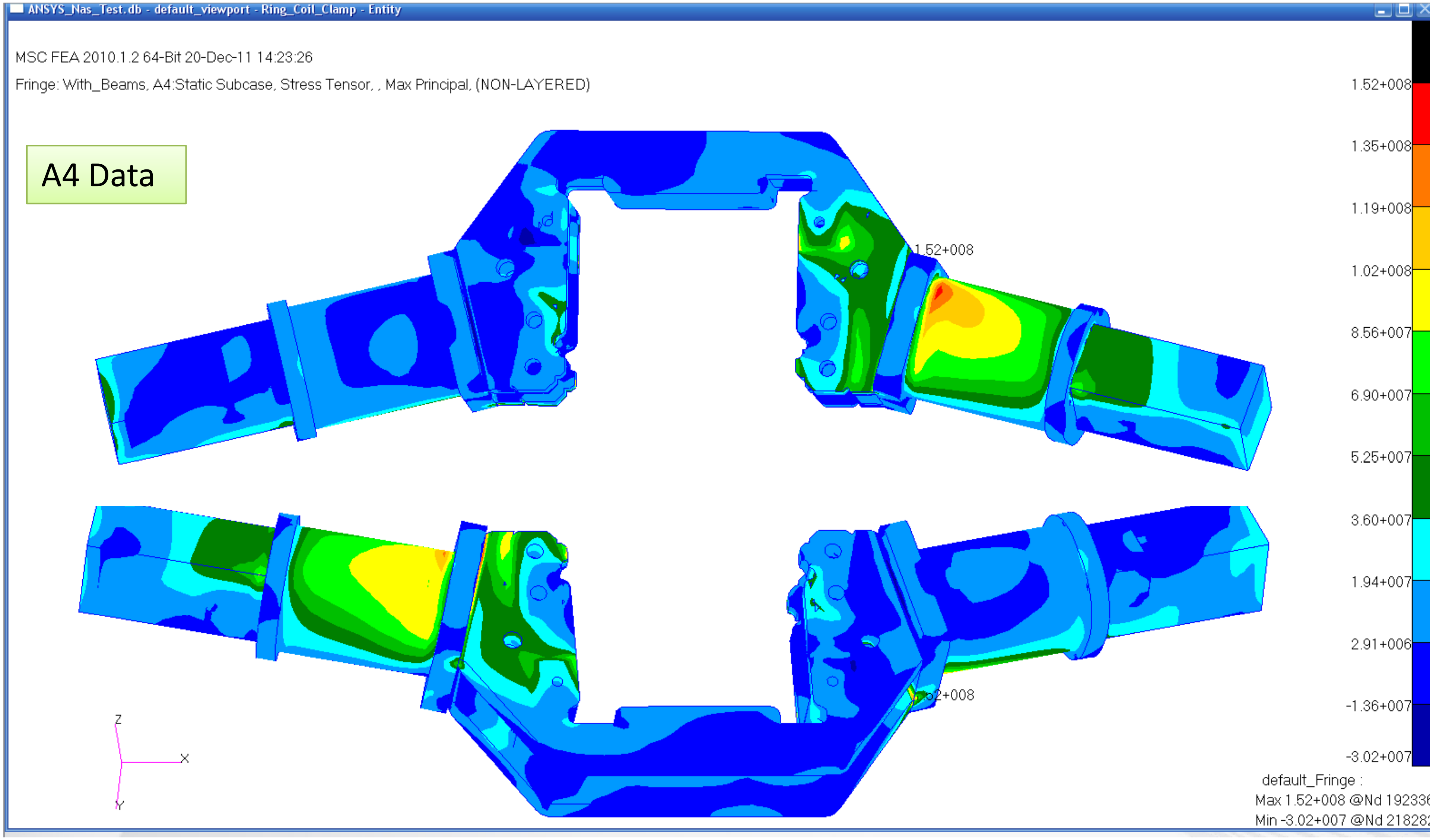

Total Max Principal back and front at pin location




Original 1.0 inch diameter pin problem area,
Was redesigned. See "New_TF-Clevis-Pin-9-28-2011PR.ppt" for complete explanation.

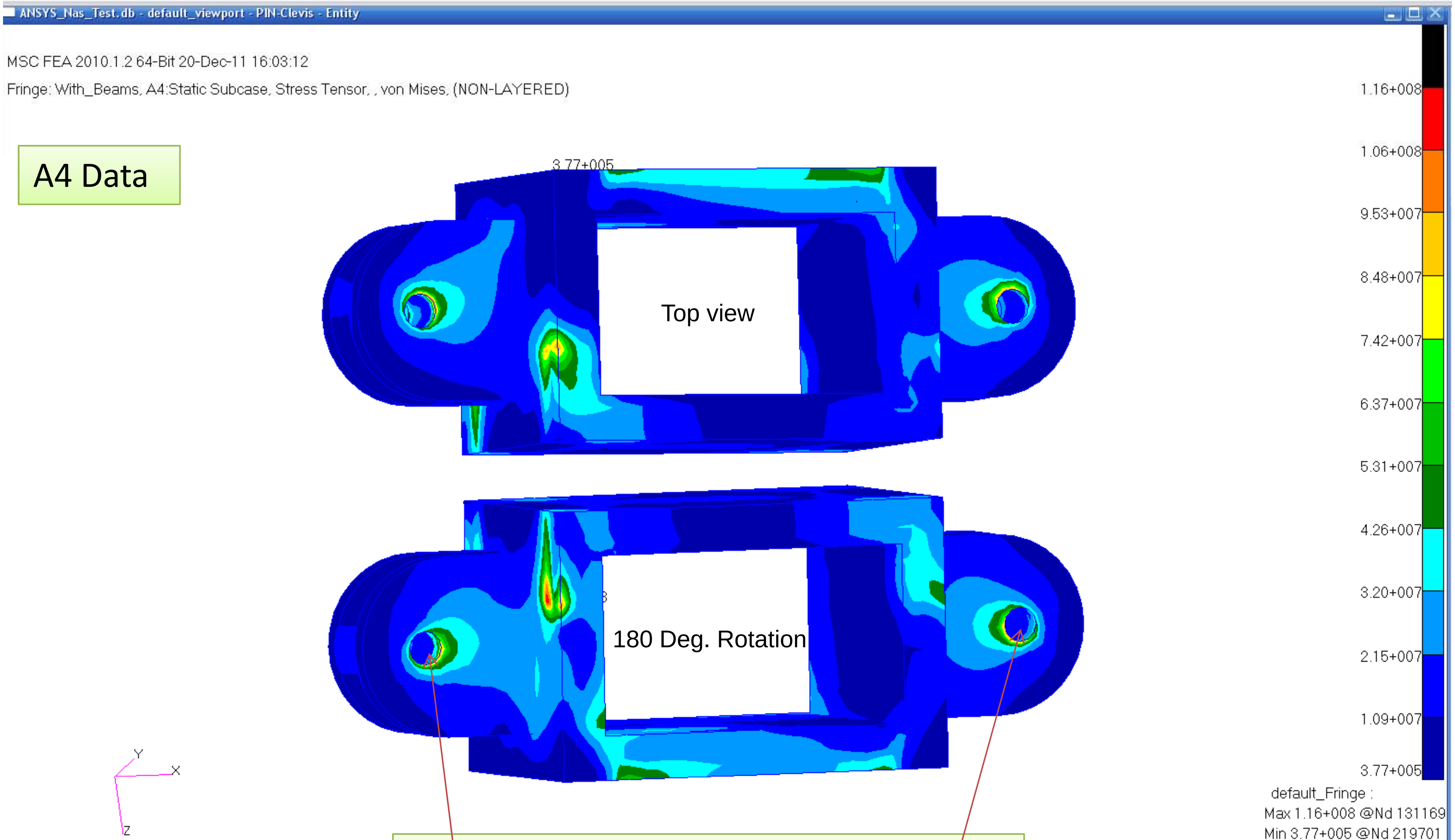



Max von Mises = 138Mpa = 20,015.psi



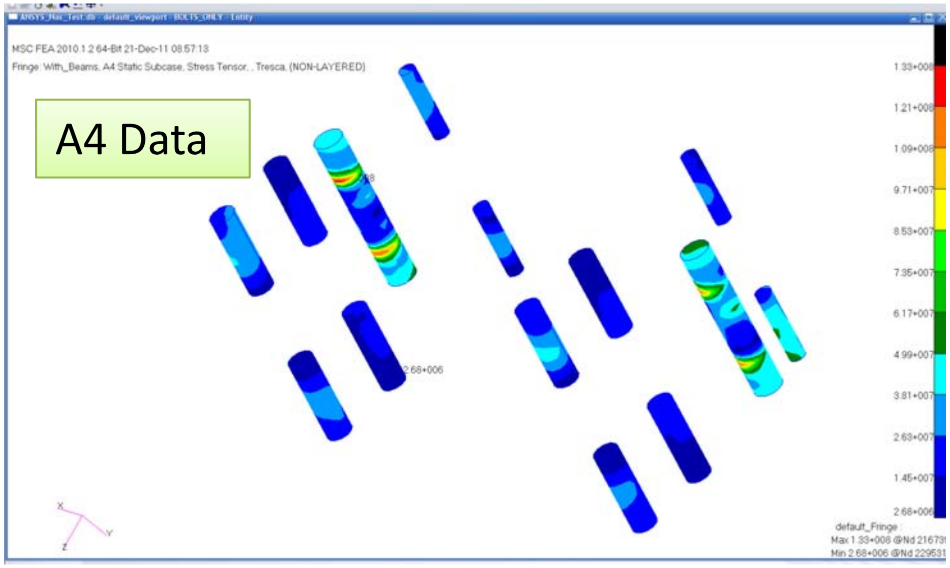

Max Principal = 152Mpa = 22,050.psi

# A4 Data

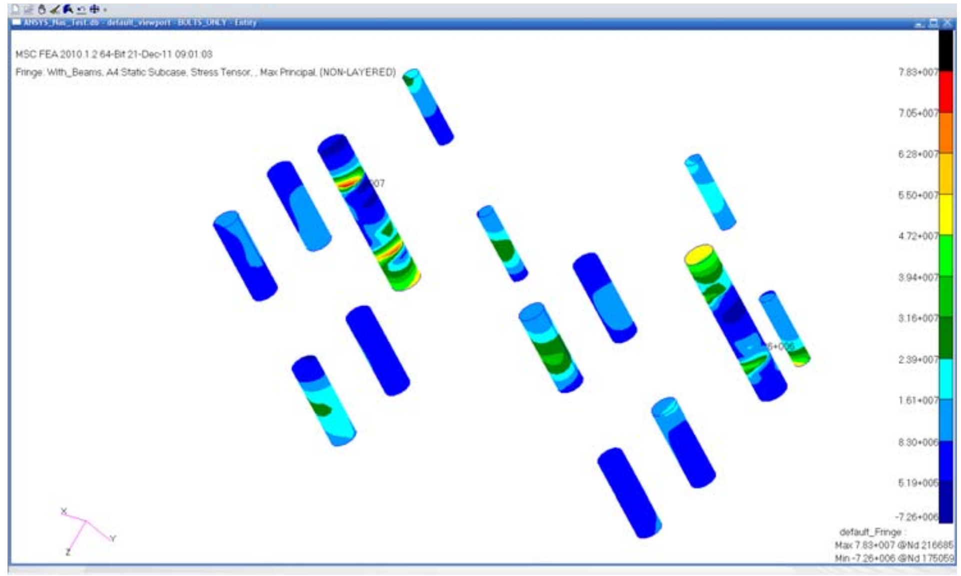



Important pin/bolt inserted here. See next slide.

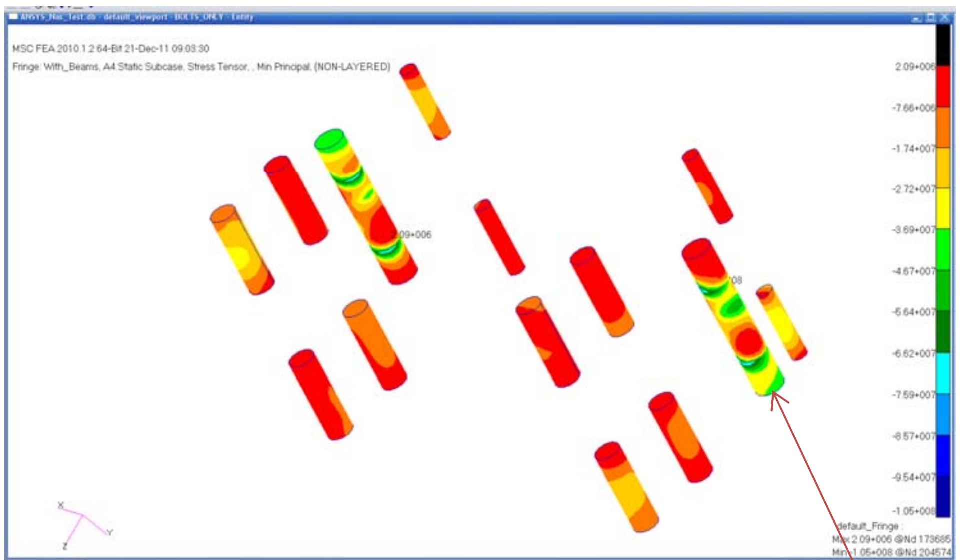
Max von Mises = 116Mpa = 16,830.psi



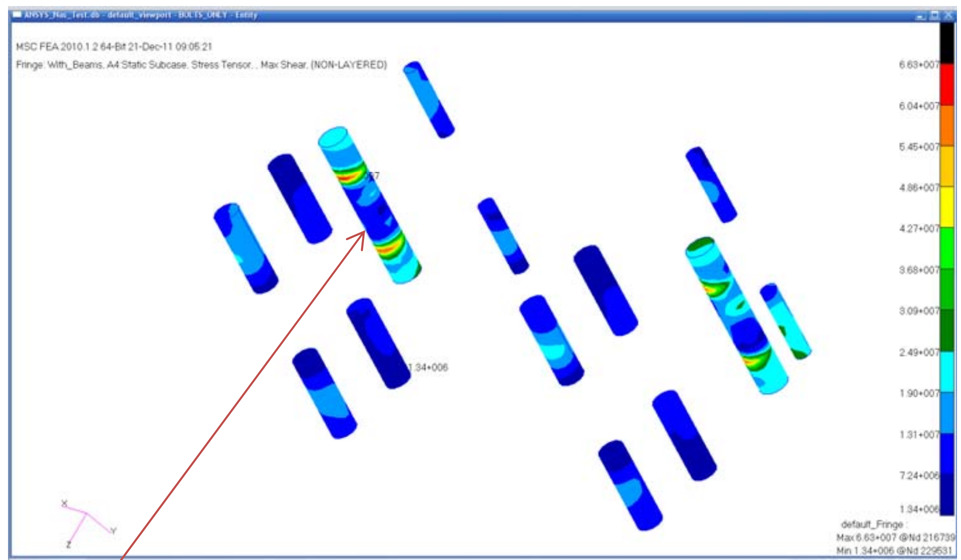

Tresca = 133Mpa = 19,300.psi




Max Principal = 78.3Mpa = 11,400.psi



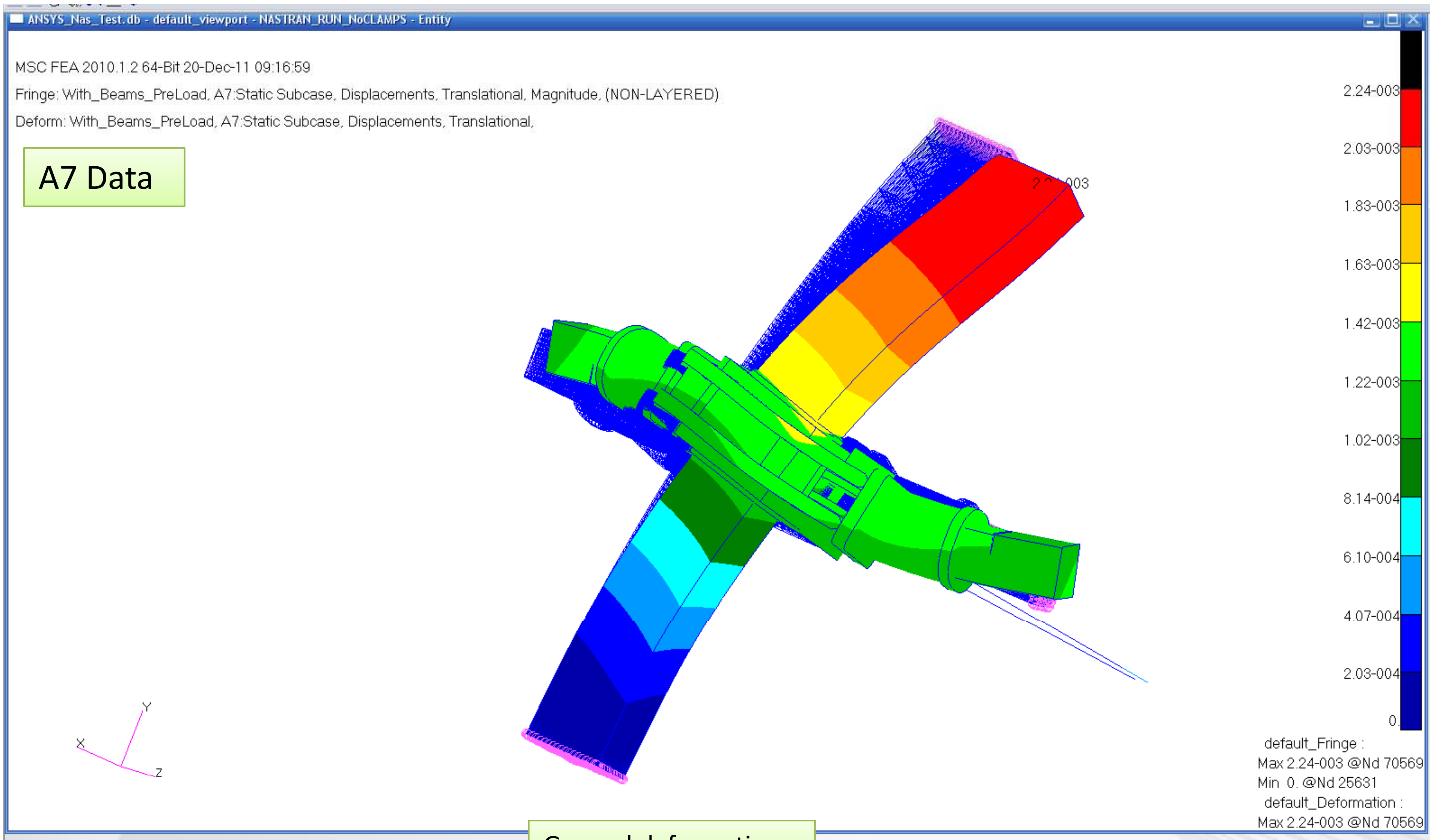

Min Principal = 105Mpa = 15,250.psi



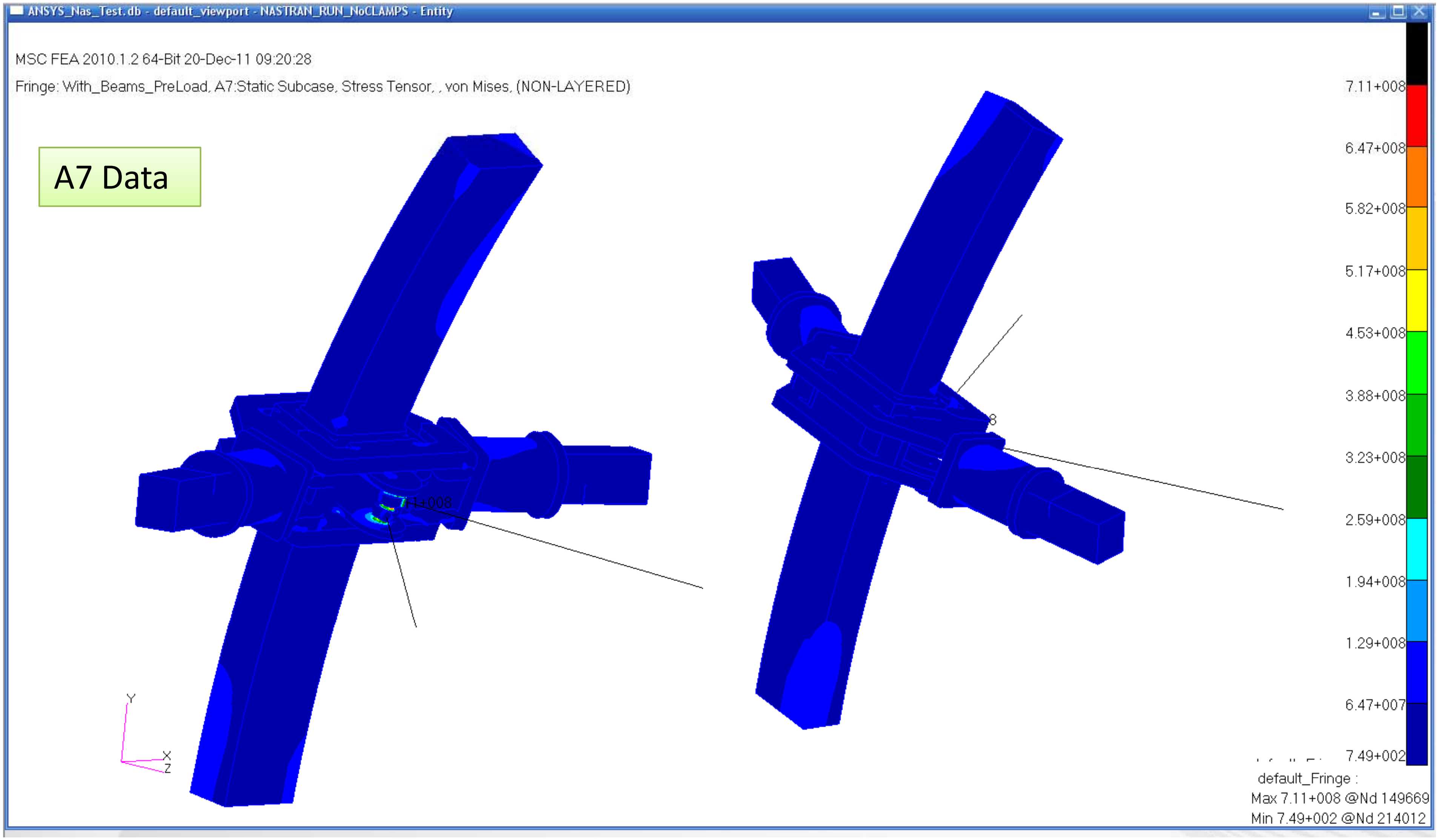

Max Shear = 66.3Mpa = 9,600.psi

Note: Main Pin/Bolts take most of the force transmission

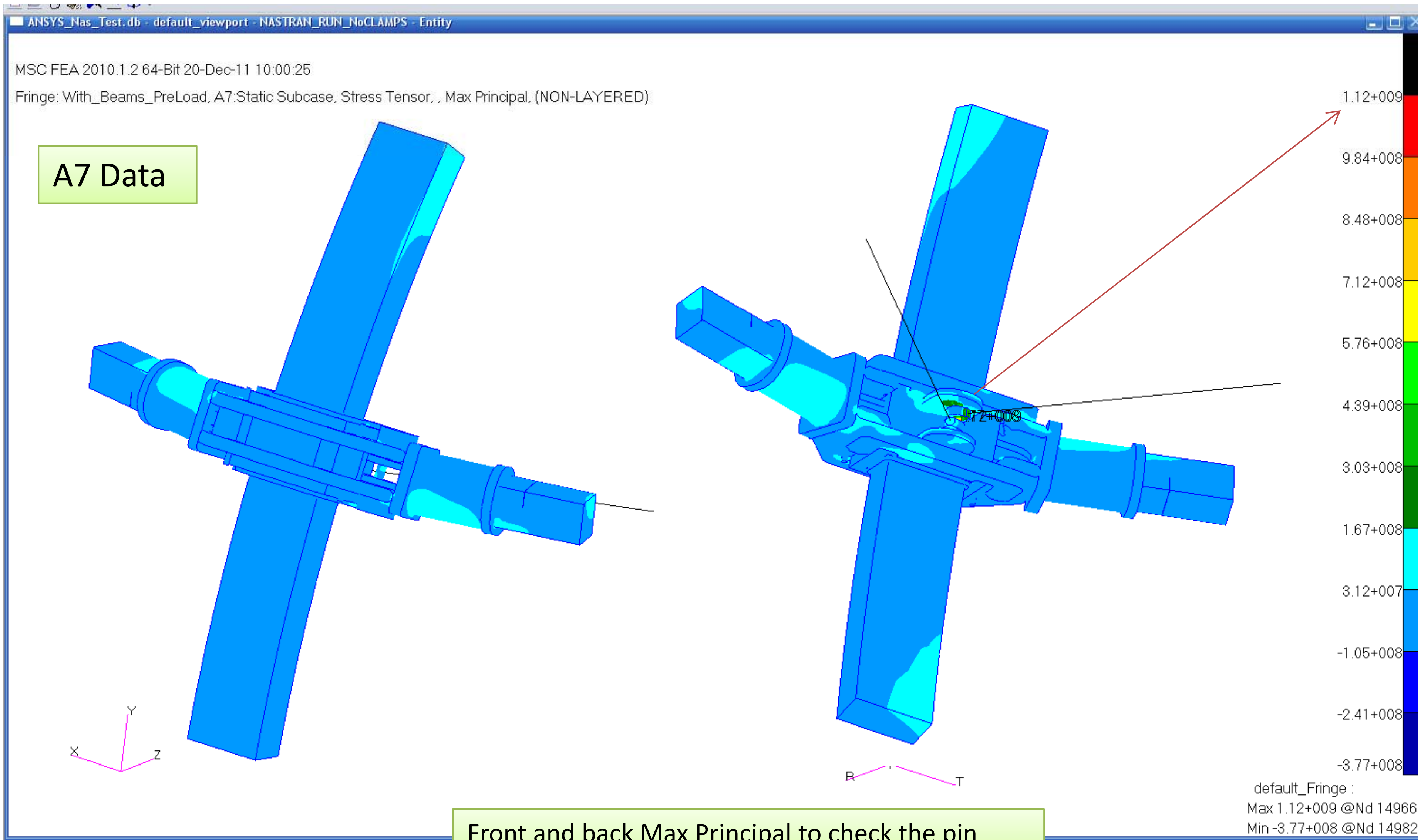
A7 Data




General deformation

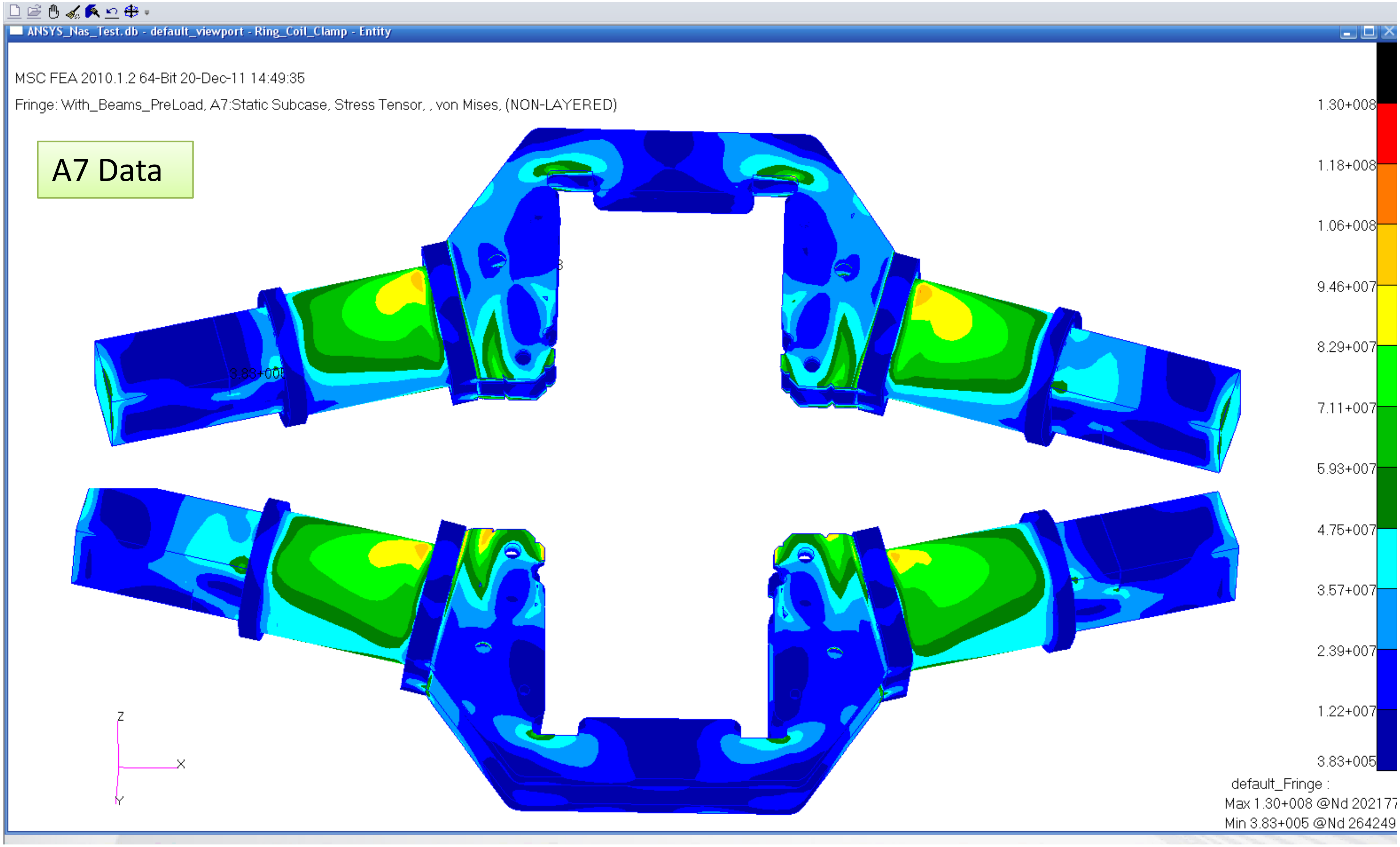

ANSYS_Nas_Test.db - default_viewport - NASTRAN_RUN_NoCLAMPS - Entity
MSC FEA 2010.1.2 64-Bit 20-Dec-11 09:20:28
Fringe: With_Beams_PreLoad, A7:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
A7 Data
7.11+008
6.47+008
5.82+008
5.17+008
4.53+008
3.88+008
3.23+008
2.59+008
1.94+008
1.29+008
6.47+007
7.49+002
default_Fringe :
Max 7.11+008 @Nd 149669
Min 7.49+002 @Nd 214012
Y
X
Z

# A7 Data

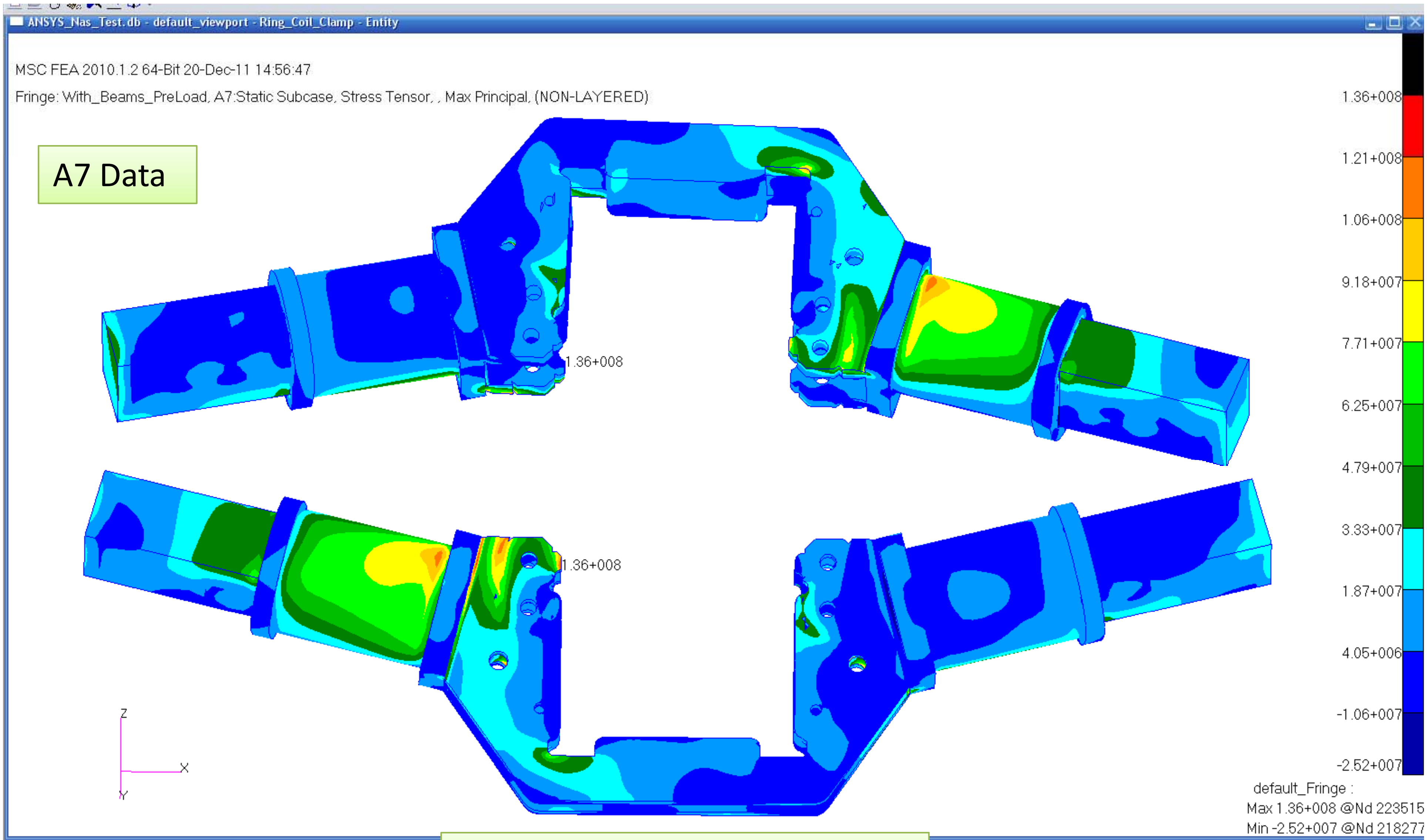



Front and back Max Principal to check the pin

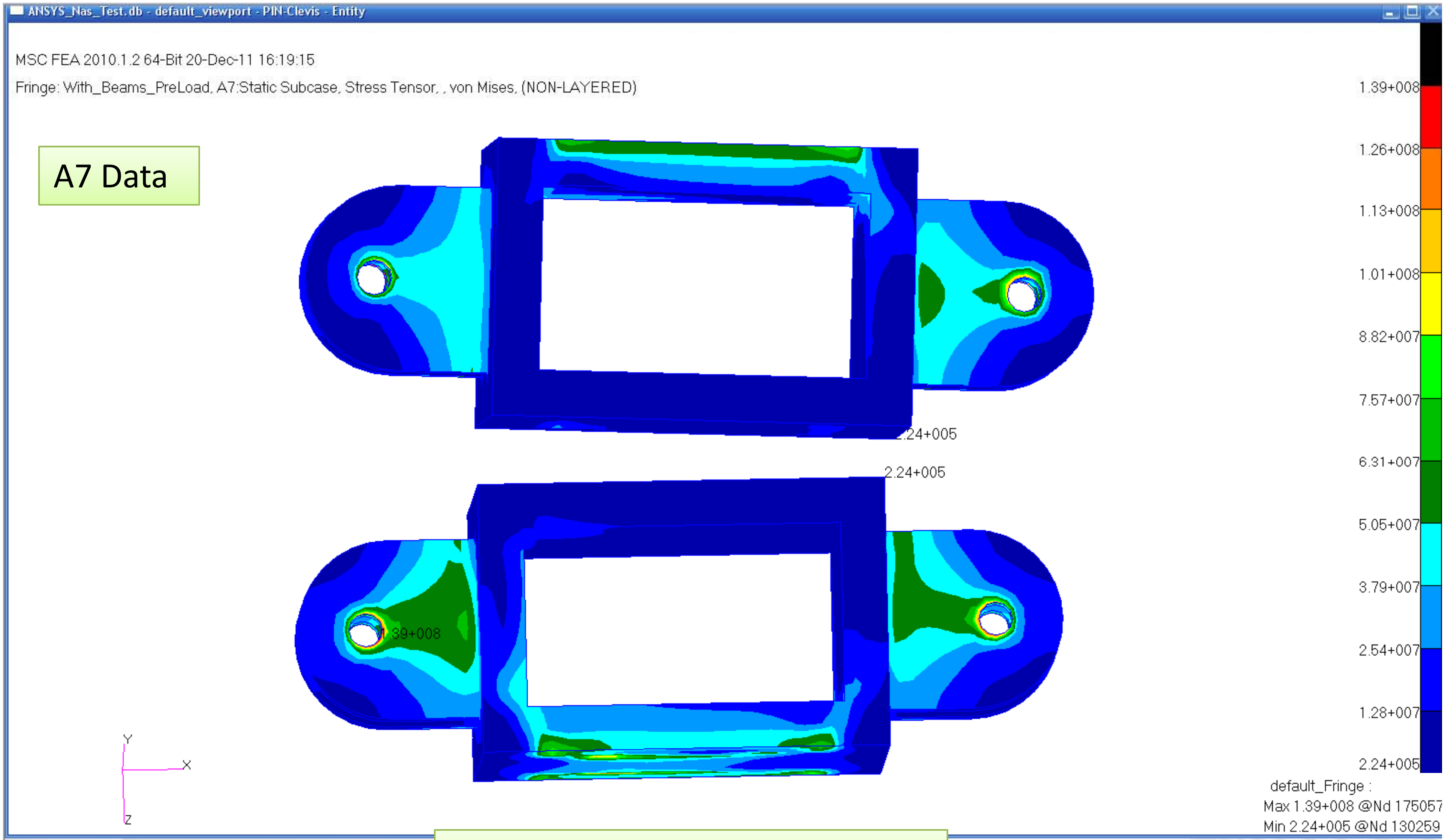
A7 Data



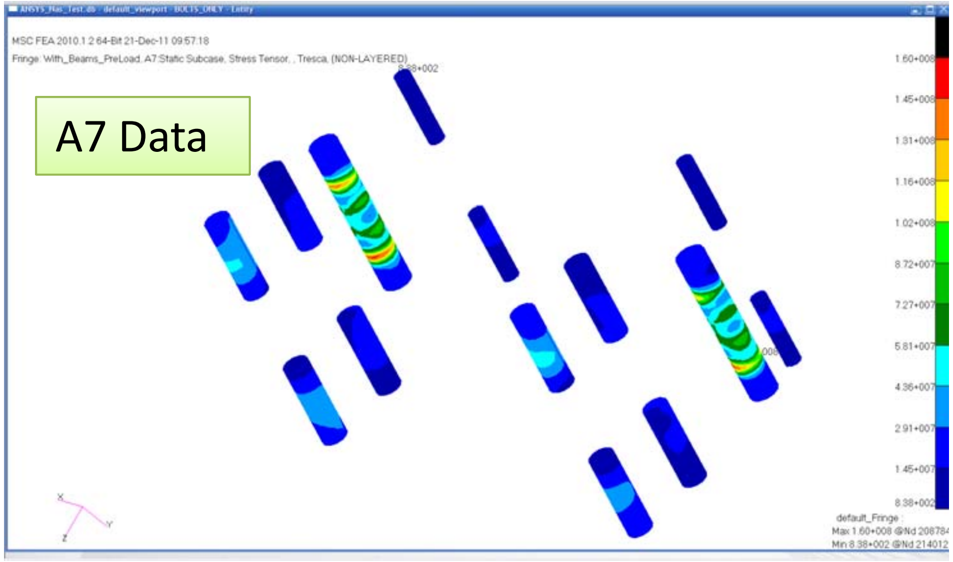

Max von Mises = 130Mpa = 18,855.psi



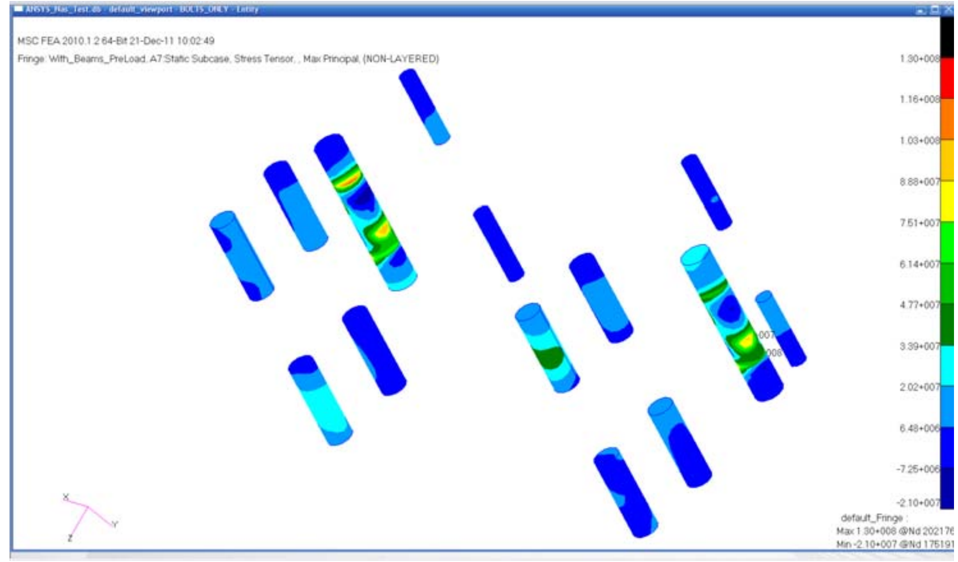

Max Principal = 136Mpa = 19,725.psi



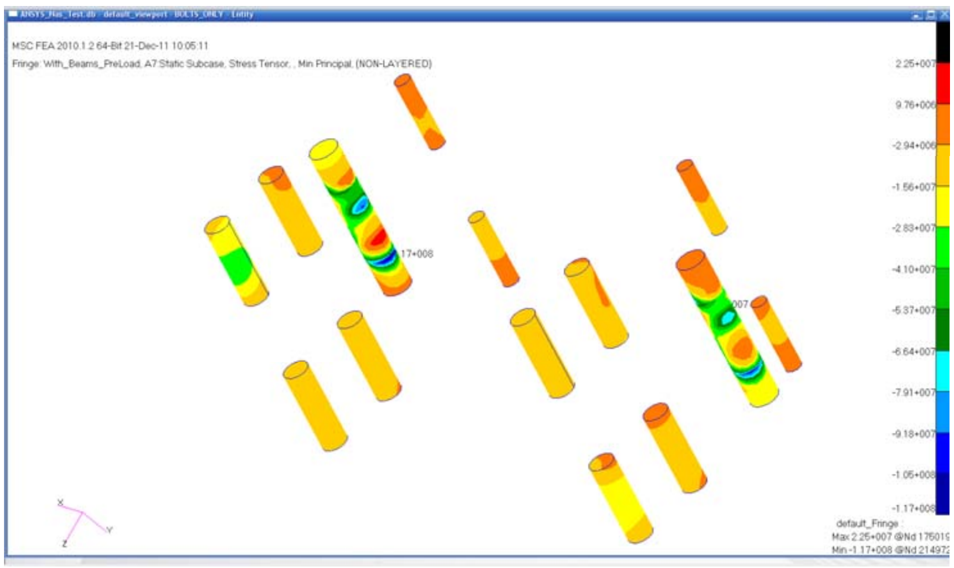

Max von Mises = 139Mpa = 20,160.psi



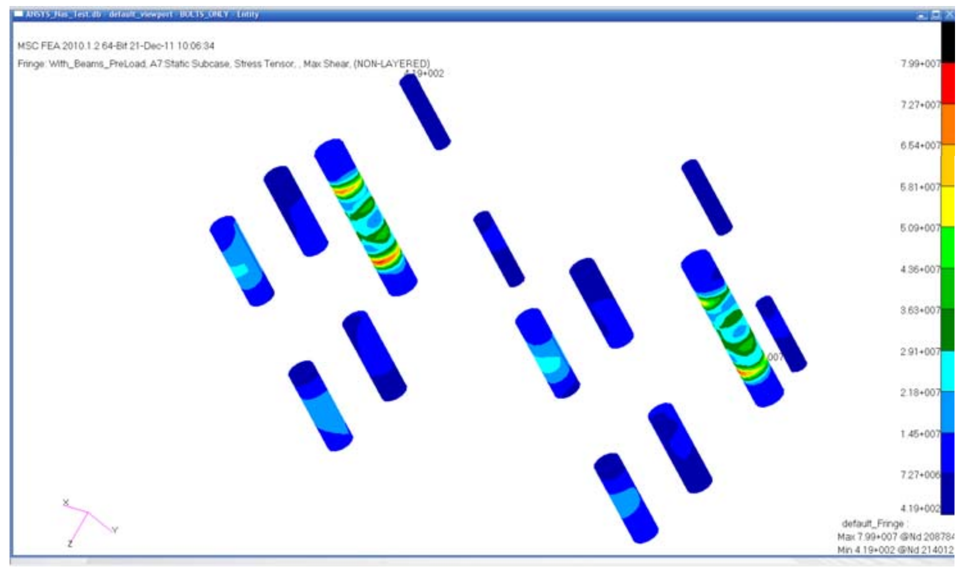

Tresca = 160Mpa = 23,200.psi




Max Principal = 130Mpa = 18,850.psi




Min Principal = 117Mpa = 16,970.psi




Max Shear = 79Mpa = 11,460.psi

## Fatigue calculations using Strain data

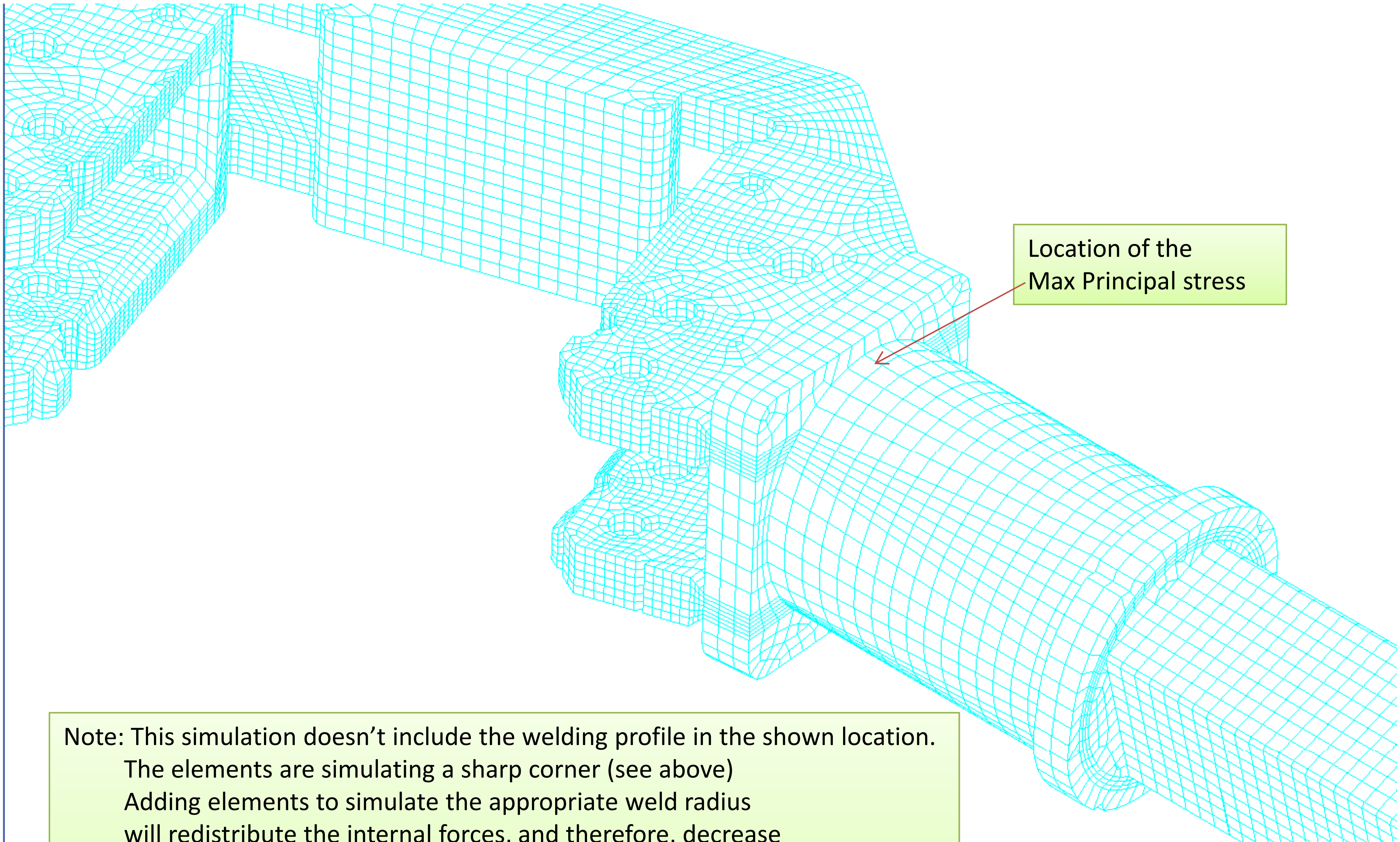
Explanation: -ITER Document No. G74 MA 16,
- File Code : ITER-AA04-2401,   Page 1
- Publication Package No: 8,

Contains Fatigue strain range data for Stainless 316L for various temperature references. This data is included in the slides which follow. This is the only useful data available at this time. Please see below.

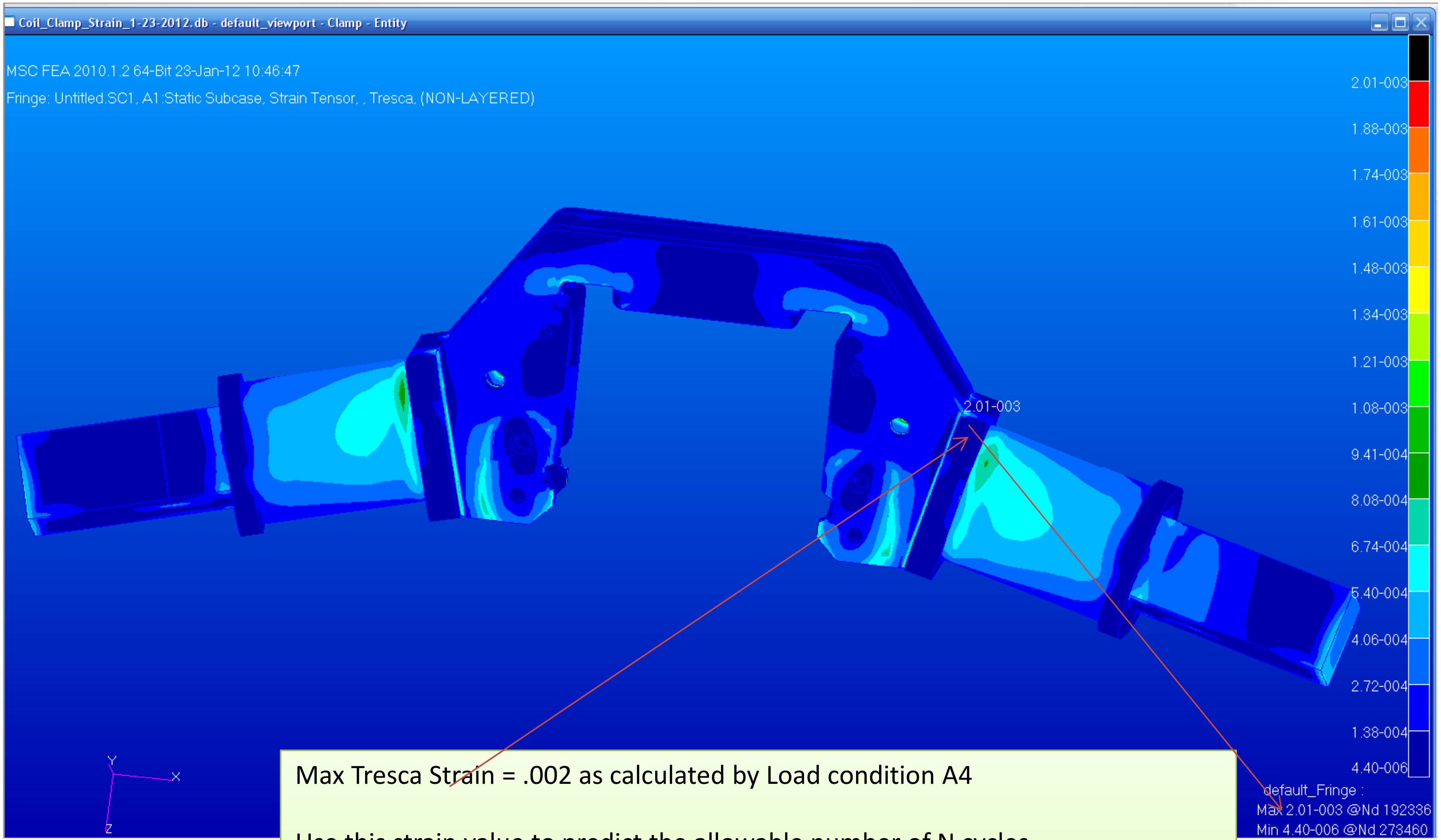
Data for the room temperature ( T = 20 deg C ) was used for this calculation.

Therefore: for max strain = .002 based on ITER data for SS316L N=1,000,000. cycles
Even at N/20 = 50,000 cycles at the maximum load conditions, should be satisfactory.
This is very conservative since, the FEA model doesn't simulate the welding profiles. The section contacts are purely perpendicular.

Note: Loads for these simulations are for R=0 environment




Note: This simulation doesn't include the welding profile in the shown location.
The elements are simulating a sharp corner (see above)
Adding elements to simulate the appropriate weld radius will redistribute the internal forces, and therefore, decrease (redistribute) stress concentrations. This shows that the calculated stresses are probably conservative! Considering both load conditions obtained max principal stress = 152.Mpa (Use this value for the fatigue/life calculations.




Max Tresca Strain = .002 as calculated by Load condition A4

Use this strain value to predict the allowable number of N cycles.
This strain value is highly concentrated at a single nodal point which
Is connected to a single "CHexa" element. This result is felt to be artificially high.
All other values are less than 0.0011. If and when the welds are simulated,
this is certain to disappear. Even using the strain value = .002 gives reasonable
life expectancy.

**Table 1/2008. Recommended Fatigue strain range εt (%) as a function of temperature and number of allowable cycles ($N_d$), RCC-MR, Edition 2007.**

| Allowable Number of cycles | Temperature, °C | | | | | |
|---|---|---|---|---|---|---|
| Nd | 20 | 450 | 500 | 550 | 600 | 650 |
| 10 | 4.291 | 2.552 | 2.459 | 2.361 | 2.260 | 2.155 |
| 20 | 2.755 | 1.931 | 1.841 | 1.748 | 1.652 | 1.553 |
| 40 | 1.931 | 1.485 | 1.403 | 1.316 | 1.231 | 1.139 |
| $10^2$ | 1.331 | 1.080 | 1.007 | 0.933 | 0.859 | 0.787 |
| $2.10^2$ | 1.057 | 0.869 | 0.805 | 0.741 | 0.678 | 0.618 |
| $4.10^2$ | 0.863 | 0.715 | 0.659 | 0.605 | 0.552 | 0.502 |
| $10^3$ | 0.698 | 0.578 | 0.532 | 0.487 | 0.445 | 0.404 |
| $2.\ 10^3$ | 0.590 | 0.488 | 0.449 | 0.412 | 0.376 | 0.342 |
| $4.\ 10^3$ | 0.498 | 0.412 | 0.380 | 0.348 | 0.317 | 0.289 |
| $10^4$ | 0.399 | 0.330 | 0.304 | 0.278 | 0.254 | 0.231 |
| $2.10^4$ | 0.341 | 0.282 | 0.260 | 0.238 | 0.217 | 0.198 |
| $4.\ 10^4$ | 0.301 | 0.249 | 0.229 | 0.210 | 0.192 | 0.174 |
| $10^5$ | 0.267 | 0.221 | 0.203 | 0.186 | 0.17 | 0.154 |
| $2.\ 10^5$ | 0.250 | 0.207 | 0.190 | 0.174 | 0.159 | 0.145 |
| $4.\ 10^5$ | 0.238 | 0.197 | 0.181 | 0.166 | 0.151 | 0.138 |
| $10^6$ | 0.225 | 0.186 | 0.171 | 0.157 | 0.143 | 0.130 |
| $5.\ 10^6$ | 0.203 | 0.168 | 0.154 | 0.141 | 0.129 | 0.117 |
| $10^7$ | 0.192 | 0.159 | 0.146 | 0.134 | 0.122 | 0.111 |
| $5.\ 10^7$ | 0.169 | 0.140 | 0.129 | 0.118 | 0.108 | 0.098 |
| $10^8$ | 0.165 | 0.136 | 0.125 | 0.115 | 0.105 | 0.095 |

**Figure 1/2008 shows the comparison of the fatigue design curves of 2002 and 2007 Editions of RCC-MR.**

[1] Design and Construction Rules for Mechanical Components of Nuclear Installation, RCC-MR, Section 1, Subsection Z, Appendix A3: Characteristics of Materials, page A3.1S/17; French Association for the Design, Construction and Operating Supervision of the Equipment for Electro-Nuclear boilers (AFCEN), Edition 2007.

Data based on R=-1 tests

# ITER MATERIAL PROPERTIES HANDBOOK

| MATERIAL<br>TYPE 316L(N)-IG STAINLESS STEEL | PROPERTY<br>FATIGUE – STRAIN CONSTANT |
|---|---|

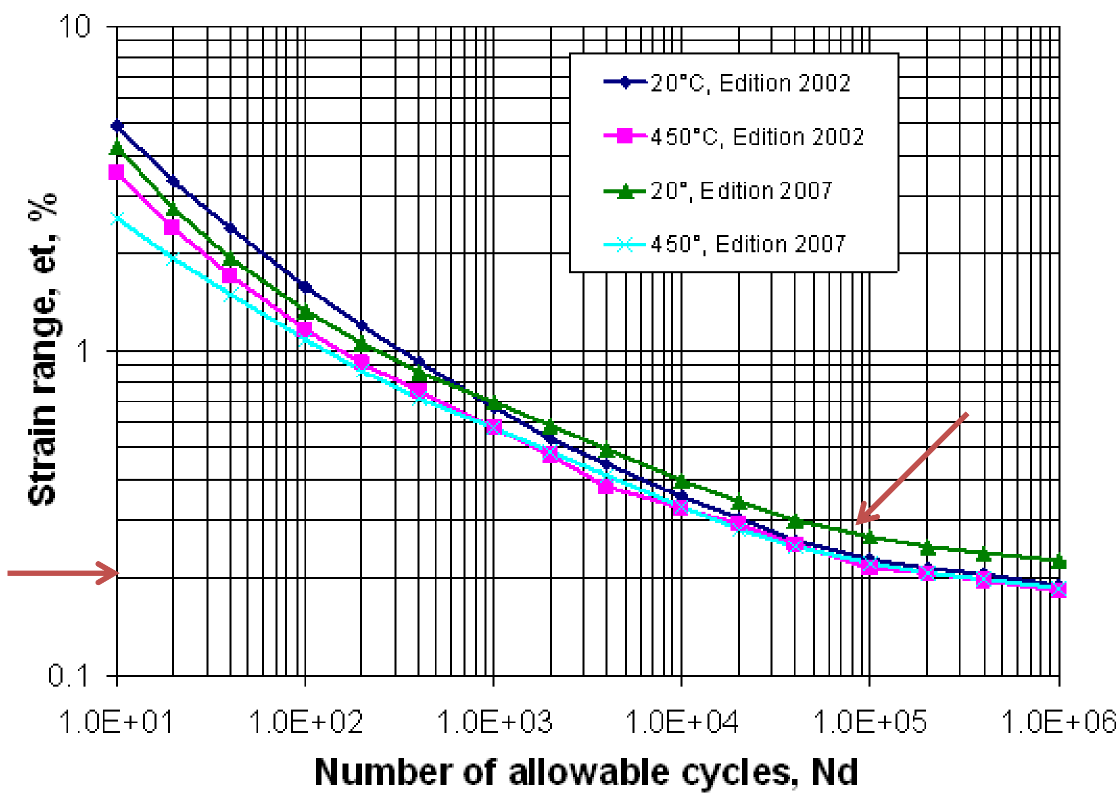


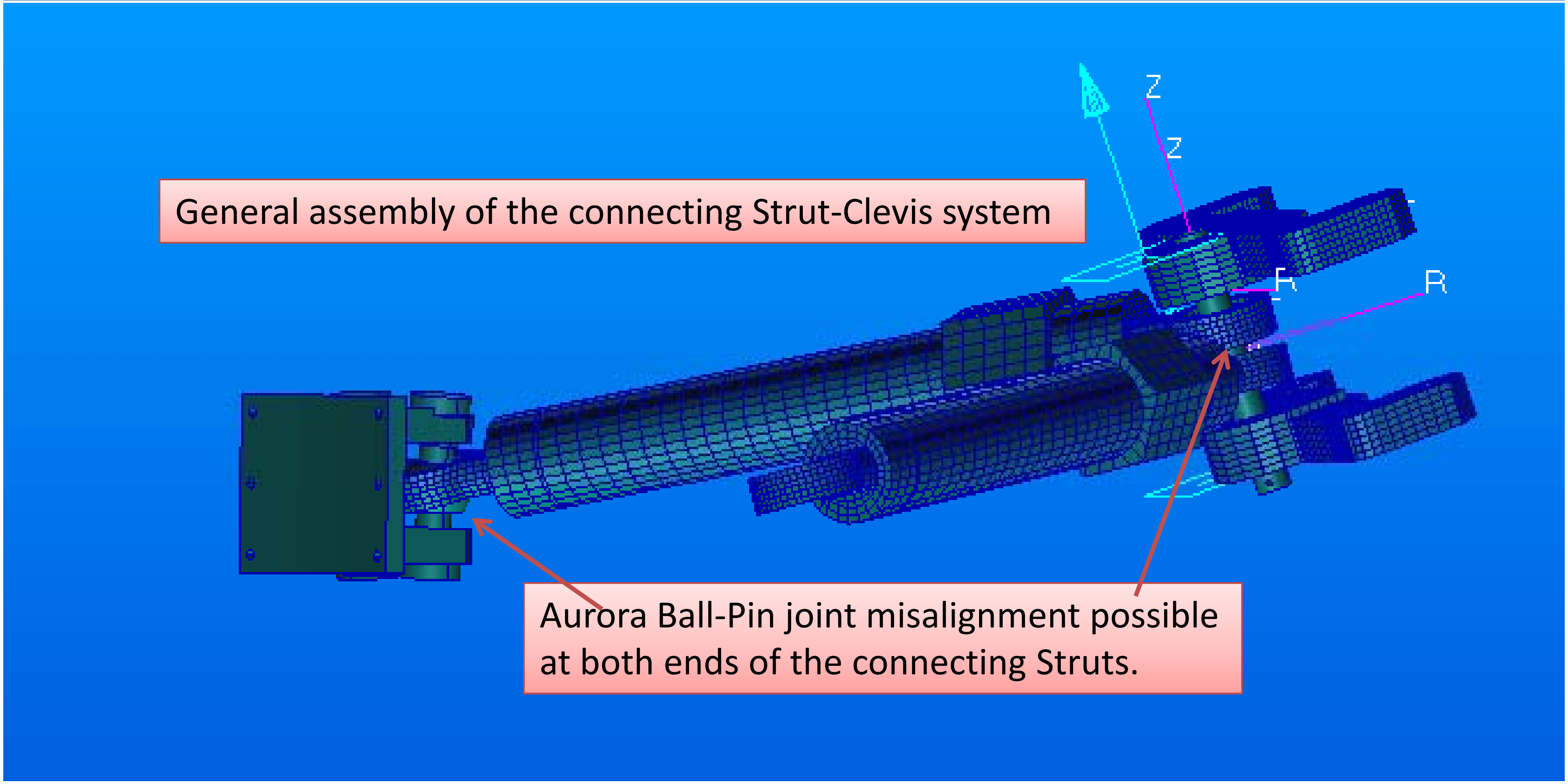

**Figure 1/2008 Design curves for fatigue.**
**Recommended are data from RCC-MR Edition 2007.**

New_TF-Clevis-Pin-9-28-2011PR.ppt

NSTXU Project

# TF Coil to VV connecting system configuration analyses

P.R. 11/28/2011

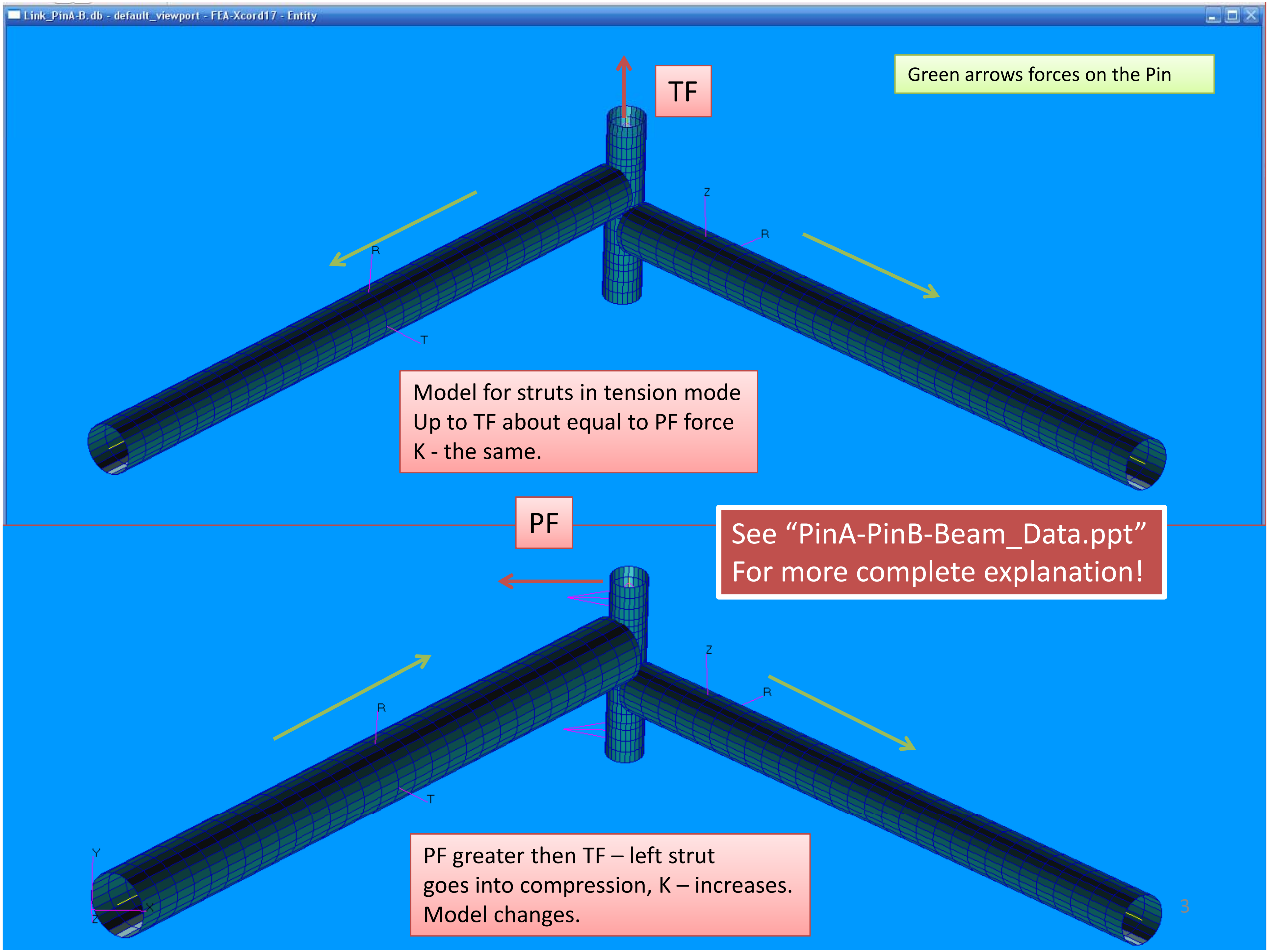


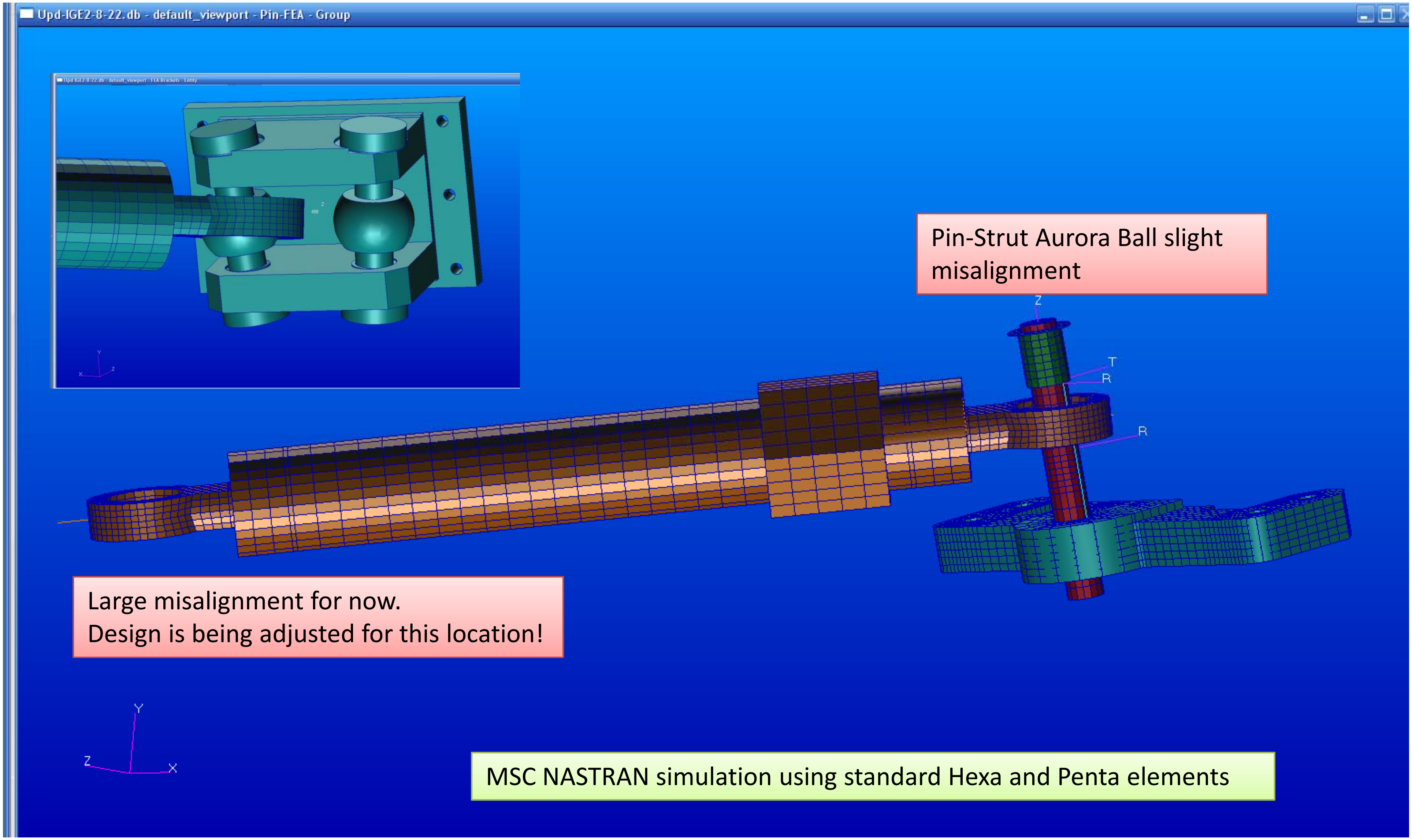

The principal stress analysis simulation was concentrated in predicting integrity of the coil clamp clevis pin. Several Global ANSYS models simulations predict forces (generated by the struts) on pin as: For the strut resisting total coil structure rotation the force is compressive, while the opposing strut is in tension. Therefore, the pin must be designed accordingly.

Originally, the pin diameter was set 1.0 inches but, using the provided forces, it was increased to 1.25 inches. Subsequent slides will demonstrate this!

Compressive force = 15,000. lbs.

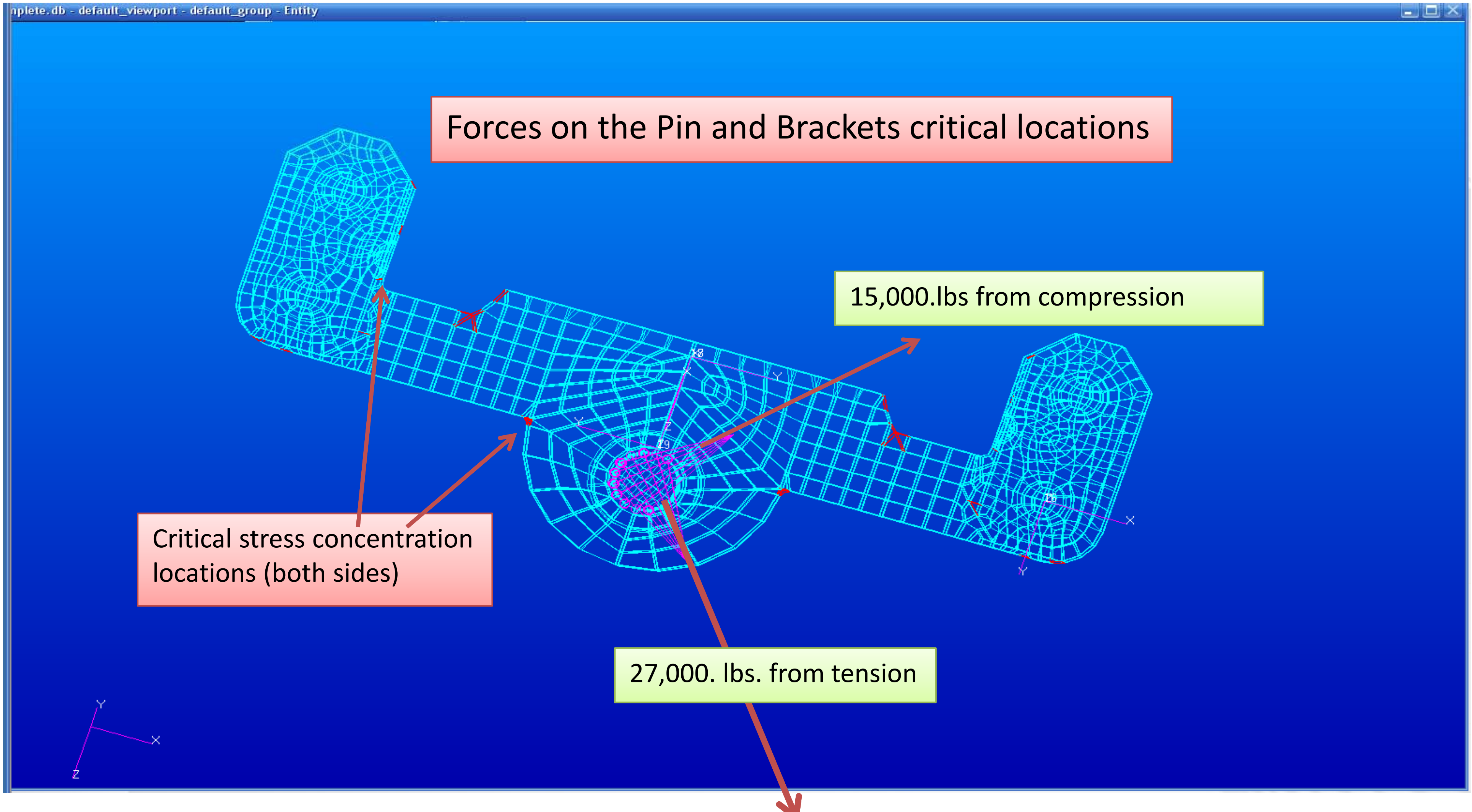
Tension force = 27,000. lbs.


Link_PinA-B.db - default_viewport - FEA-Xcord17 - Entity
TF
Green arrows forces on the Pin
Model for struts in tension mode
Up to TF about equal to PF force
K - the same.
PF
See "PinA-PinB-Beam_Data.ppt"
For more complete explanation!
PF greater then TF – left strut
goes into compression, K – increases.
Model changes.

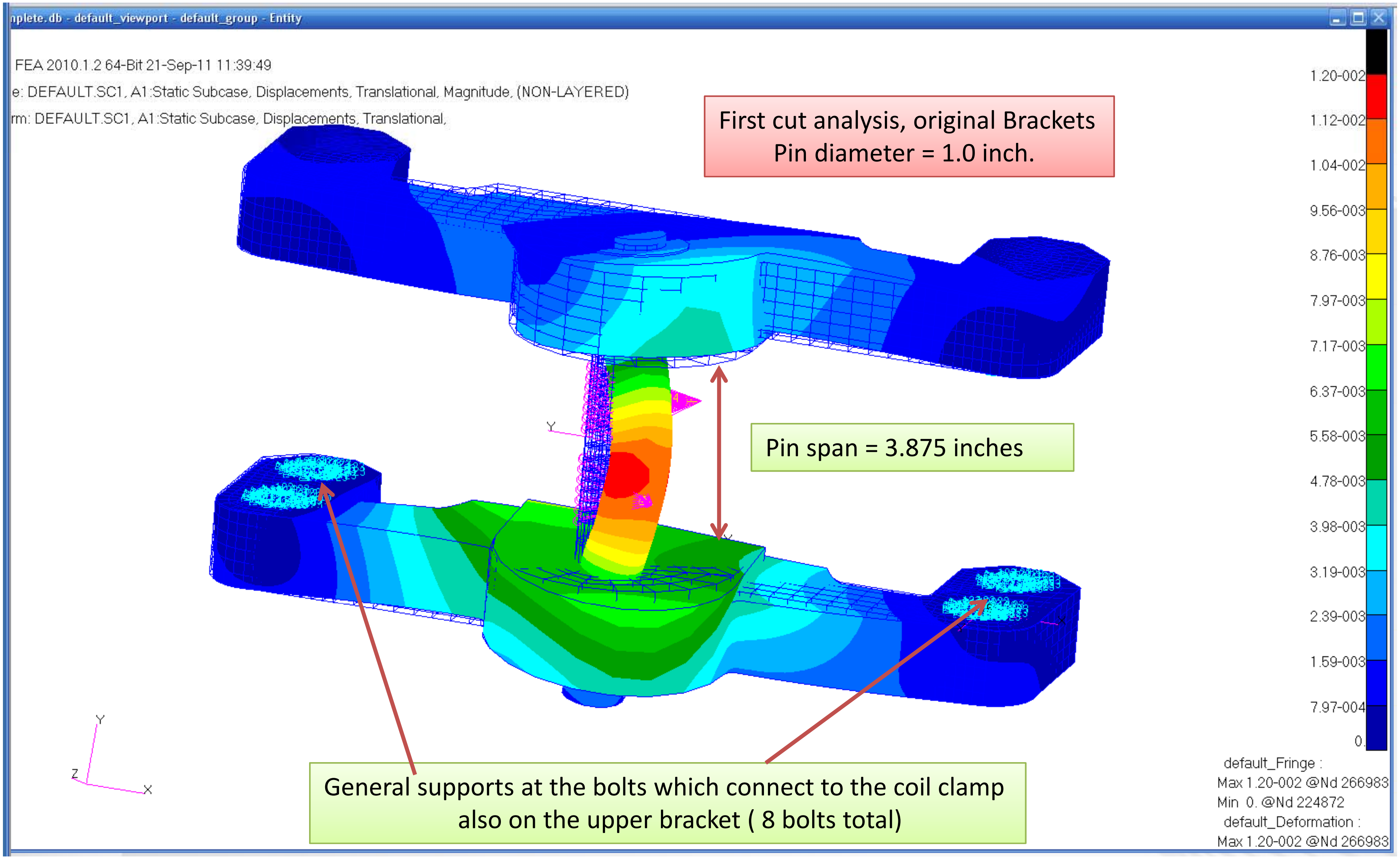


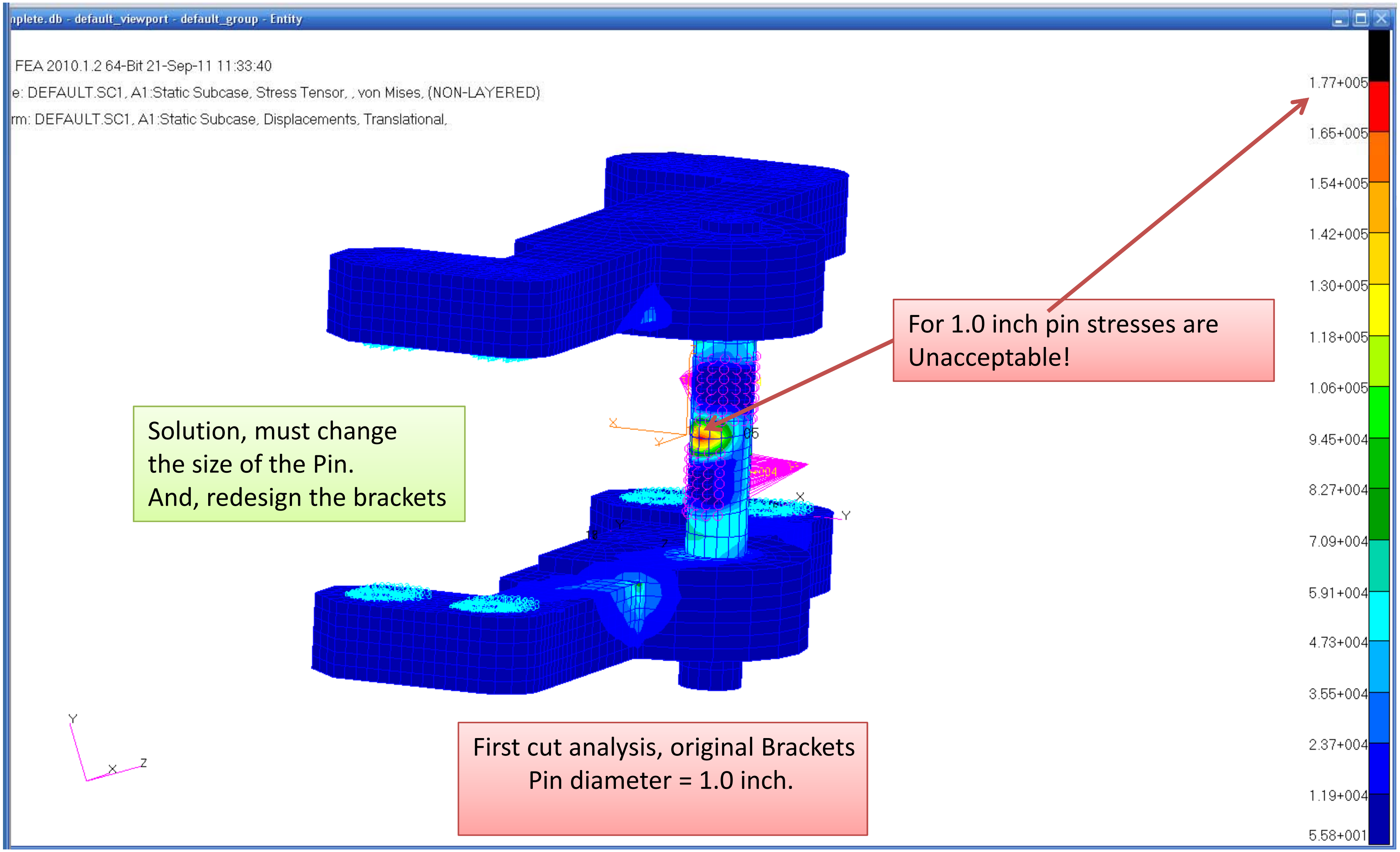

Pin ,Struts, inserts are made from Inconel-718

Brackets are stainless steel




Comments about the above specified forces:
Global ANSYS models simulate struts via Link elements which have the same coefficients of elasticity for tension as well as compression. In reality, the present strut design with connections to the Pins, as the coils system rotates, the compression coefficient of elasticity is about 45% greater. In turn, this should yield smaller displacements, and therefore, different force distribution (values). It is shown below that these forces are well within the values used in this analysis. Therefore, calculated stresses are most likely conservative.


nplete.db - default_viewport - default_group - Entity
FEA 2010.1.2 64-Bit 21-Sep-11 11:39:49
e: DEFAULT.SC1, A1:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
rm: DEFAULT.SC1, A1:Static Subcase, Displacements, Translational,
First cut analysis, original Brackets
Pin diameter = 1.0 inch.
Pin span = 3.875 inches
General supports at the bolts which connect to the coil clamp
also on the upper bracket ( 8 bolts total)
1.20-002
1.12-002
1.04-002
9.56-003
8.76-003
7.97-003
7.17-003
6.37-003
5.58-003
4.78-003
3.98-003
3.19-003
2.39-003
1.59-003
7.97-004
0.
default_Fringe :
Max 1.20-002 @Nd 266983
Min 0. @Nd 224872
default_Deformation :
Max 1.20-002 @Nd 266983




Comments on the fixity of the simulation:
The bolts which hold the brackets to the clamp-coil assembly will be installed with the allowable preload. This creates a constant pressure load on the surfaces in contact, based on the metal-to-metal friction rules. This would create additional support forces (especially in shear). These forces were not considered so that Results presented should be considered even more conservative .

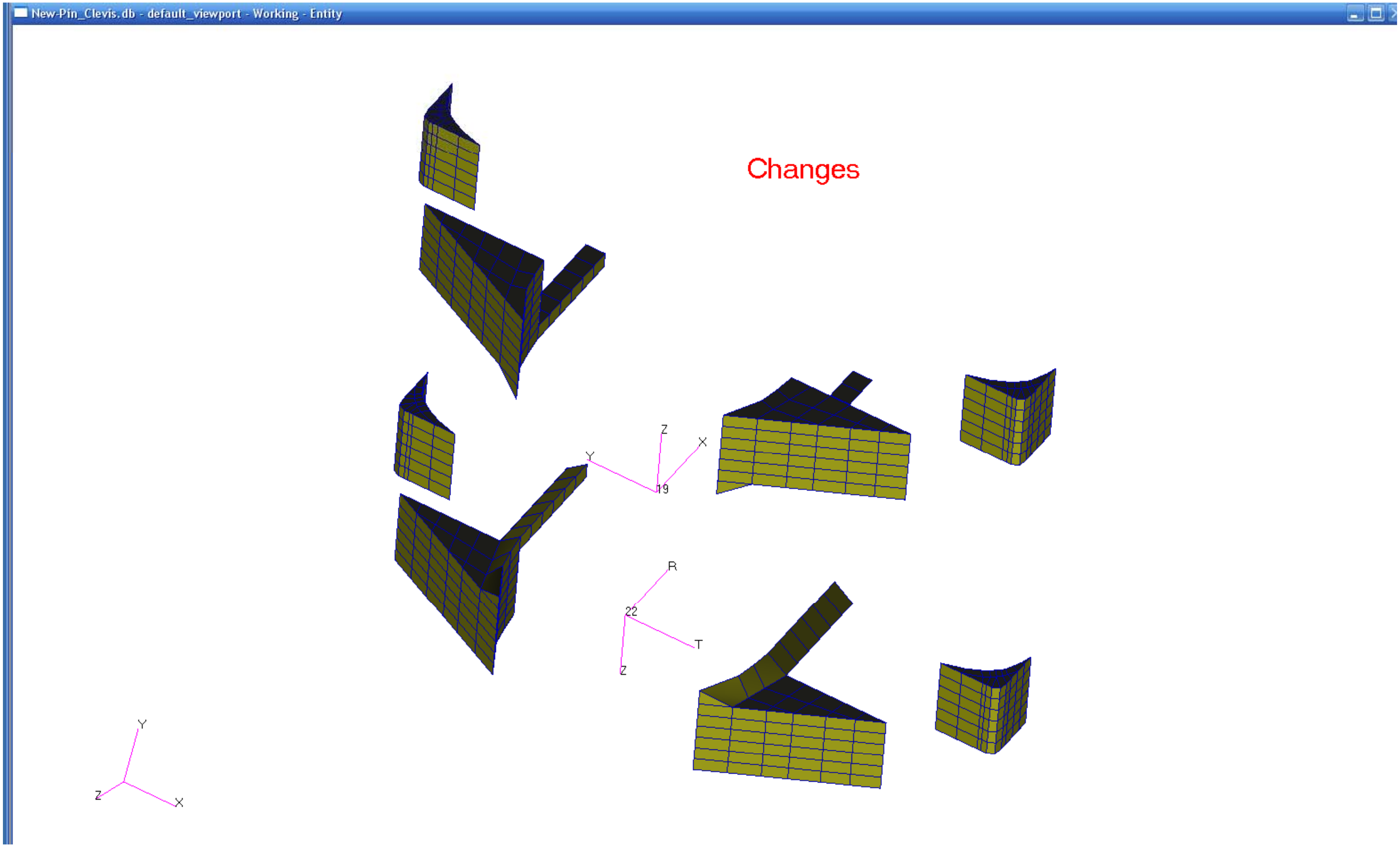
# Calculations for the Pin redesign

Since this Pin is principally stresses by the bending deflections due to the applied forces, let's use the standard formula "Sb=Mc/I" to approximate the necessary pin diameter which would show acceptable bending stresses. In this case, M (moment) remains the same, the value of "c" increases as well as "I=0.0491(D)^4" as diameter of the pin increases. Let's increase "D" from 1.0 to 1.25 inch and see how the bending stress can change.

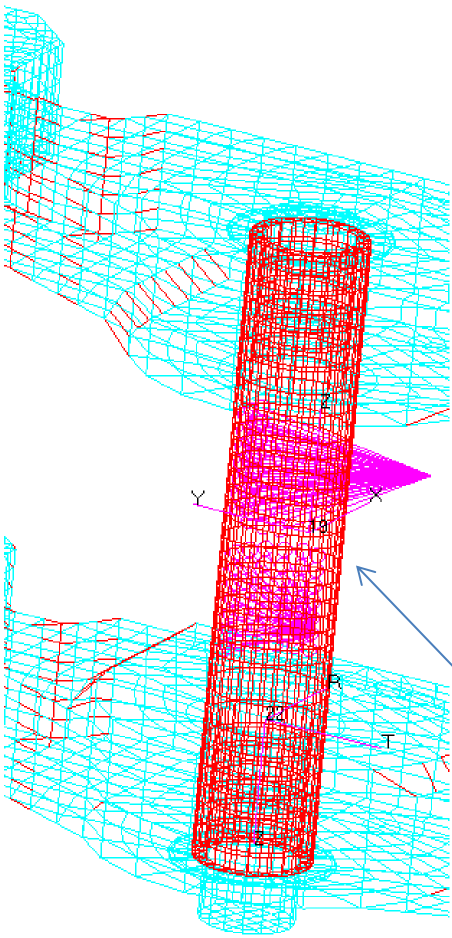
Therefore,
For Pin diameter = 1.0 inch: Sb = 177,000.psi (calculated above)
c = .5 inch , I =.0491( 1.0)^4 = .0491 in^4

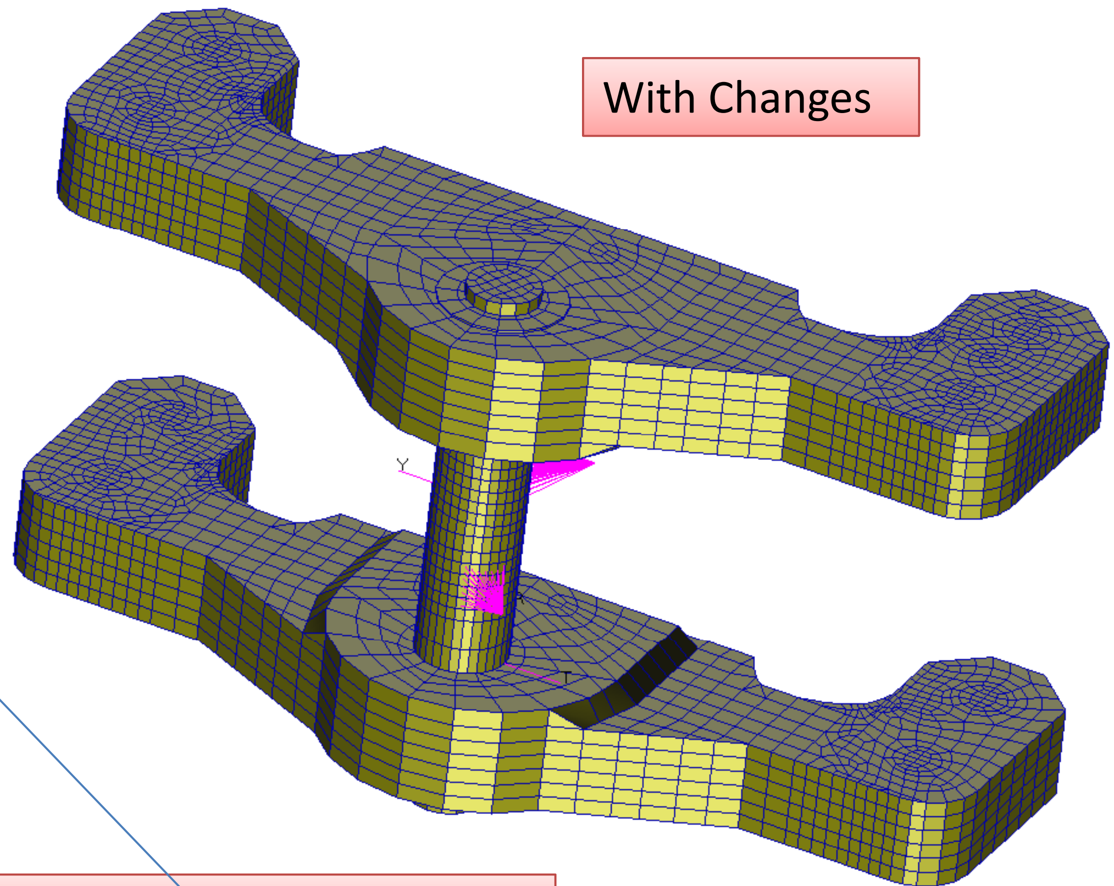
For Pin diameter = 1.25 inch : Sb = ? ( will be calculated)
c = .625 inch, I = .0491 (1.26)^4 = .11987 in^4
Since M is constant, [(Sb)(I)]/c (for D=1.0) = [(Sb?)(I)]/c (for D=1.25)
So that, Sb? = (177,000. x .0491 x .625) / (.5 x .11987)
= 90,625. psi.
Increasing the Pin diameter to 1.25 in decreases the bending stress to about half of the original FEA calculations.
This bring the allowable stress in the pin for Inconel-718 to acceptable levels.
Stiffening some of the critical areas of the supporting brackets should decrease Pin deflections, decreasing bending stresses further. See next slide!

The following slides show changes in the original NASTRAN simulation with positive results. Stresses and displacements are significantly decreased. See slides.




These additions were needed to eliminate stress concentrations in some critical locations

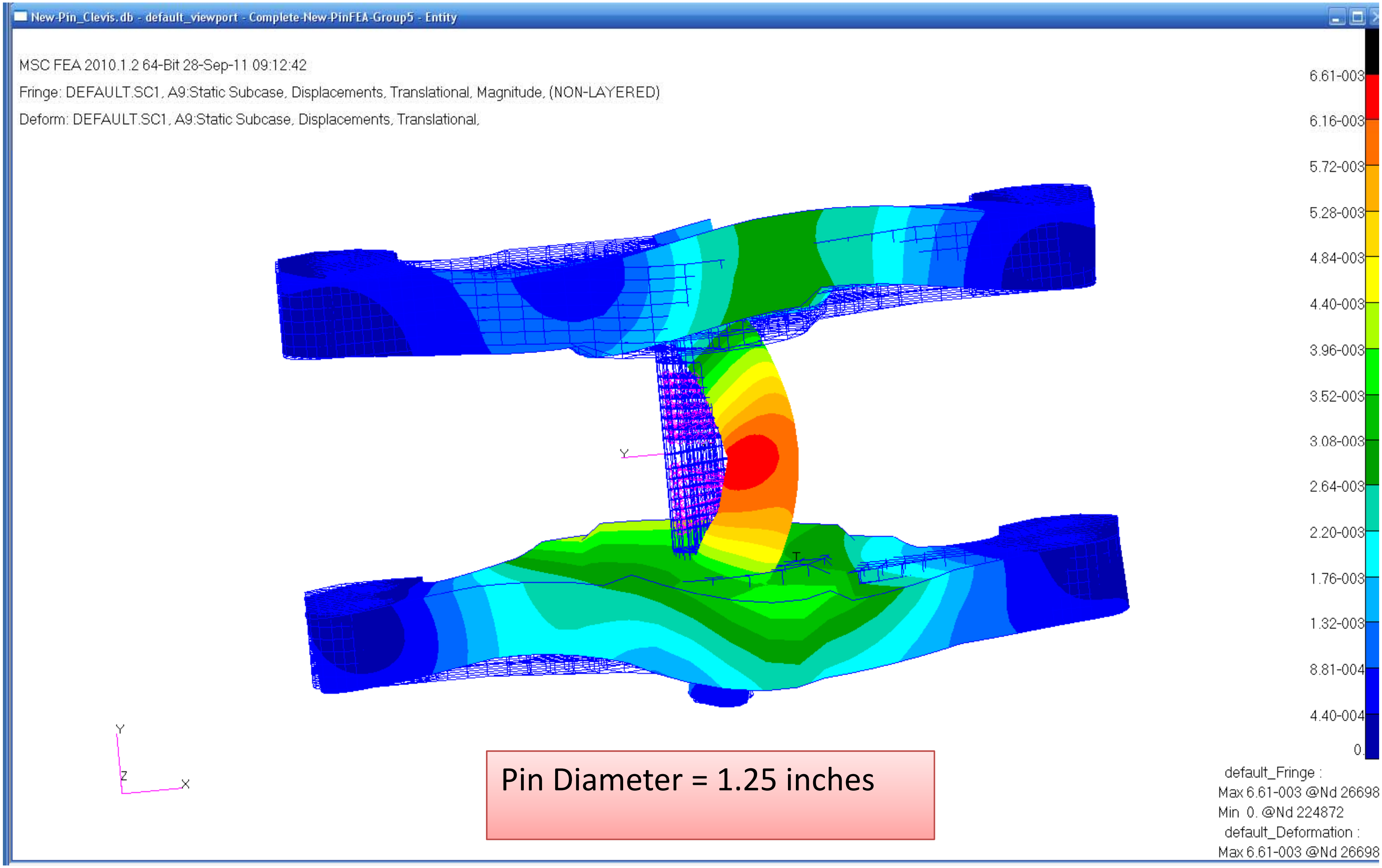




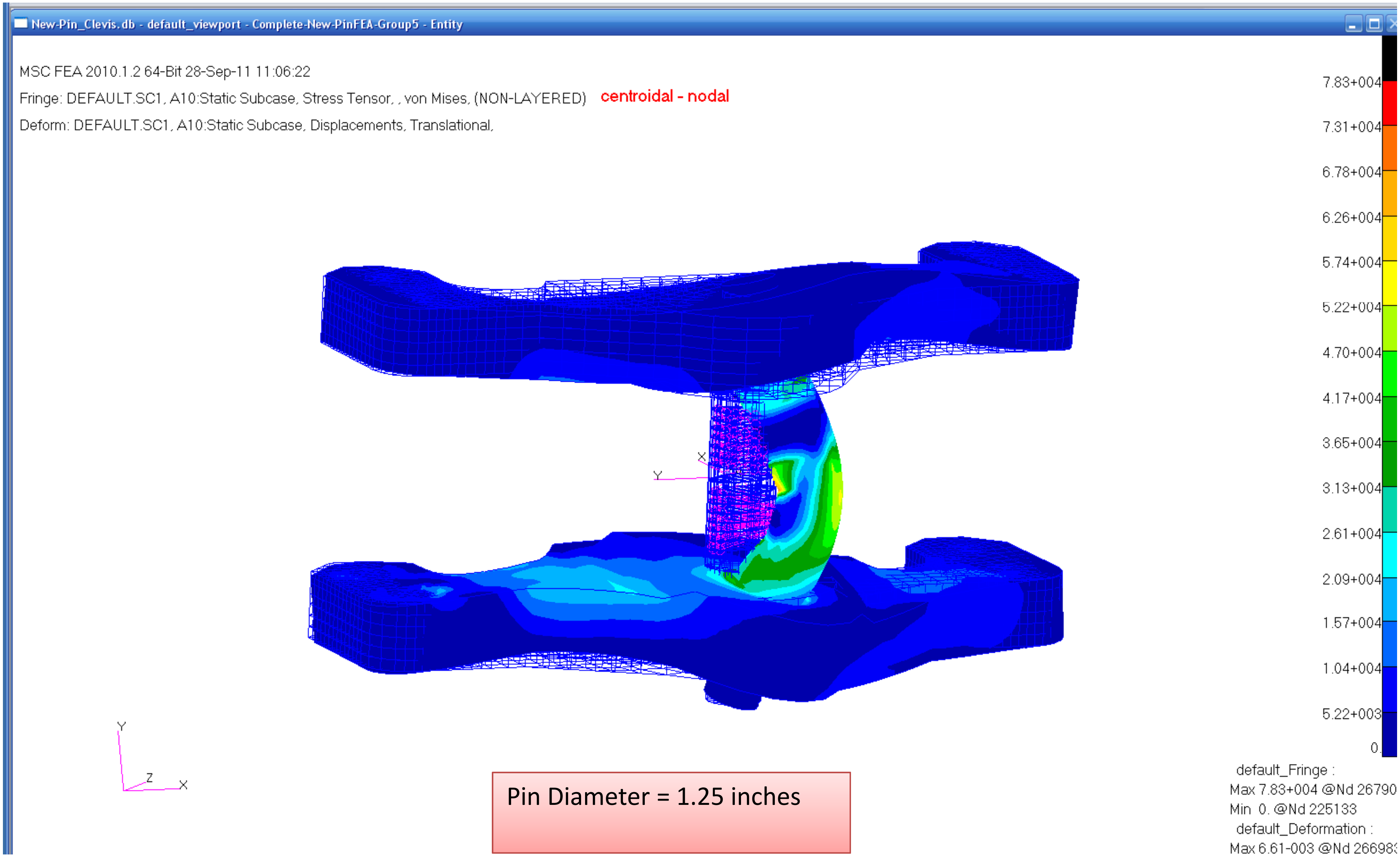
With Changes

Pin Diameter = 1.25 inches

Complete new configuration, pink arrows show the force application directions

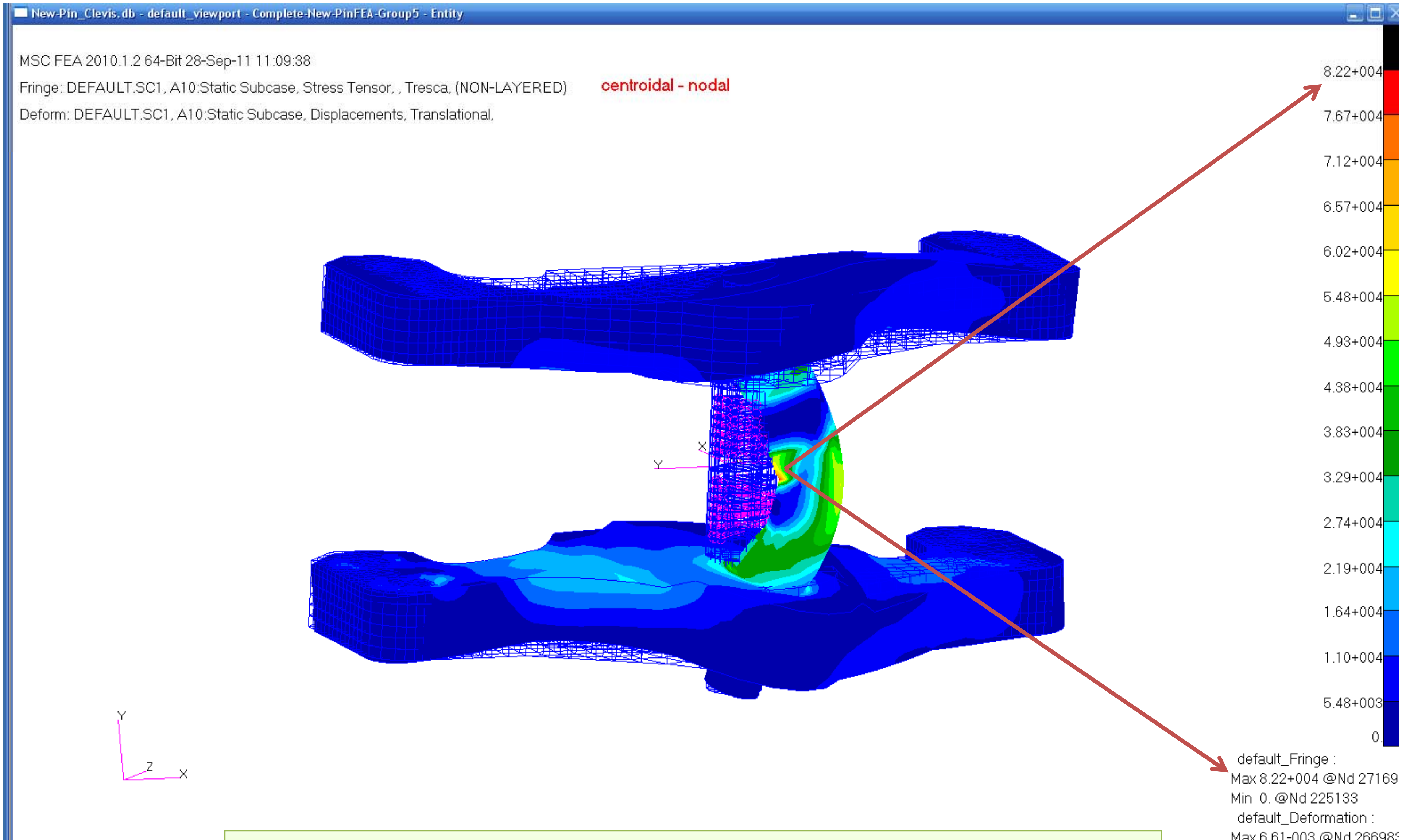



Maximum deformation changed from .012 inch to .0066 inch. Big improvement!

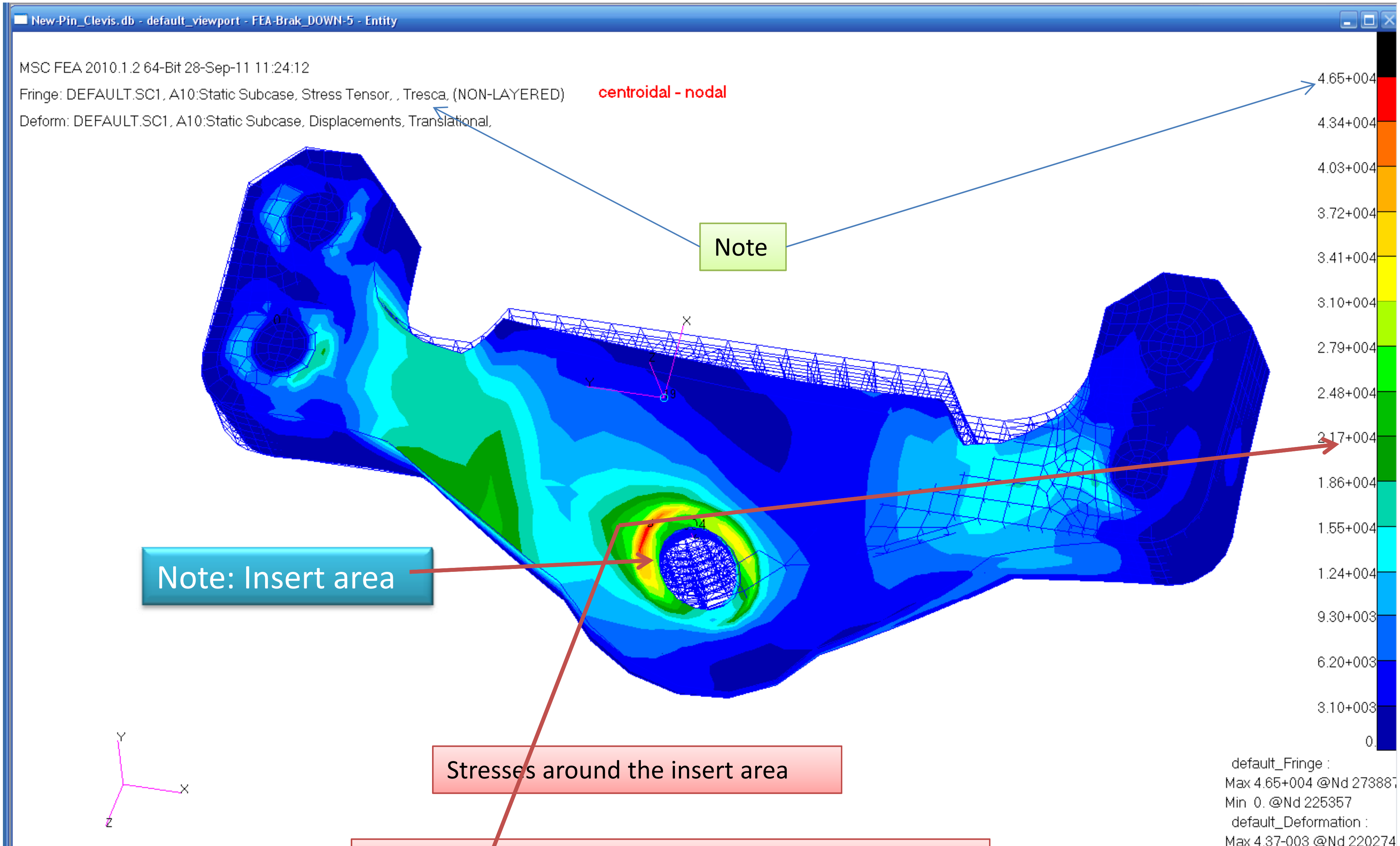



Pin Diameter = 1.25 inches

"von Mises" Stress decreases from 177,000. psi to 78,300.psi.
This is okay since Inconel 718 Yield = 150,000. psi.



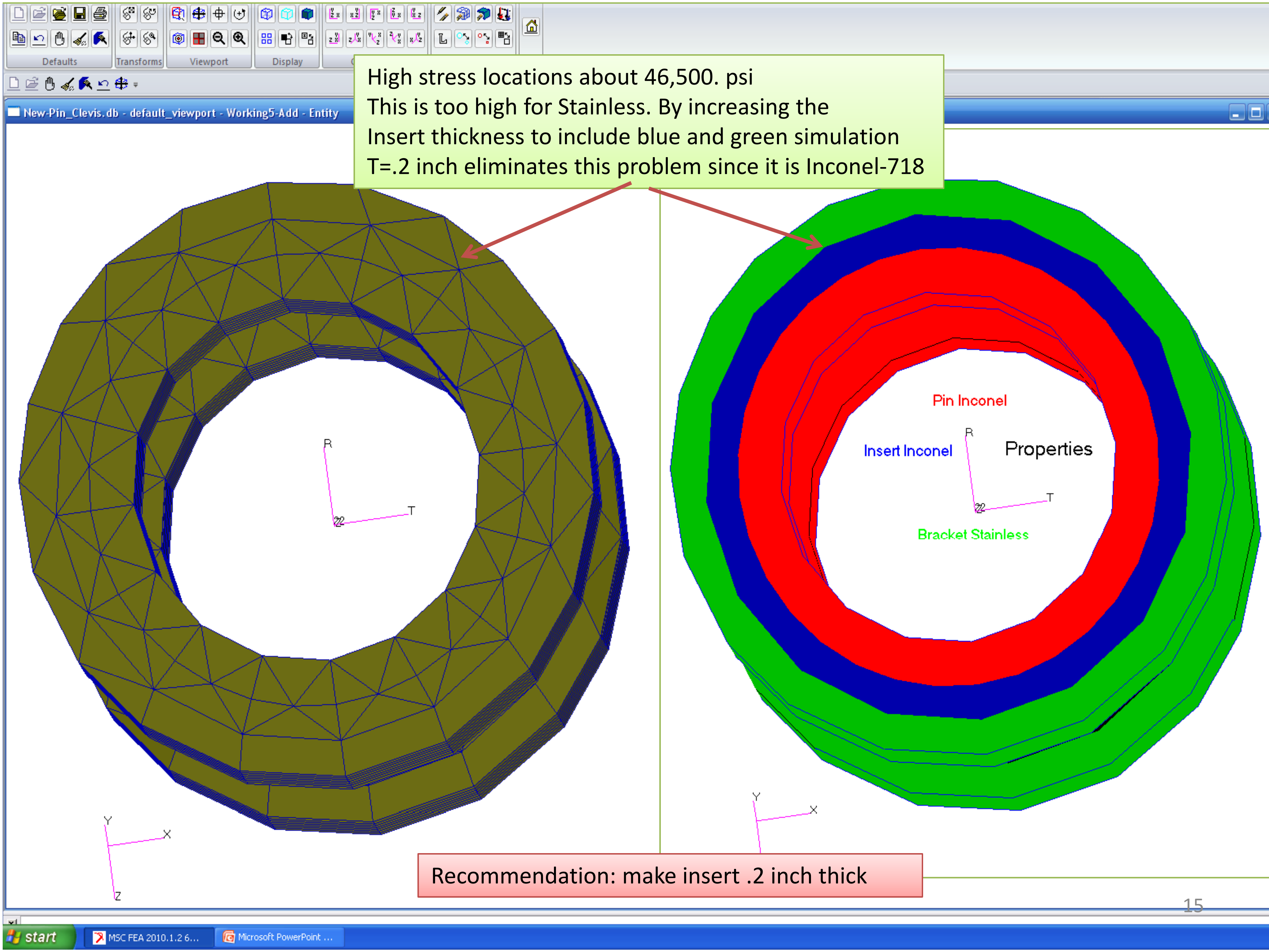

As a check, Tresca Stress (78,300.psi) is slightly higher than von Mises (82,200.psi ) but still okay.



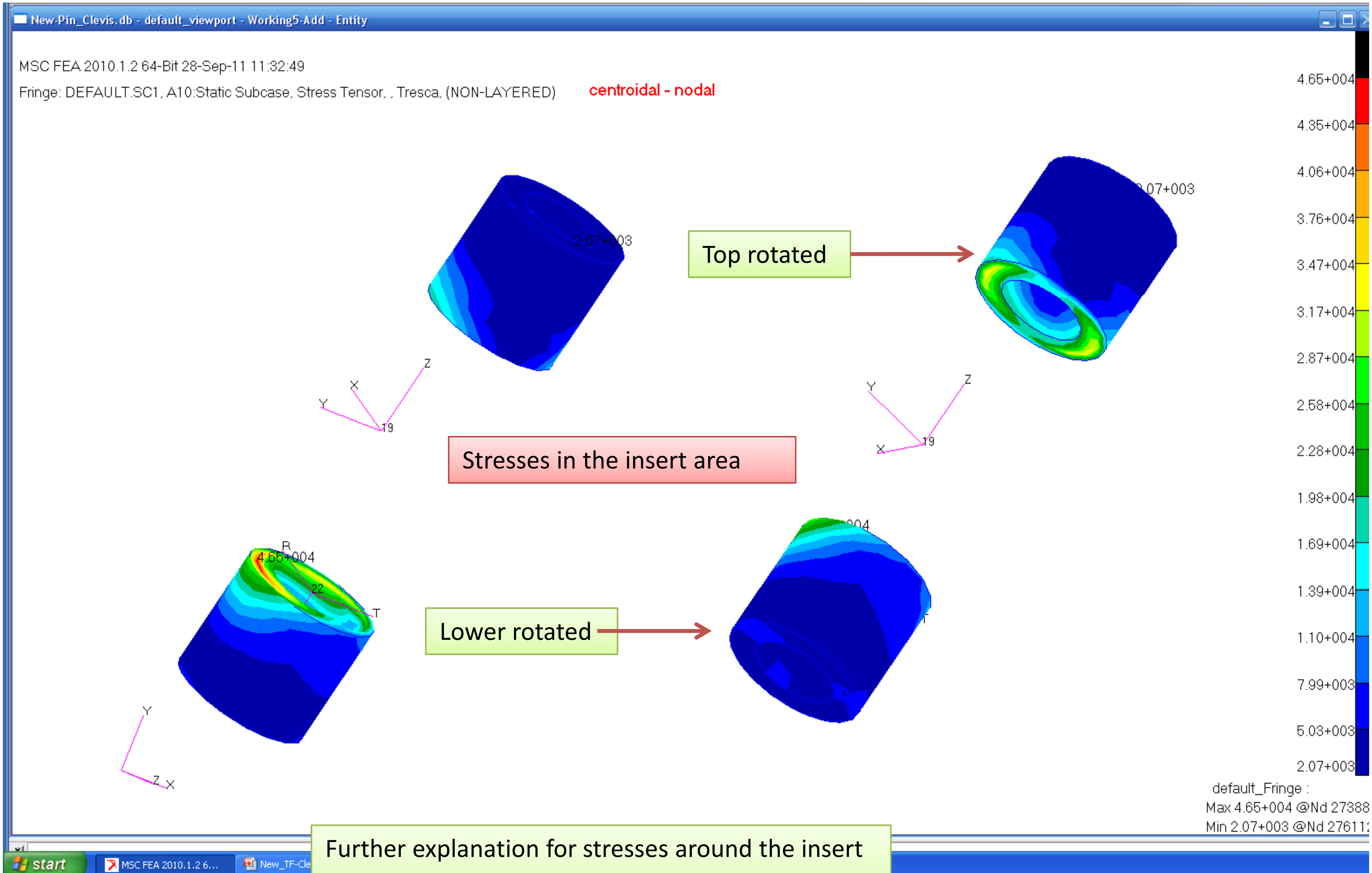

See the following slides for additional explanations

Max stress in the Bracket = 21,700. psi, Stainless 2/3Yield = 2/3(42,000,)= 28,000.psi, O.K.
Note: From case#2.

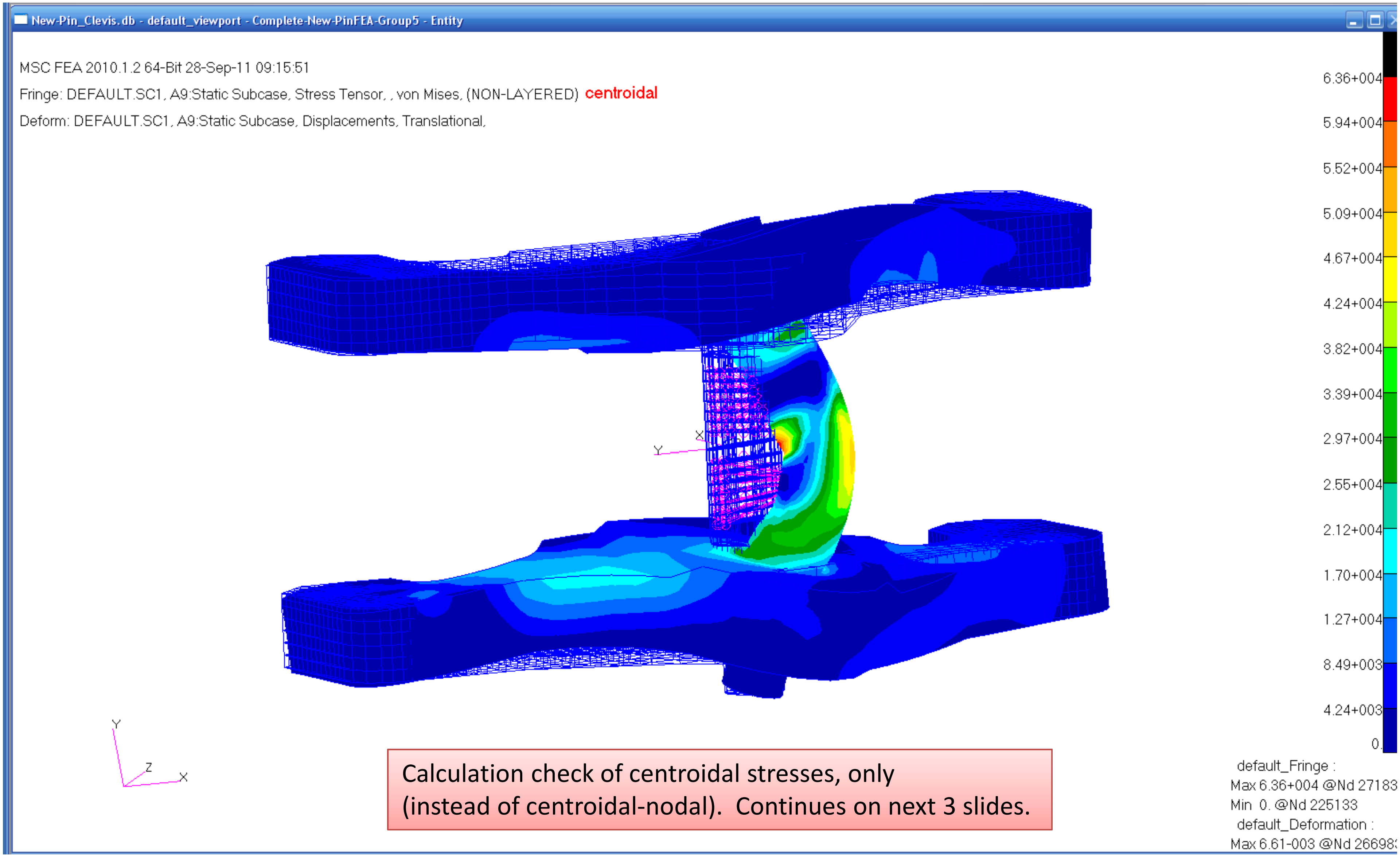
High stress locations about 46,500. psi
This is too high for Stainless. By increasing the
Insert thickness to include blue and green simulation
T=.2 inch eliminates this problem since it is Inconel-718

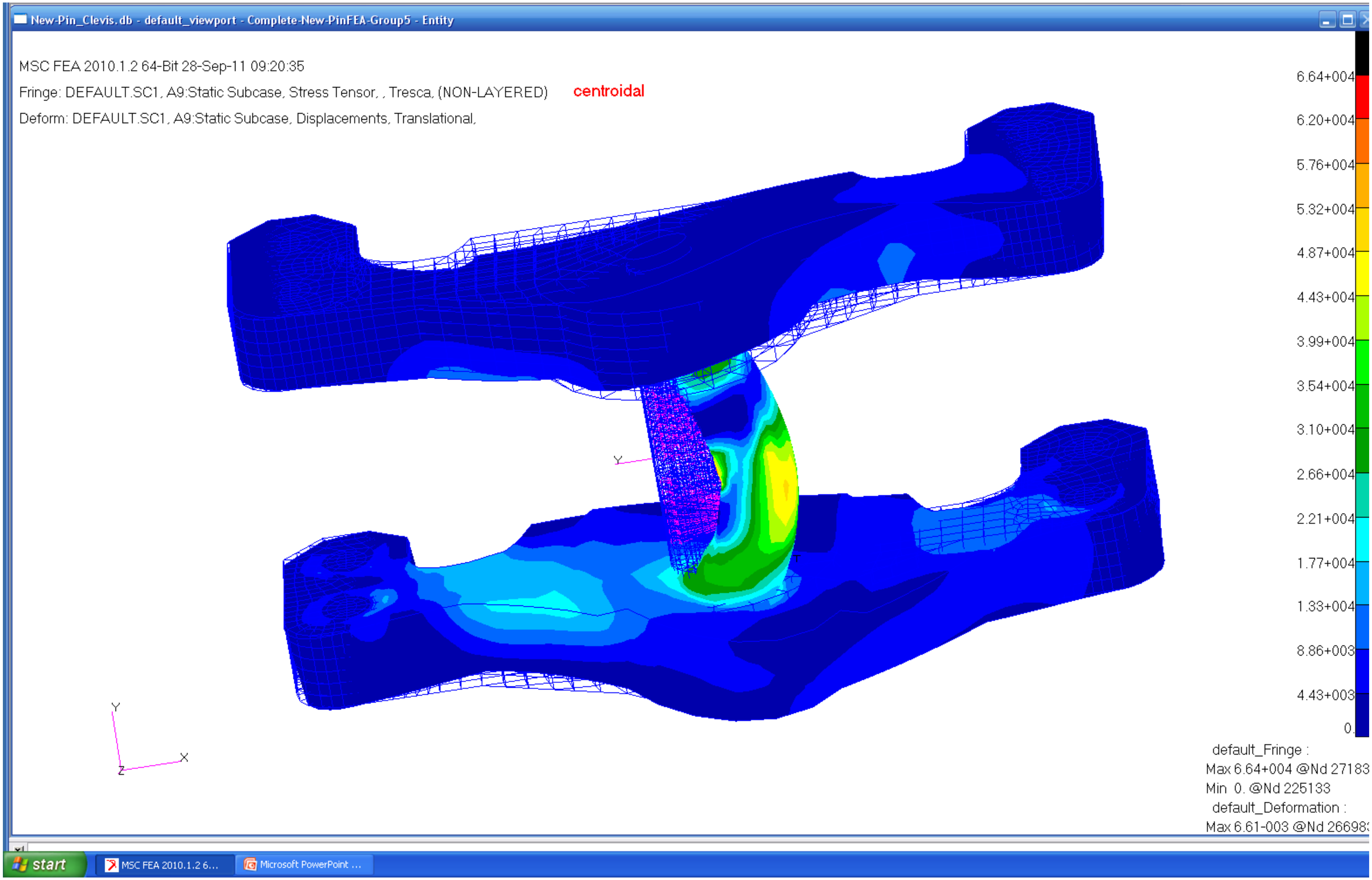


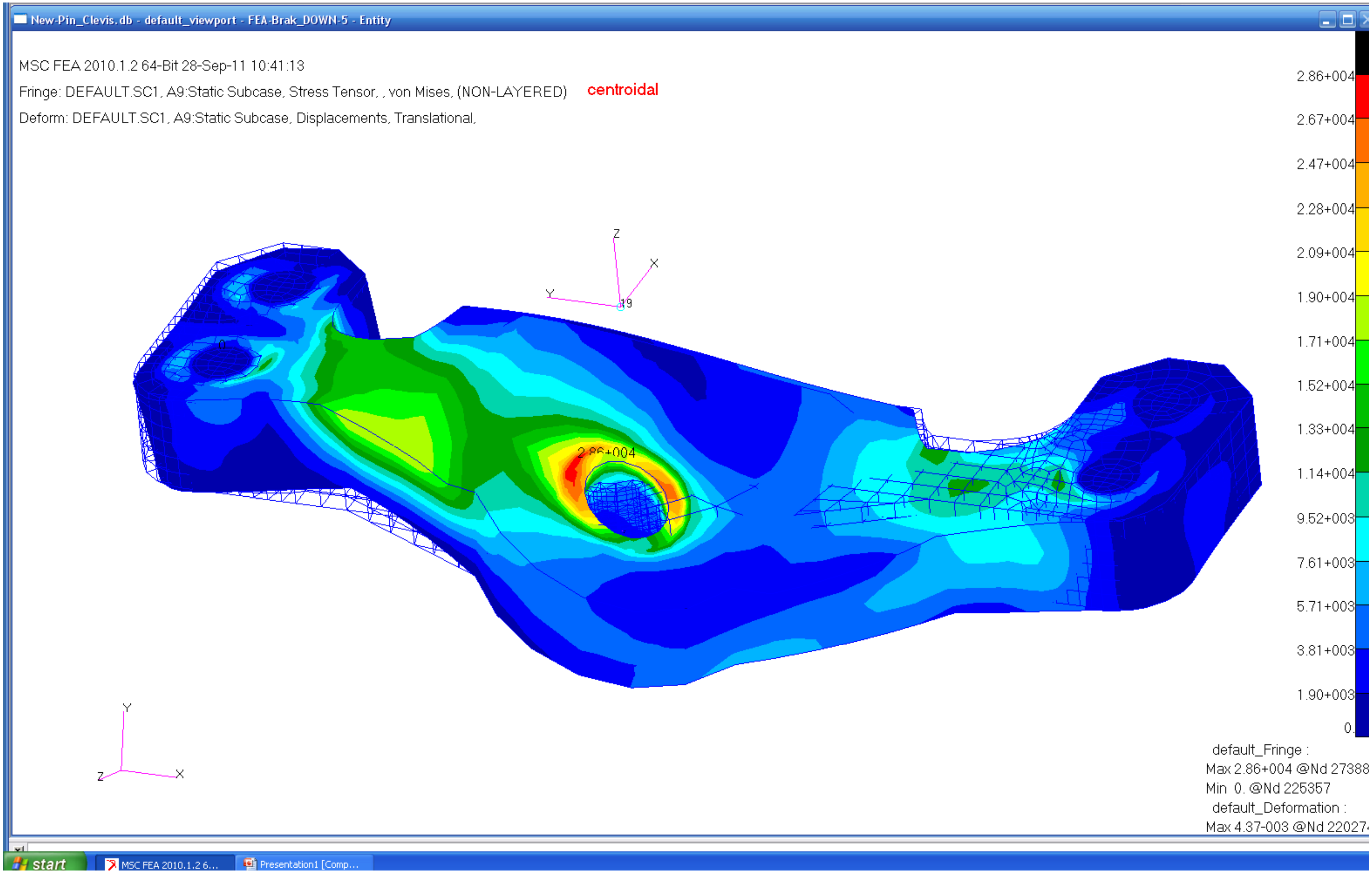

Recommendation: make insert .2 inch thick

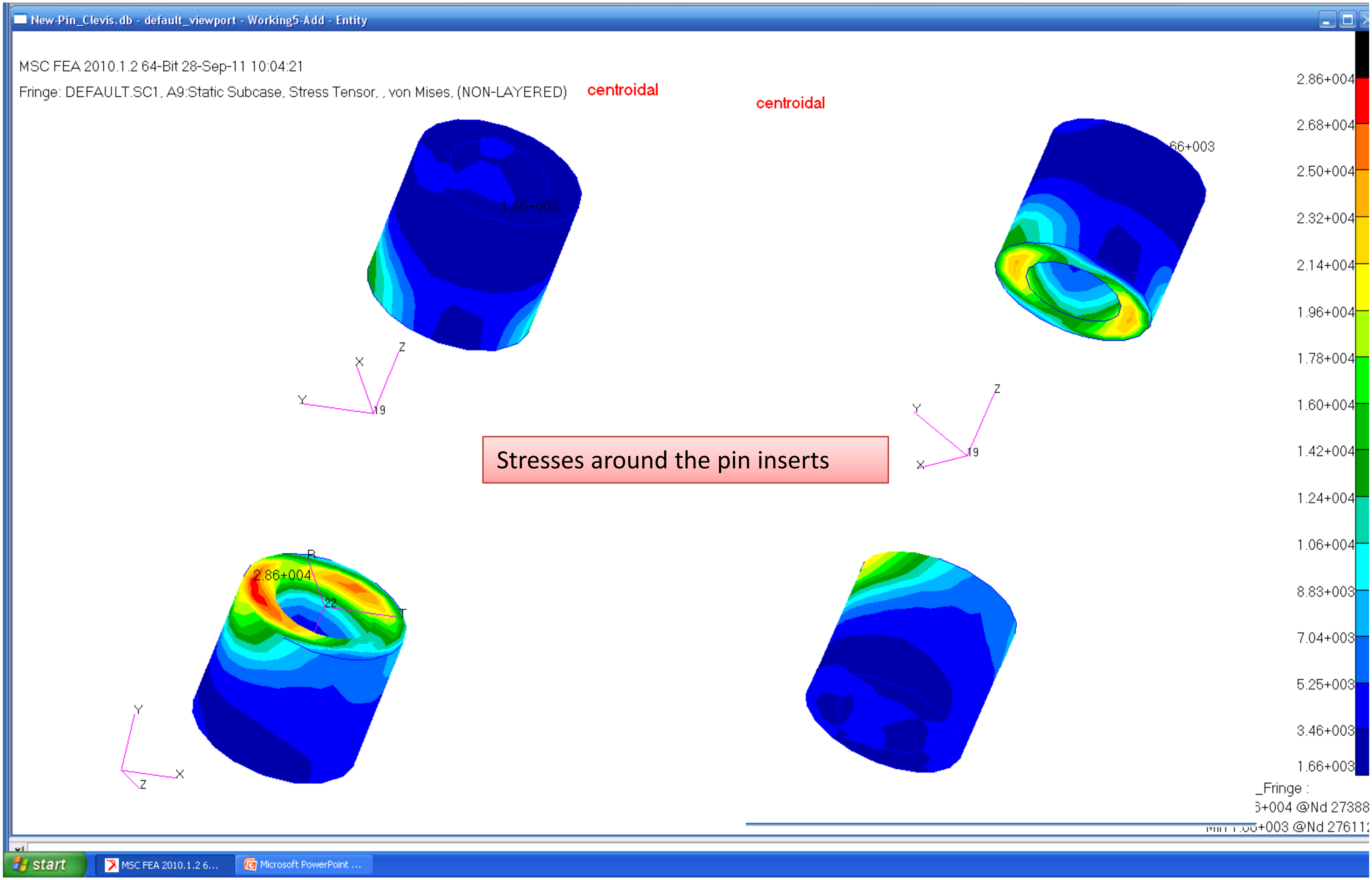



Further explanation for stresses around the insert
Insert is Inconel 718 with thickness = .2 inch, O.K.


New-Pin_Clevis.db - default_viewport - Complete-New-PinFEA-Group5 - Entity
MSC FEA 2010.1.2 64-Bit 28-Sep-11 09:15:51
Fringe: DEFAULT.SC1, A9:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED) centroidal
Deform: DEFAULT.SC1, A9:Static Subcase, Displacements, Translational,
6.36+004
5.94+004
5.52+004
5.09+004
4.67+004
4.24+004
3.82+004
3.39+004
2.97+004
2.55+004
2.12+004
1.70+004
1.27+004
8.49+003
4.24+003
0.
default_Fringe :
Max 6.36+004 @Nd 27183
Min 0. @Nd 225133
default_Deformation :
Max 6.61-003 @Nd 26698
Calculation check of centroidal stresses, only (instead of centroidal-nodal). Continues on next 3 slides.


New-Pin_Clevis.db - default_viewport - Complete-New-PinFEA-Group5 - Entity
MSC FEA 2010.1.2 64-Bit 28-Sep-11 09:20:35
Fringe: DEFAULT.SC1, A9:Static Subcase, Stress Tensor, , Tresca, (NON-LAYERED)
centroidal
Deform: DEFAULT.SC1, A9:Static Subcase, Displacements, Translational,
6.64+004
6.20+004
5.76+004
5.32+004
4.87+004
4.43+004
3.99+004
3.54+004
3.10+004
2.66+004
2.21+004
1.77+004
1.33+004
8.86+003
4.43+003
0.
default_Fringe :
Max 6.64+004 @Nd 27183
Min 0. @Nd 225133
default_Deformation :
Max 6.61-003 @Nd 26698
Y
X
Z
start
MSC FEA 2010.1.2 6...
Microsoft PowerPoint ...


New-Pin_Clevis.db - default_viewport - FEA-Brak_DOWN-5 - Entity
MSC FEA 2010.1.2 64-Bit 28-Sep-11 10:41:13
Fringe: DEFAULT.SC1, A9:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
centroidal
Deform: DEFAULT.SC1, A9:Static Subcase, Displacements, Translational,
2.86+004
2.67+004
2.47+004
2.28+004
2.09+004
1.90+004
1.71+004
1.52+004
1.33+004
1.14+004
9.52+003
7.61+003
5.71+003
3.81+003
1.90+003
0.
default_Fringe :
Max 2.86+004 @Nd 27388
Min 0. @Nd 225357
default_Deformation :
Max 4.37-003 @Nd 22027
start
MSC FEA 2010.1.2 6...
Presentation1 [Comp...


New-Pin_Clevis.db - default_viewport - Working5-Add - Entity
MSC FEA 2010.1.2 64-Bit 28-Sep-11 10:04:21
Fringe: DEFAULT.SC1, A9:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
centroidal
centroidal
Stresses around the pin inserts
2.86+004
2.68+004
2.50+004
2.32+004
2.14+004
1.96+004
1.78+004
1.60+004
1.42+004
1.24+004
1.06+004
8.83+003
7.04+003
5.25+003
3.46+003
1.66+003
_Fringe :
start
MSC FEA 2010.1.2 6...
Microsoft PowerPoint ...

# Conclusion

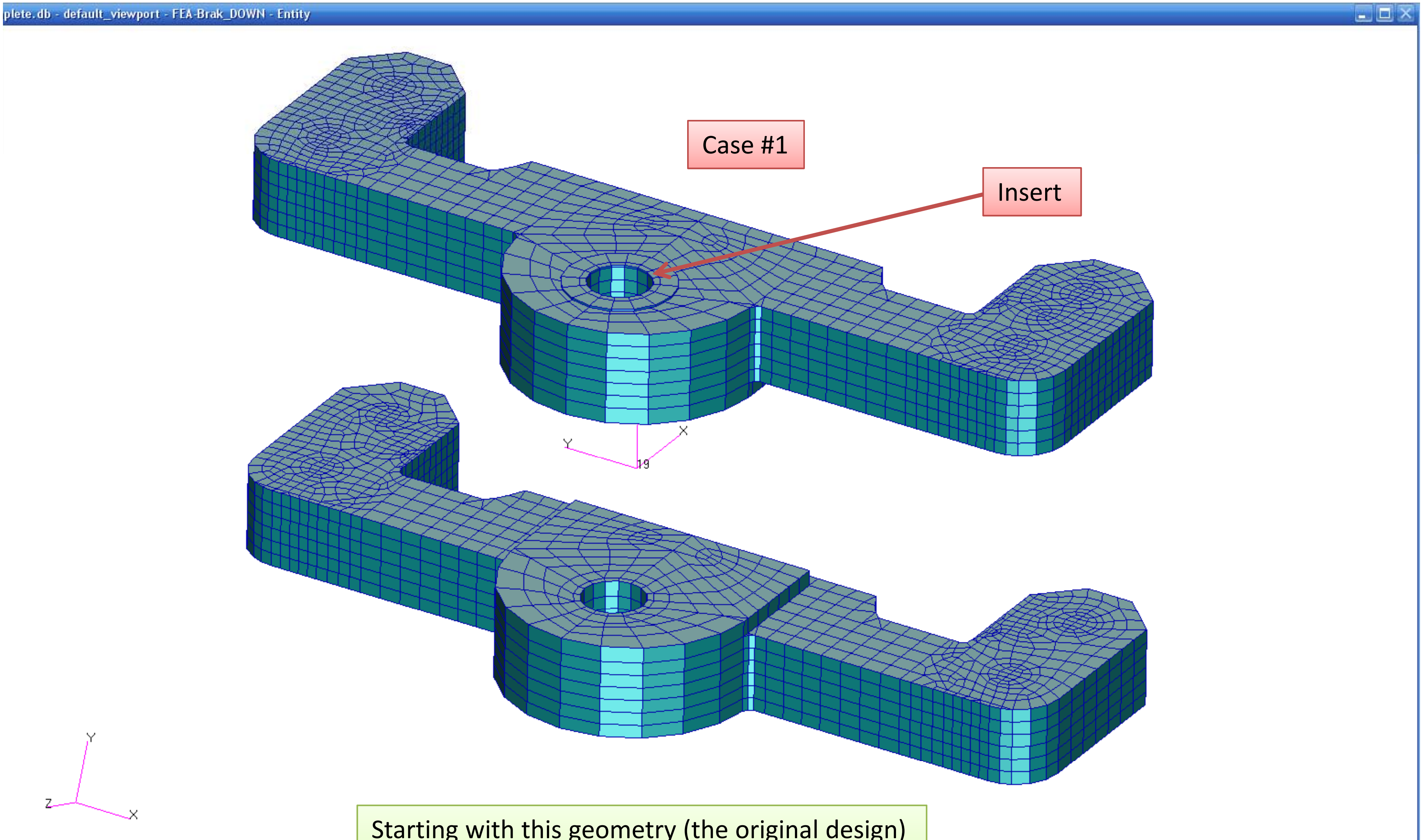
- Max Pin stress from 78,300 psi (von Mises) to 82,200 psi (Tresca)
- Max Insert stress = 46,500  psi (Tresca)
- Max Brackets stress = 21,700 to 24,800 psi

- Update the geometry of the Brackets (Case#3)
- Increase Pin diameter to 1.25 inches, Inconel 718, Yield = 150,000. psi
- Increase the Insert thickness to .2 inches, Inconel 718, Yield = 105,000. psi
- Keep the Bracket as Stainless Steel, Yield = 42,000.psi

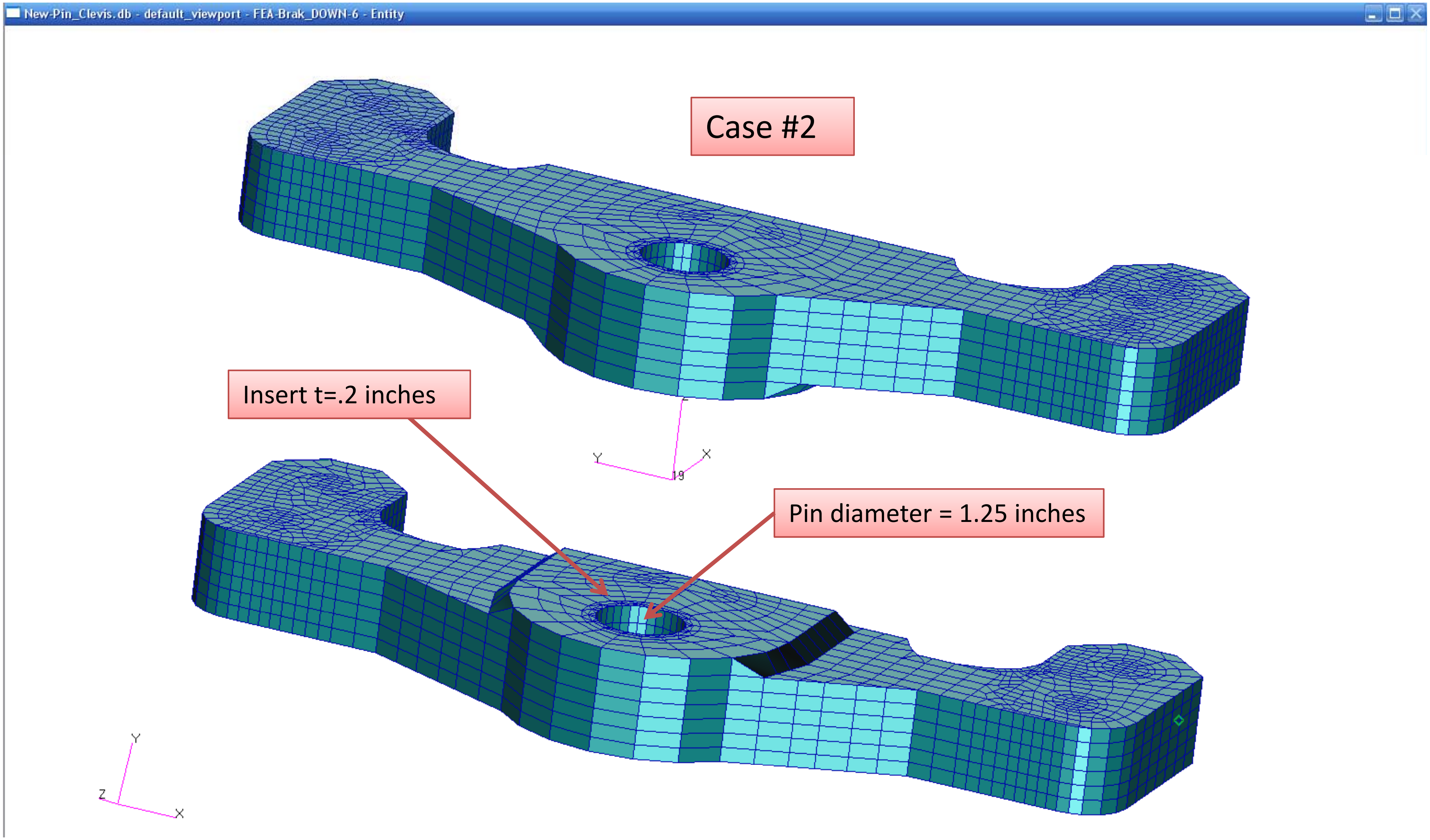
Based on the Case #2 NASTRAN simulation above numbers are acceptable

- Next: Case #3 can be executed if it becomes necessary? However, obtained data for Case #2 is already conservative for used geometry and applied loads. Case #3 bracket configuration increases the bracket stiffness, which in turn, will decrease the pin displacement, and therefore, decrease the pin bending stresses. This would make the assembly behavior even more conservative?

See the next three slides for the description of the analyses progression based on shown cases.

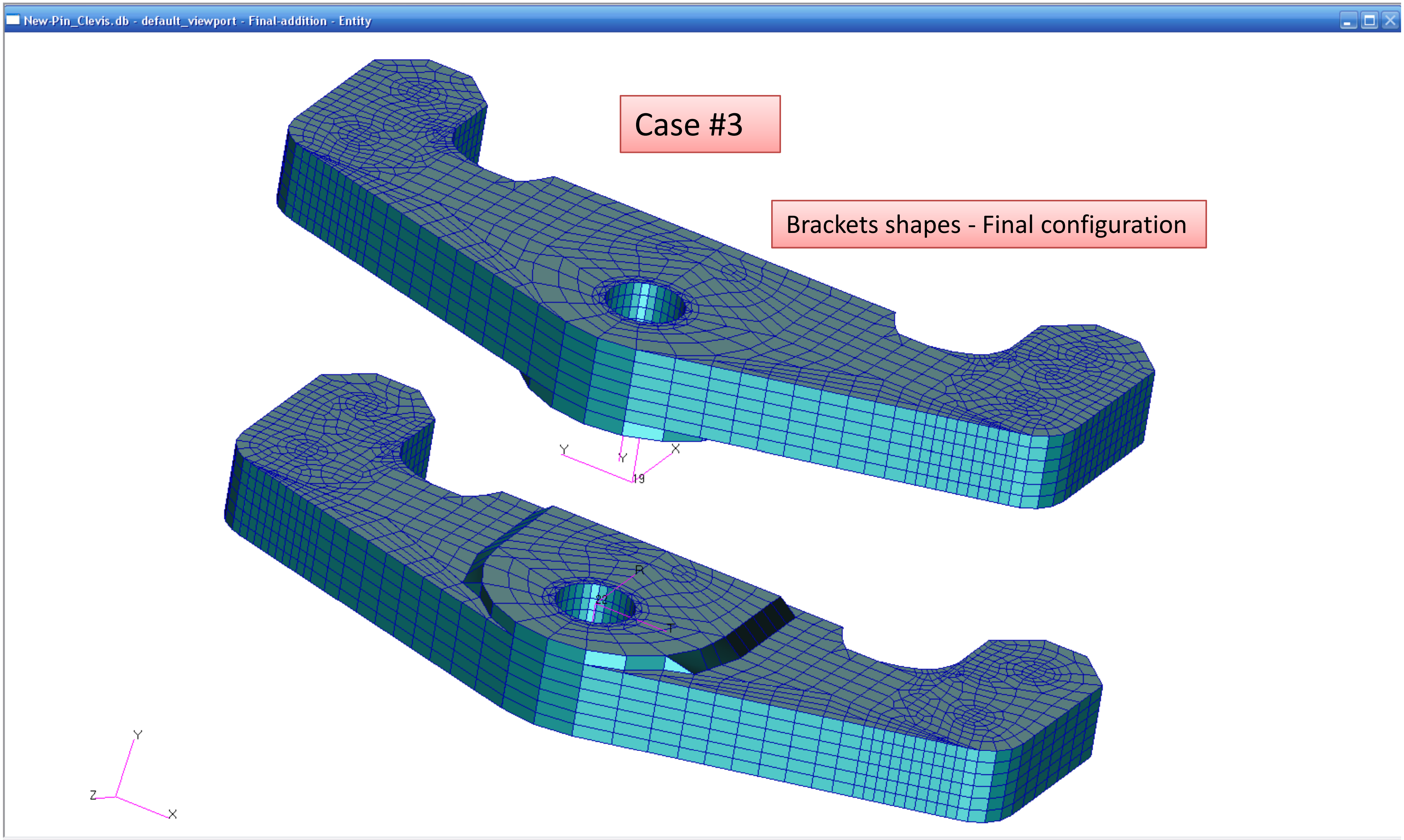



Starting with this geometry (the original design)
Where the pin diameter is 1.0 inches
Pin insert thickness is = .1 inches




First improvement in the bracket geometry:

- Elimination of the stress concentration area which were identified in Case #1
- Pin diameter increased to 1.25 inches decreases the pin bending characteristics.
- Explain the necessity of increasing the insert thickness to .2 inches.




Note: This configuration was not executed through NASTRAN yet because of time constraint. It could be examined at a later date if it becomes necessary? However, results from this simulation should be considered even more conservative than the already acceptable Case #2.
In Case #3, the structure is stiffer and displacements become smaller!

## Appendix
Locations of the important NASTRAN data bases

On "ASALEHZ-64PC"
G:\Nastran P.R.\TF Coil_Clamps

OR:

P:\public\Snap-srv\progoff\Nastran P.R\TF Coil_Clamp