NSTX-U

Redesigned Vacuum Vessel Support Bracket

NSTXU-CALC-12-10-01

Rev 1

February 2013

Prepared By:

Peter Rogoff Digitally signed by Peter Rogoff
DN: cn=Peter Rogoff, o=MED, ou=PPPL, email=progoff@pppl.gov, c=US
Date: 2013.02.15 09:59:13 -05'00'

Peter Rogoff, PPPL, MED Engineering

Reviewed By:

Irving Zatz Digitally signed by Irving Zatz
DN: cn=Irving Zatz, o=PPPL, ou, email=zatz@pppl.gov, c=US
Date: 2013.02.15 10:42:39 -05'00'

Irving Zatz, PPPL, MED Engineering

Reviewed By:

Peter Titus Digitally signed by Peter Titus
DN: cn=Peter Titus, o, ou, email=ptitus@pppl.gov, c=US
Date: 2013.02.15 11:07:52 -05'00'

Peter H. Titus, PPPL, MED Branch Head

PPPL Calculation Form

Calculation # **NSTXU-CALC-12-10-01** Revision #**01** WP # **1677**
(ENG-032)

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

The NSTX Vacuum Vessel (VV) main support structure consists of a specially designed "Chair" which is welded at four locations ninety degrees apart along the surface of the Vessel. However, the clevis assemblies are required at thirty degree intervals to provide connectivity between the lower TF coil ring/clamp assembly and the VV. This mechanism provides better structural support for total NSTX-U device. The original device (the NSTX) had a completely different support structure which permitted a symmetric "Chair" design. Because of the introduction of the heavy link member (see, NSTX-U-CALC-132-12-00), the existing structure had to be redesigned eliminating part of the assembly. This new geometry and the FEA model simulation are provided in the enclosed power point presentation. Therefore, a complete set of analyses to check the integrity of the redesign were required for various new loads and VV temperature conditions.

The original calculations (March 2012) , were done using the most ideal VV – Support Chair connectivity assumptions. The weld size of 3/8 inch was assumed and simulated only in most accessible locations. However, upon the NSTX further disassembly, it become clear and necessary to implement a more representative welding scenario and replace some chair members to assure a better total device assembly effort. To check and justify new changed parts and increase of the welds to 5/8 inch, the As-Built justification became necessary. This initiated Revision 1.

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

A complete set of FEA simulation runs were performed utilizing the "FEA 2010.1r.2, NASTRAN code". Applied forces and boundary conditions were extracted from various ANSYS code Global Models simulations. Results are presented in the best possible graphics format clearly marking maximum stress and/or strain values and locations.

For Revision 1, an additional FEA calculation was performed (Run#5-MOD1-106NL-fea) using the same code and modified weld geometry (see below, the new power point file 'VV-Support-Chair-as-Built.ppt)

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

For this analysis the following assumptions are important: The ninety degree VV sector is simulated to provide the complete symmetry representation in the vicinity of the chair/support structure. This provides sufficient simulation for the boundary conditions to permit the critical location stress/strain values predictions. NASTRAN MPC conditions are utilized to simulate the VV symmetry condition. Loads are applied as pressure on the vertical open VV surface as demonstrated in the power point slides. The vertical support beam is fully fixed at the location where angled brackets connect it to the test cell floor. This is expressed with the NASTRAN SPCs application set to zero for three principal coordinates. (Note: since the used solid elements require only translational (X ,Y, Z) degrees of freedom). The assumption is that at this location, the complete support structure remains in a steady state condition because of the angled floor brackets.

Calculation (Calculation is either documented here or attached

All the existing calculations are provided and explained by the following Power Point file:

Vessel-Suppot-Chair.ppt (original calculations – March 2012)

This presentation describes the complete set of the NASTRAN simulation of the VV Chair- Support structure. This file presents simulations inputs as well as the outputs for the six individual runs as follows below:

RUN #1 – Static (Solution 101) with extreme boundary conditions that react the entire load through the four bolts in the vacuum vessel support - total vessel load of 160,000 lbs, but, scaled to 40,000.lbs for a ninety degree sector. This simulation was used to investigate the maximum possible stresses on the VV/Chair welded locations. This case is considered overly conservative.

RUN #2 – Static simulation with relaxed boundary condition, spreading the support loads through the pad and not just the bolts, using the same SPCs, MPCs, and loads as in the RUN #1. With this simulation additional information was demonstrated for the maximum expected calculations.

RUN #3 – Static simulation with final and complete model with RBE2(rigid link) elements instead of GAPs. SPCs are changed to the vertical beam location where angled floor brackets connect to it. MPCs and applied load pressures are as in the prior runs. This run correctly simulates the overall intended analyses approach.

RUN #4 – Non linear run (Solution 106) where RBE2s were replaced with the proper GAP elements which are aligned to Coordinate #2 with element X-direction in the negative vertical direction. Elements slide (with a coefficient of friction assumed to be 0.3) in the element Y and Z direction. The obtained results are very similar to the data from RUN #3. The reason for this is that there is not much global radial motion without the thermal effects on the Vacuum Vessel.

RUN #5 – Non linear run with the addition of the 37,000. Lbs. link/clevis load as shear parallel to the vacuum vessel surface. All other loads and boundary conditions are identical to RUN #4. Obtained results values for this case increase since an addition load was added. Locations for the maximum values are relocated but remain reasonable for the material allowable. This simulation is most representative of the standard device operating environment since it includes the ring/clamp vacuum vessel clevis forces.

RUN #6 – Non linear “BAKEOUT” simulation. This is a special run which includes the combination of the mechanical loads due to the vacuum vessel weight and the possible worst case thermal differences during the “BAKEOUT” event as the structure reaches a steady state condition after 1,200 minutes of temperature application. Same scenario was used to predict this condition for the PF 4 and 5 support brackets where temperature profile was measured and recorded during the actual “BAKEOUT” operation. For RUN #6, previous data was converted from degrees C to F and distributed accordingly on the complete simulation using element groups as described in the appropriate powerpoint slides. It is assumed that this is the best possible representation for this RUN #6 case. (please see the complete set of these attached powerpoint slides).

VV-Support-Chair-as-Built.ppt (update February 2013)

This presentation describes the As-Built VV Support Chair configuration. After inspecting the photos provided by the NSTXU assembly team, the actual chair assembly appeared well connected to the vacuum vessel. These photos show that, the every contact edge between the chair and the vessel is welded using 5/8 inch welds. The welds in the original simulations were modeled on the outside connecting edges only and are sized at 3/8 inch. However, the team reported, that the chair support plate may not be completely welded to the vessel in the upper location. It is difficult to visually inspect because certain coil assemblies are obstructing the view. This initiated a modification of the Run #5 simulation by removing the upper weld elements (see the reported slides for the exact location). The results from this modified simulation show the absolute maximum stress concentration for 3/8 inch welds (appropriate slides).

Therefore, the obtained maximum Von Misses stress = 22,100 psi, (upper right corner). By converting 3/8 inch to 5/8 inch weld areas, the maximum stress for a 5/8 inch weld decreases to 7,840 psi. This clearly demonstrates the superiority of the As-Built configuration and no further FEA analyses are necessary.

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

Based on the above described simulation the following final list shows important findings for the Redesigned Vessel Support Bracket assembly and the connecting structure. Also, please see the summary slide in the power point file. Conclusions are as follows:

- Calculated stresses and strains are well within the material allowable,
- Maximum stresses for the welded locations are equal to 16,200 psi
- Maximum strains for the welded locations are equal to .00047
- Maximum stresses for the VV-chair location are equal to 21,500 psi
- Maximum strains for the VV-chair location are equal .00062

- Based on the total set of simulations done earlier, the As-Built visual examination and weld size comparison, it is concluded that the present connection between the Vessel and existing supports are acceptable during maximum load application. Stresses in the chair structure, its sliding cradle surfaces and newly implemented 5/8 inch welds are well within the material allowable.

  Previous calculations demonstrated the historical simulations to understand the general behavior of the entire system. The As-Built results describe the justification for the present weld size and configuration.

Note: All the maximum weld stress values are calculated by RUN #6 which includes the “BAKEOUT” temperature scenario. The results obtained in RUN #5 are slightly smaller but important peak stress locations are different.

Strong Recommendation: to carefully weld the assembly connecting corners since all maximum strains and stresses are concentrated in these locations. These additional welds should decrease the peak values making the design stronger! Please see slide #30.

.

Preparing Engineer’s printed name, signature, and date

Peter Rogoff 2/14/2013 Peter Rogoff

Digitally signed by Peter Rogoff
DN: cn=Peter Rogoff, o=MED, ou=PPPL, email=progoff@pppl.gov, c=US
Date: 2013.02.15 10:03:24 -05'00'

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: 2013.02.15 10:42:58 -05'00'

References

Reference for 37000 lbs: *VV-Clevis applied load. (see power point slides/data) – [Added by P. Rogoff]*

[1] Analysis of TF Outer Leg, Han Zhang, Calculation Number NSTXU-CALC-132-04,

Reference for standard brackets/chairs and chair loads:

[2] NSTX-CALC-13-001-00 Rev 1 Global Model – Model Description, Mesh Generation, Results, Peter H. Titus

Reference for PF4/5

[3] Analysis of Existing and Upgrade PF4/5 Coils and Supports – With Alternating Columns. NSTX-CALC-12-05-00 Rev 0 P. Titus March 2011

Reference for Criteria:

[4] NSTX Structural Design Criteria Document, NSTX_DesCrit_IZ_080103.doc I. Zatz

[5] NSTX Design Point Sept 8 2009 http://www.pppl.gov/~neumeyer/NSTX_CSU/Design_Point.html

[6] Fusion Engineering and Design 54 (2001) 275–319, Engineering design of the National Spherical Torus, Experiment, C. Neumeyer et.al, Page 286 quotes 100,000 lbs for the total vacuum vessel weight

DCPS Input:

In deriving the net loads on the vertical columns, it is assumed that all the magnetic loads sum to zero, and that the centerstack load inventory - PF1 a,b, upper and lower, and the OH, is supported by the pedestal, and an equal and opposite load is imposed on the vessel legs. The net load from the coils is 53445 lbs [5], rounded up to 60,000 lbs The tokamak is assumed to weigh 100,000 lbs [6]. The net dead load is 160,000 lbs or 40,000 lbs per column, or 177935 N per column. An initial setting for the DCPS would be to limit the sum of PF1a,b, U&L, and OH Upward vertical loading to 60,000 lbs. Upward vertical loading of the inner PF coils supported by the pedestal implies an additional downward loading on the vessel support columns of an equal magnitude.

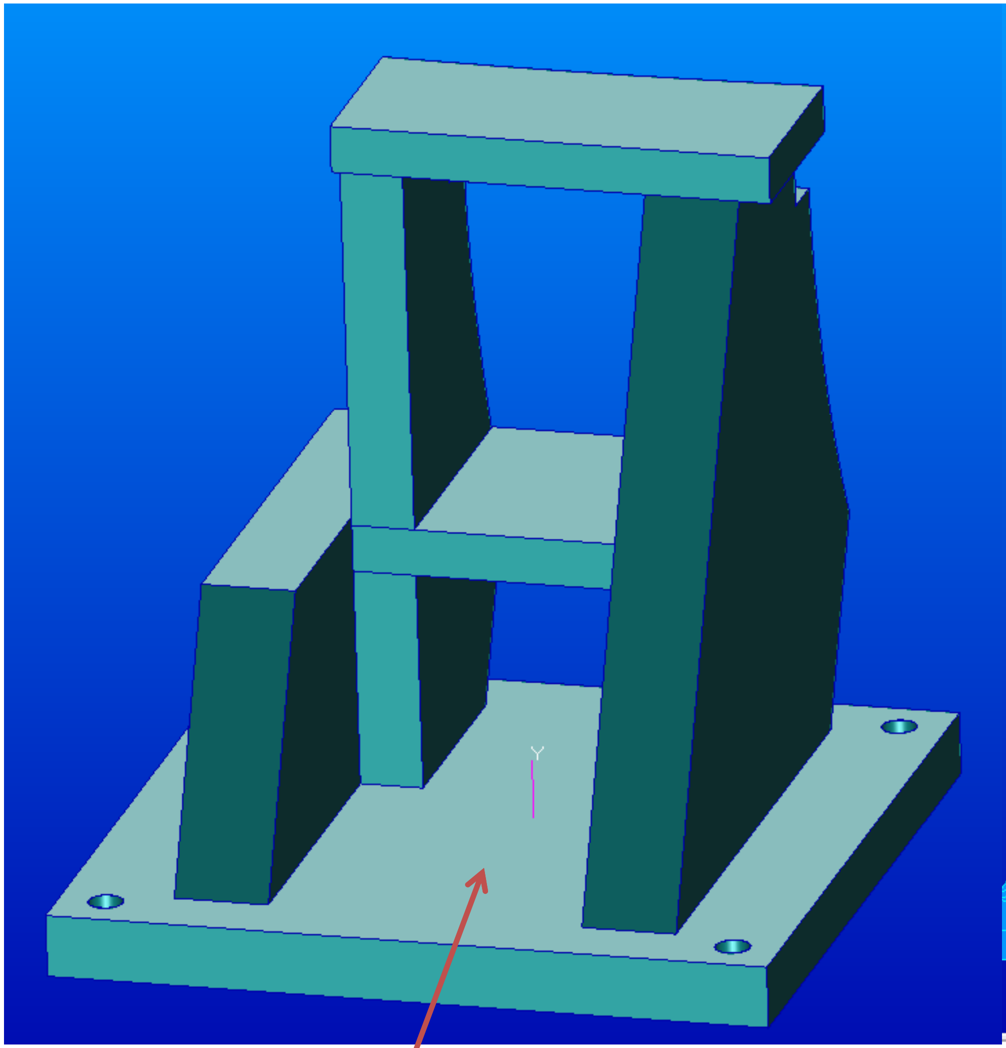
The total mechanical load applied to all simulation runs is 40,000 lbs for the 90 degree VV sector. This is based on the Ref (5) and (6) requirements. [Added by P. Rogoff]

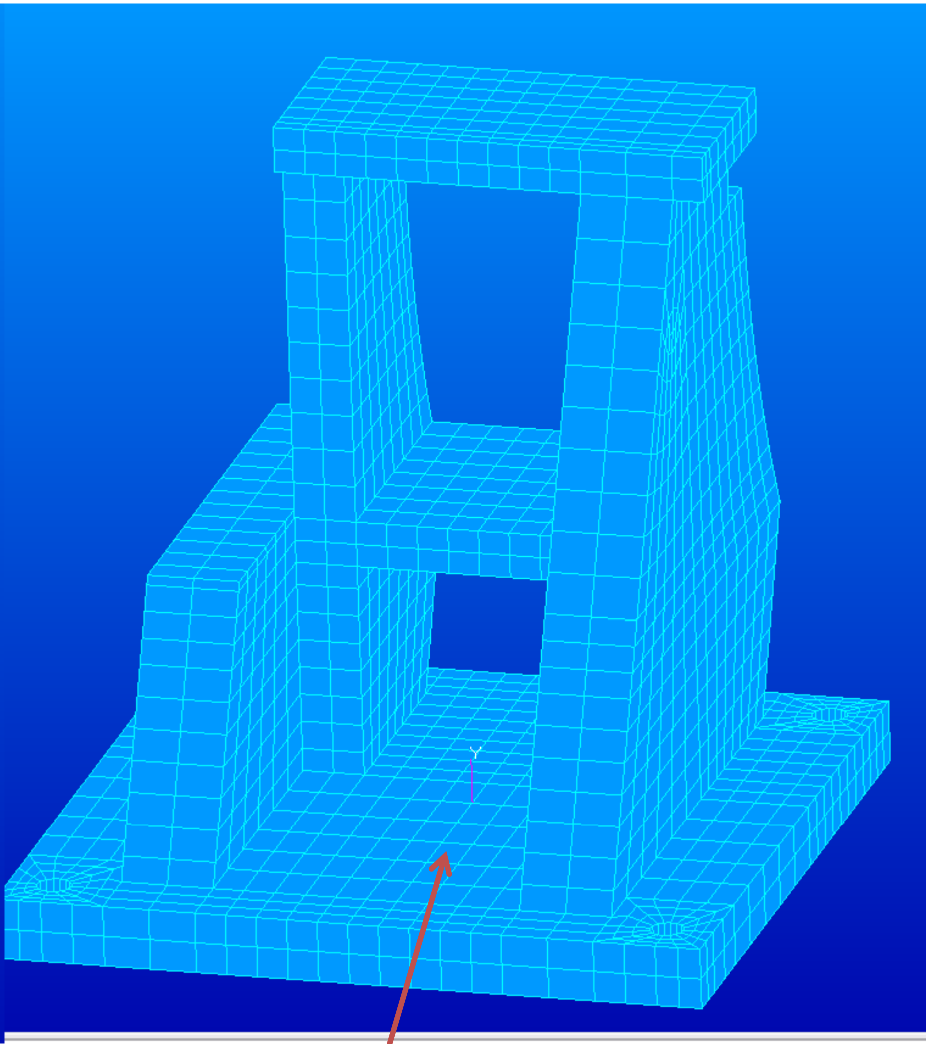
NSTXU PROJECT

Vessel-Support-Chair.ppt

Complete simulation justification

P.R. March 2012



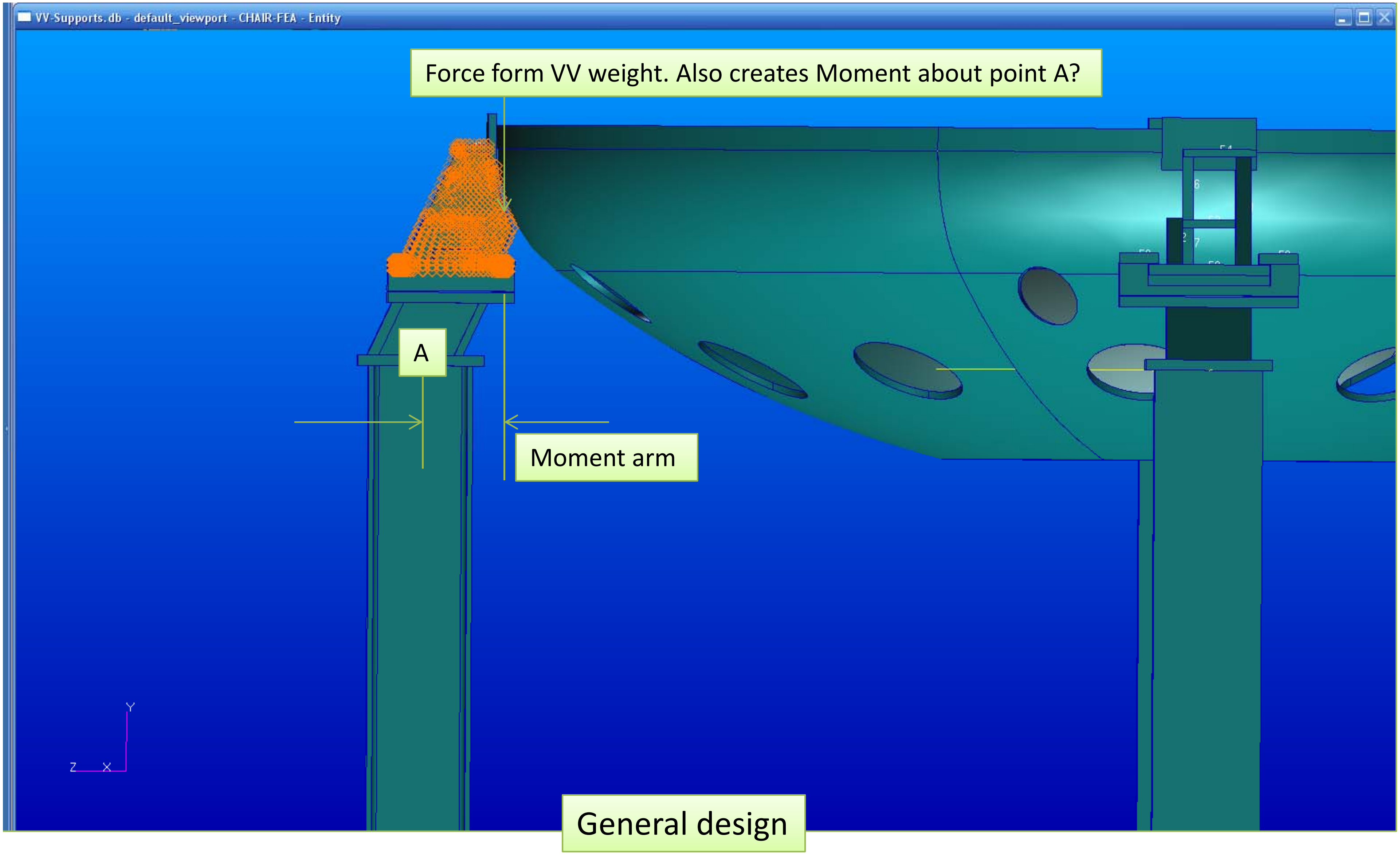


VV-support-attach-011812.igs

Nastran FEA all Hex8 elements

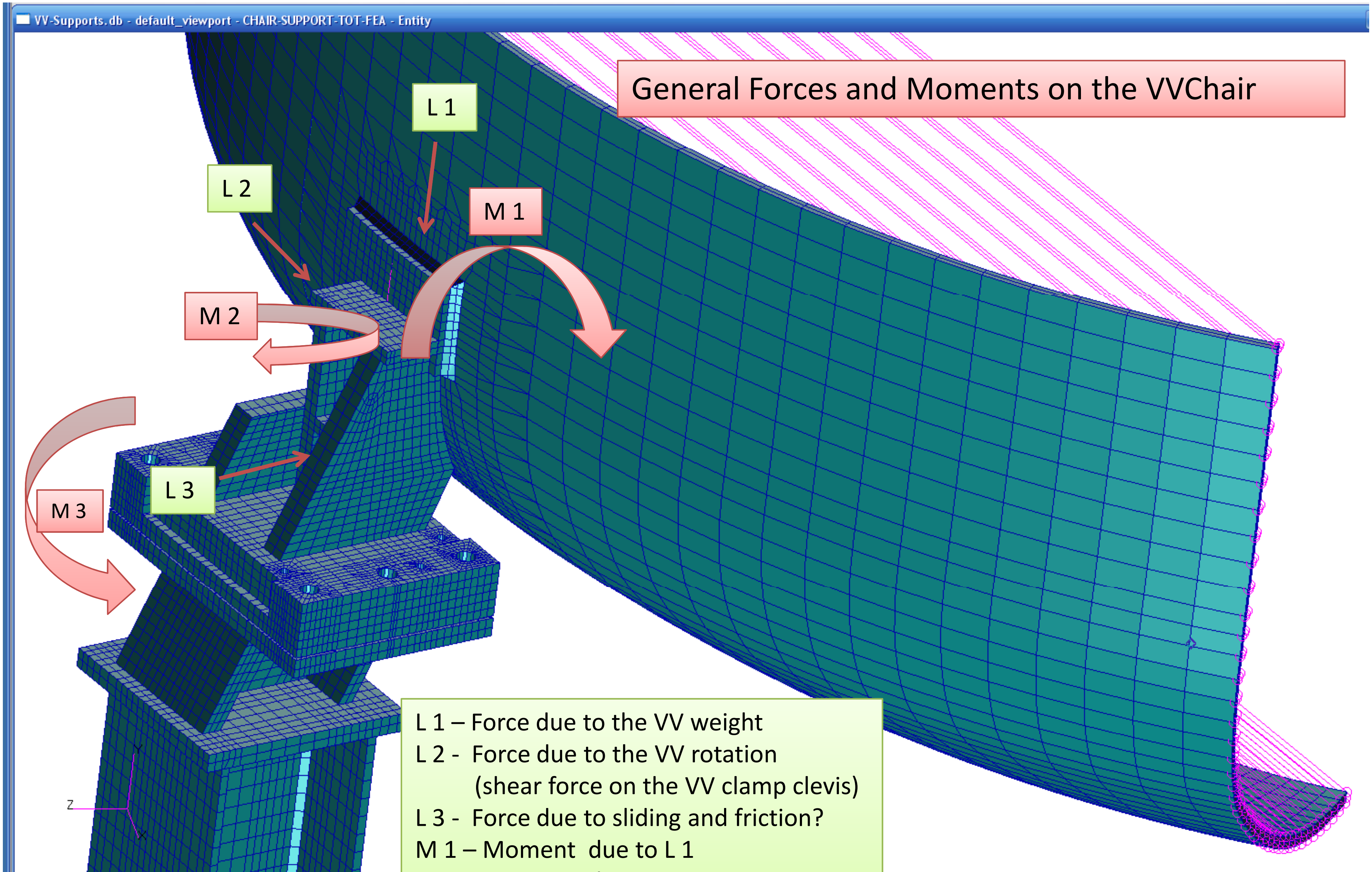
Question: How is this welded, assume weld size = 3/8 inch?

Redesigned VV support chair



VV – Chair support configuration
[Complete geometry provided by M. Smith, 1/2012]

General Forces and Moments on the VVChair



L 1 – Force due to the VV weight
L 2 - Force due to the VV rotation
(shear force on the VV clamp clevis)
L 3 - Force due to sliding and friction?
M 1 – Moment due to L 1
M 2 – Moment due to L 2
M 3 – Moment due to L 3

Note: Force L 3 is a function of "L 1" and the coefficient of friction at the slide plate/cradle interface

Complete model input file set-up

Note RUN #5 chosen for the demonstration (other runs are similar).

```
$ NASTRAN input file created by the MSC FEA 2010.1.2 64-Bit input file
$ translator on March     08, 2012 at 12:06:33.
$ Nonlinear Static Analysis, Database
SOL 106
CEND
TITLE = MSC.NASTRAN JOB CREATED ON 17-FEB-12 AT 08:48:38
ECHO = NONE
MPC = 20134
SUBCASE 1
   TITLE=Non-Linear
   SUBTITLE=RUN5-Loads
   NLPARM = 1
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM    POST    0
PARAM    AUTOSPC YES
PARAM   PRTMAXIM YES
NLPARM   1       10              AUTO    5       25      P       NO
                .01
```

General Executive deck

```
$ Elements and Element Properties for region : Sliding-Gaps
PGAP     2       0.              3.+7    500.    1.+7    .3      .3
        -1.
$ Pset: "Sliding-Gaps" will be imported as: "pgap.2"
CGAP     94612   2       102192  127702                          2
CGAP     94613   2       102195  128013                          2
```

GAPs definition

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

    NUMBER OF GRID      POINTS   =    34815

    NUMBER OF CGAP      ELEMENTS =      660
    NUMBER OF CHEXA     ELEMENTS =    23184
    NUMBER OF CPENTA    ELEMENTS =     1336
    NUMBER OF CTETRA    ELEMENTS =       33

    NUMBER OF RBE2      ELEMENTS =      977
17-FEB-12 AT 08:48:38                            MARCH    8, 2012
```

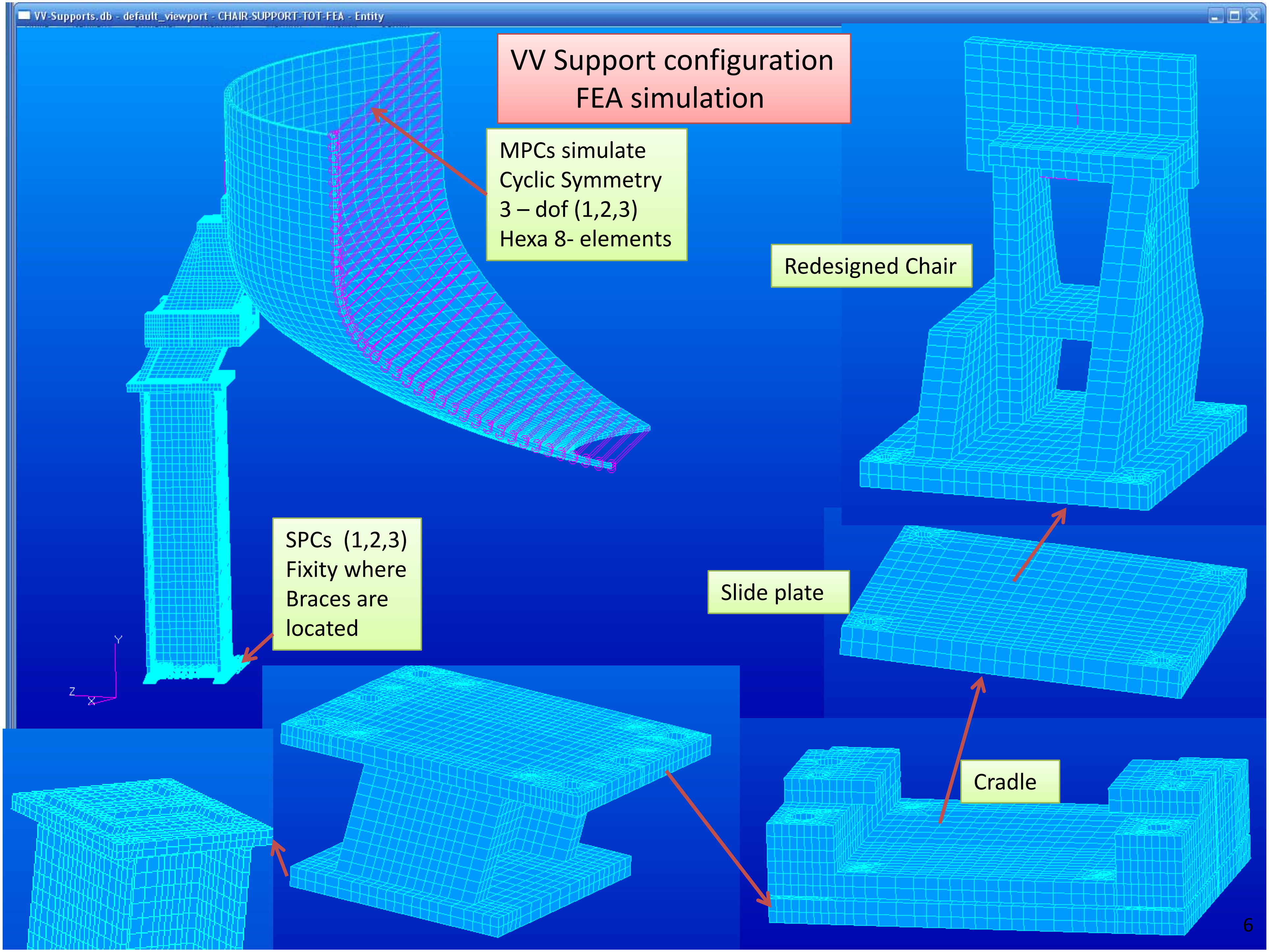
Simulation grid and element statistics

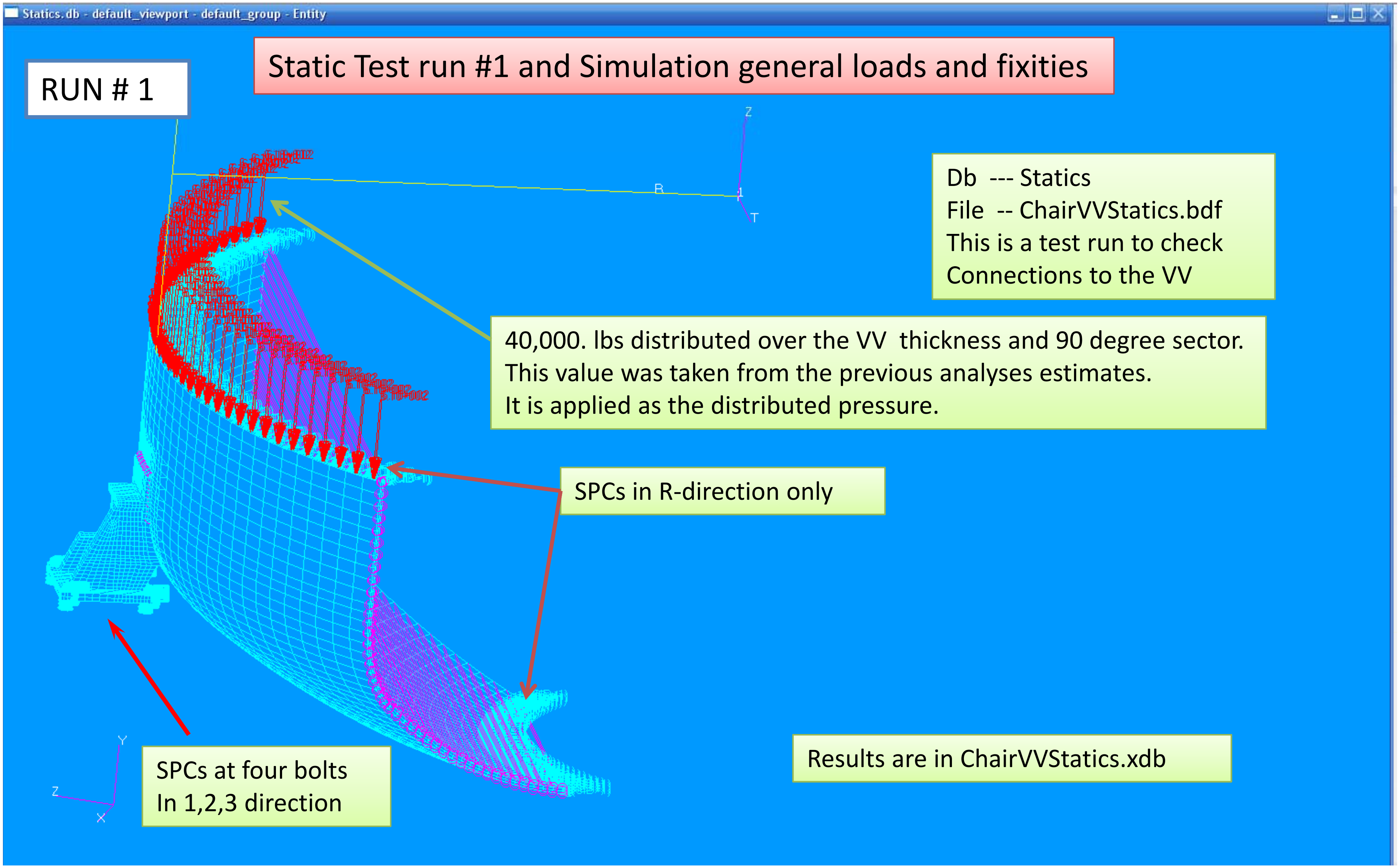
Total load due to the Pressure

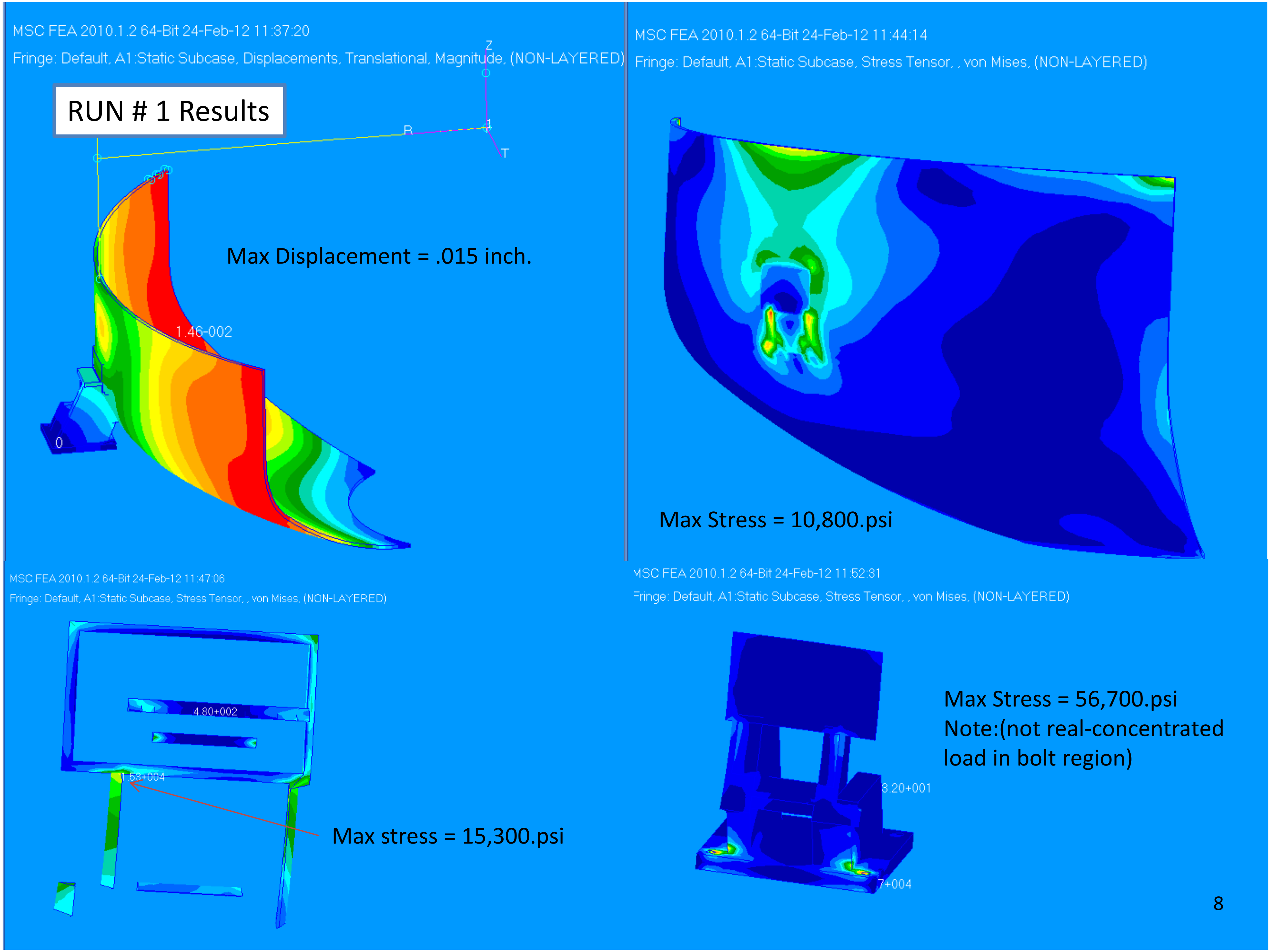
Clevis loads (only for run #5)

```
                                         OLOAD     RESULTANT
  SUBCASE/     LOAD
  DAREA ID     TYPE       T1            T2            T3            R1            R2            R3
0        1      FX    3.576299E+04     ----          ----          ----      2.319899E+06  1.403519E+06
                FY        ----     -3.998594E+04     ----      2.221206E+06      ----     -9.220052E+05
                FZ        ----         ----     -9.487278E+03  3.723282E+05  1.632618E+05      ----
                MX        ----         ----          ----      0.000000E+00      ----          ----
                MY        ----         ----          ----          ----      0.000000E+00      ----
                MZ        ----         ----          ----          ----          ----      0.000000E+00
              TOTALS  3.576299E+04 -3.998594E+04 -9.487278E+03  2.593534E+06  2.483160E+06  4.815134E+05
1    MSC.NASTRAN JOB CREATED ON 17-FEB-12 AT 08:48:38                       MARCH    8, 2012  MSC.NASTRAN
```

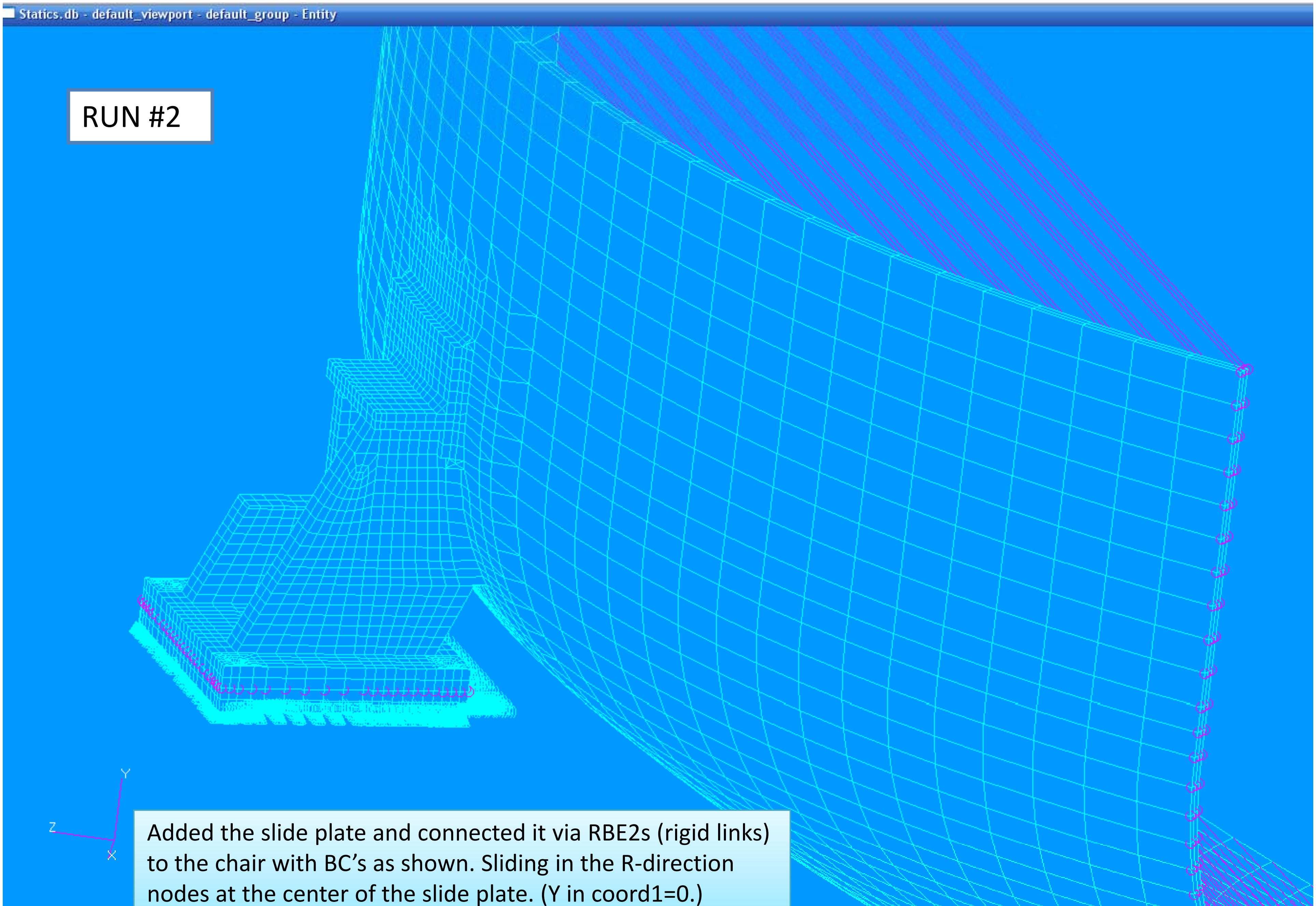
Resultant applied loads from the .f06 output


VV-Supports.db - default_viewport - CHAIR-SUPPORT-TOT-FEA - Entity
VV Support configuration FEA simulation
MPCs simulate Cyclic Symmetry 3 – dof (1,2,3) Hexa 8- elements
Redesigned Chair
SPCs (1,2,3) Fixity where Braces are located
Slide plate
Cradle

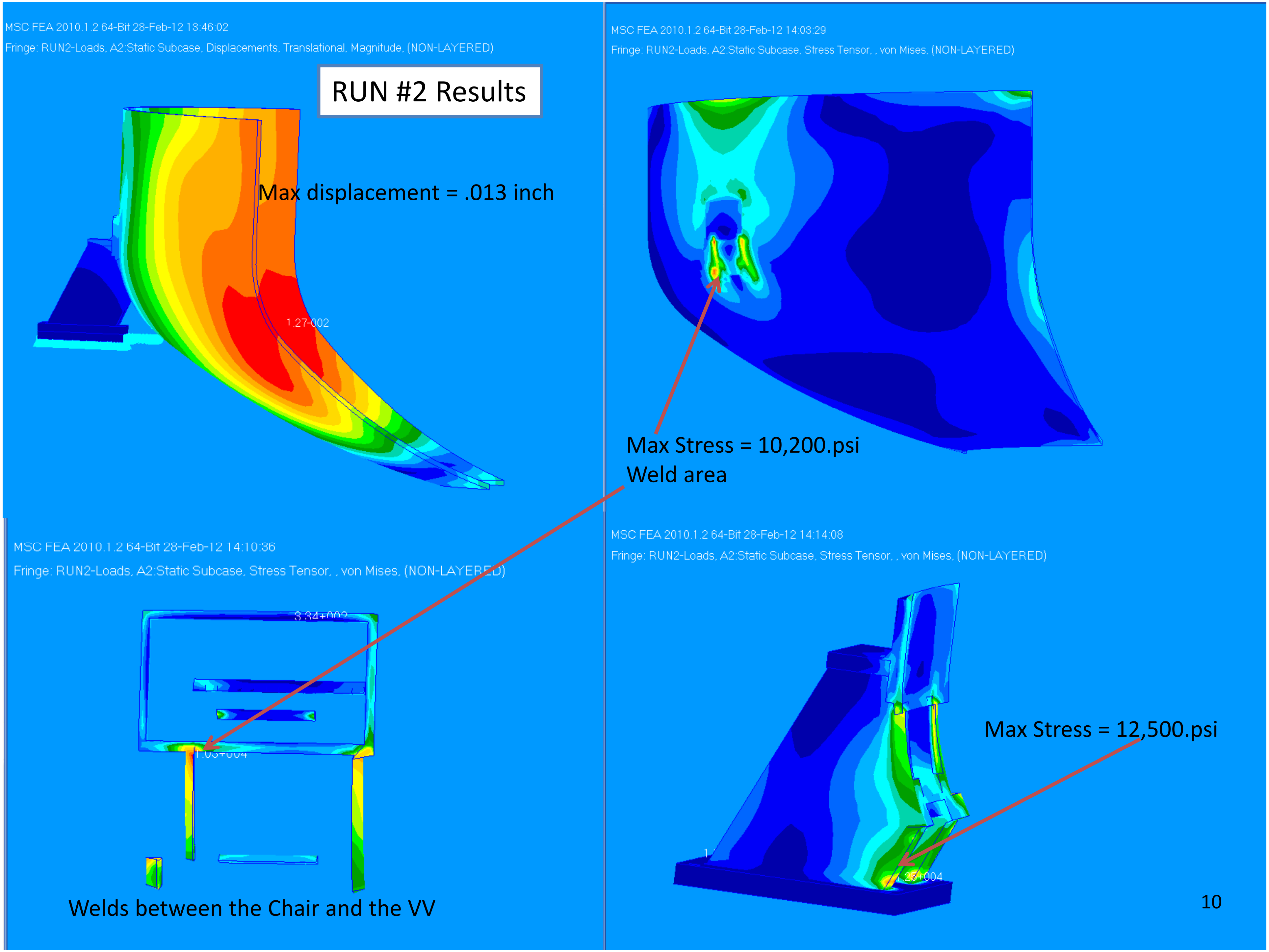

Statics.db - default_viewport - default_group - Entity
RUN # 1
Static Test run #1 and Simulation general loads and fixities
Db --- Statics
File -- ChairVVStatics.bdf
This is a test run to check
Connections to the VV
40,000. lbs distributed over the VV thickness and 90 degree sector.
This value was taken from the previous analyses estimates.
It is applied as the distributed pressure.
SPCs in R-direction only
SPCs at four bolts
In 1,2,3 direction
Results are in ChairVVStatics.xdb

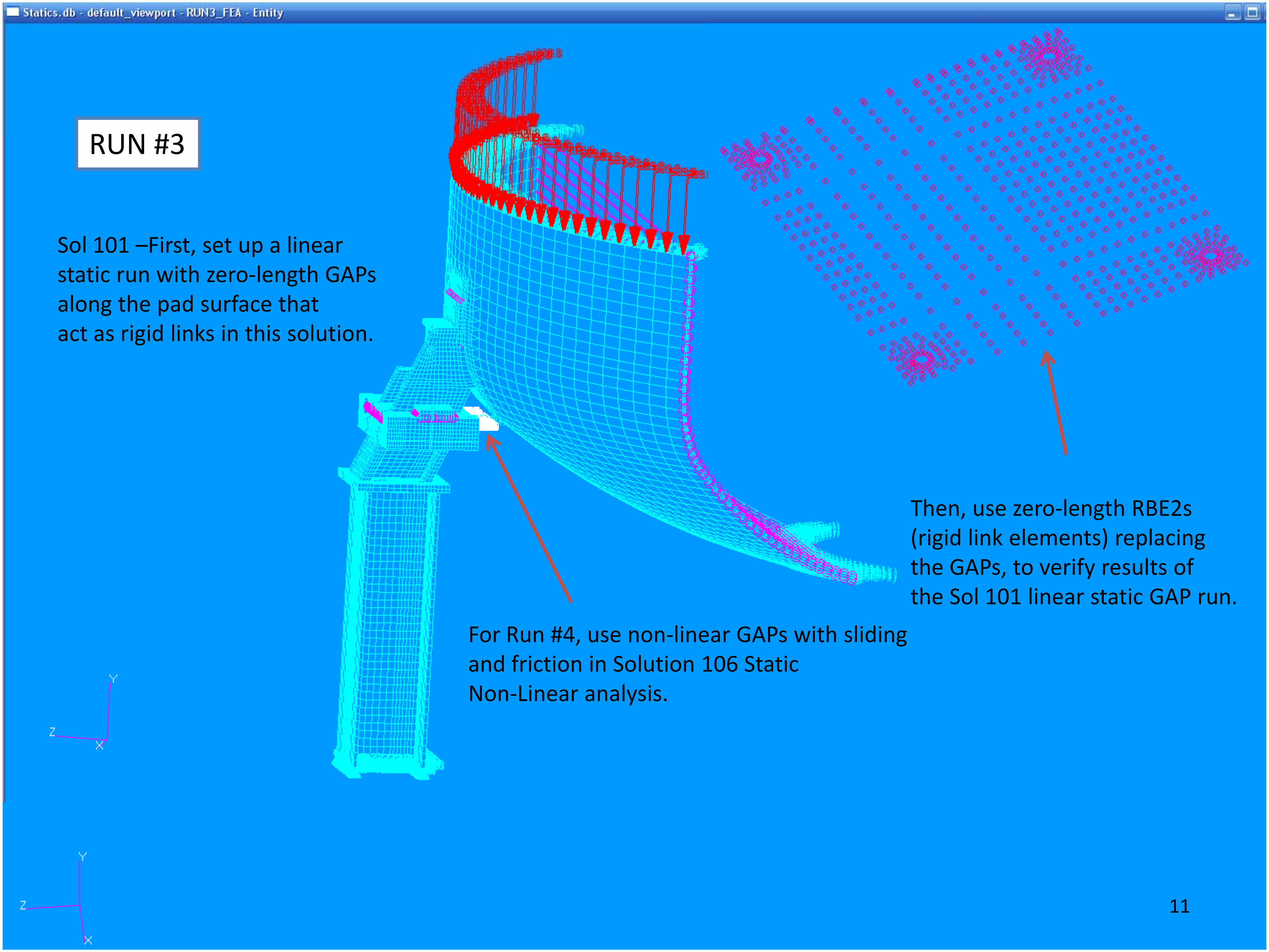

MSC FEA 2010.1.2 64-Bit 24-Feb-12 11:37:20
Fringe: Default, A1:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
RUN # 1 Results
Max Displacement = .015 inch.
1.46-002
0
MSC FEA 2010.1.2 64-Bit 24-Feb-12 11:44:14
Fringe: Default, A1:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Max Stress = 10,800.psi
MSC FEA 2010.1.2 64-Bit 24-Feb-12 11:47:06
Fringe: Default, A1:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
4.80+002
1.53+004
Max stress = 15,300.psi
MSC FEA 2010.1.2 64-Bit 24-Feb-12 11:52:31
Fringe: Default, A1:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Max Stress = 56,700.psi
Note:(not real-concentrated
load in bolt region)
3.20+001
8

RUN #2

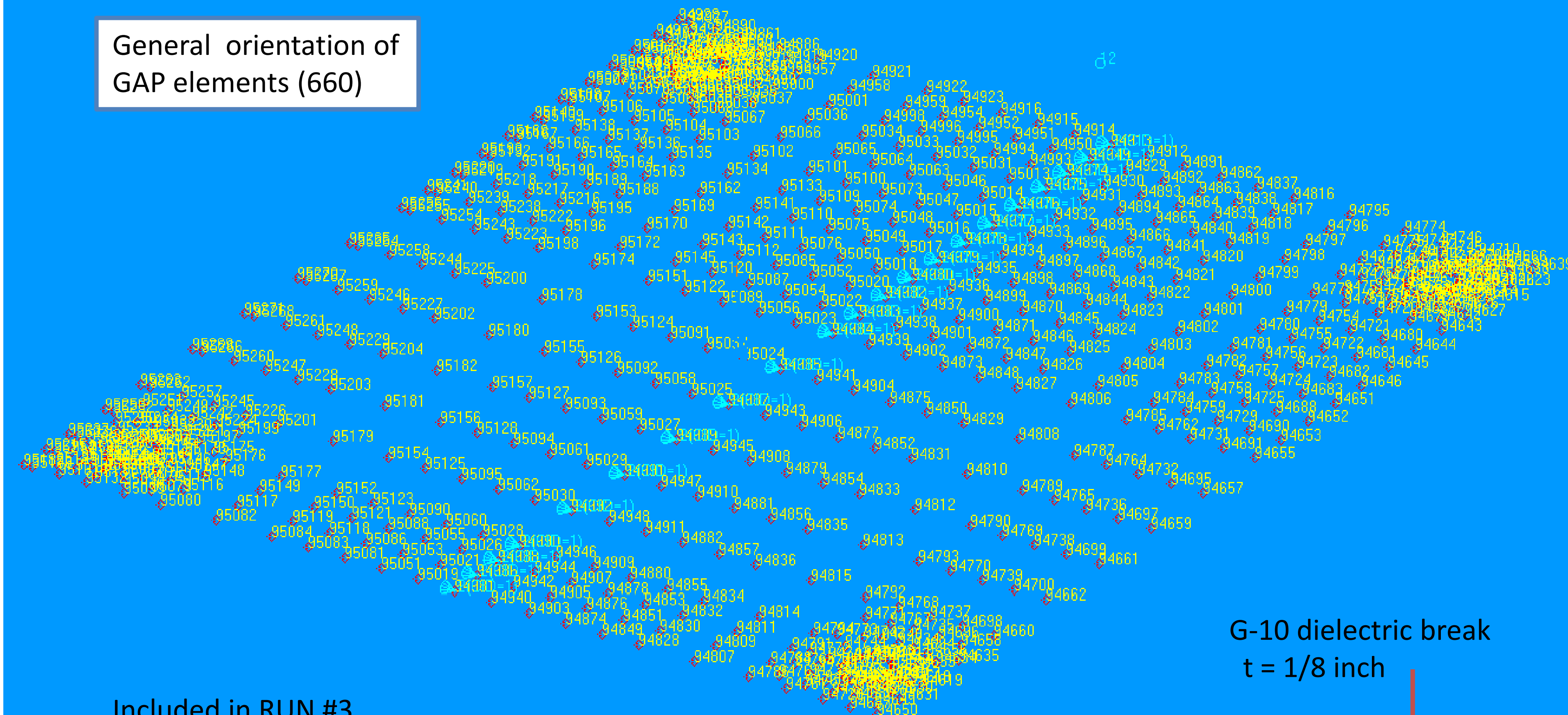


Added the slide plate and connected it via RBE2s (rigid links) to the chair with BC's as shown. Sliding in the R-direction nodes at the center of the slide plate. (Y in coord1=0.)


RUN #2 Results
MSC FEA 2010.1.2 64-Bit 28-Feb-12 13:46:02
Fringe: RUN2-Loads, A2:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Max displacement = .013 inch
1.27-002
MSC FEA 2010.1.2 64-Bit 28-Feb-12 14:03:29
Fringe: RUN2-Loads, A2:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Max Stress = 10,200.psi
Weld area
MSC FEA 2010.1.2 64-Bit 28-Feb-12 14:10:36
Fringe: RUN2-Loads, A2:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Welds between the Chair and the VV
MSC FEA 2010.1.2 64-Bit 28-Feb-12 14:14:08
Fringe: RUN2-Loads, A2:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Max Stress = 12,500.psi
10


Statics.db - default_viewport - RUN3_FEA - Entity
RUN #3
Sol 101 –First, set up a linear static run with zero-length GAPs along the pad surface that act as rigid links in this solution.
Then, use zero-length RBE2s (rigid link elements) replacing the GAPs, to verify results of the Sol 101 linear static GAP run.
For Run #4, use non-linear GAPs with sliding and friction in Solution 106 Static Non-Linear analysis.
Y
Z
X
Y
Z
X

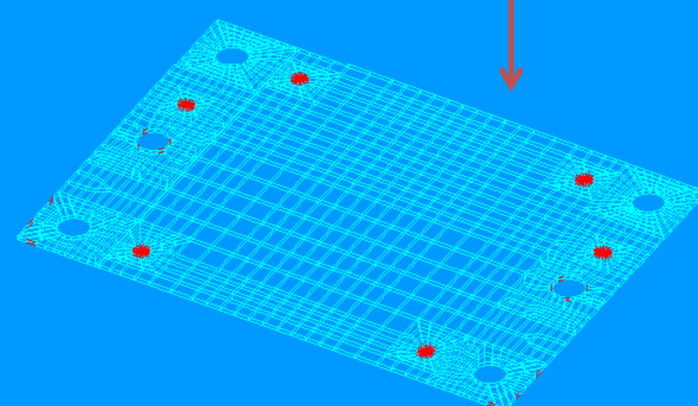
General orientation of GAP elements (660)



Included in RUN #3

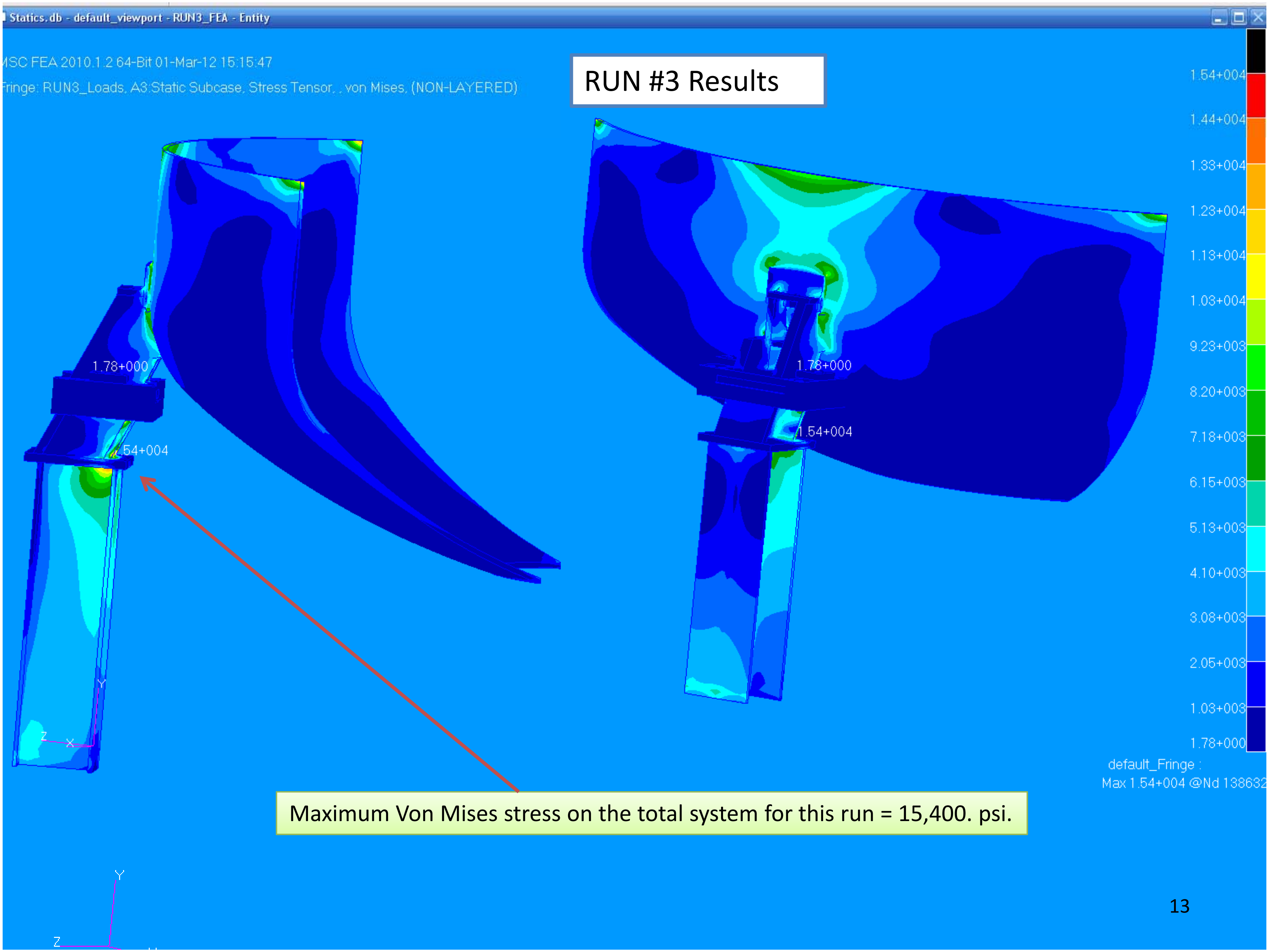
```
PARAM    POST     0
PARAM   PRTMAXIM YES
$ Elements and Element Properties for region : Sliding-Gaps
PGAP     1        0.             3.+7    500.    1.+7    .3      .3
        -1.
$ Pset: "Sliding-Gaps" will be imported as: "pgap.1"
CGAP     94612   1       102192  127702  0.      -1.      0.
CGAP     94613   1       102195  128013  0.      -1.      0.
CGAP     94614   1       102199  128016  0.      -1.      0.
CGAP     94615   1       102201  128015  0.      -1.      0.
CGAP     94616   1       102204  131457  0.      -1.      0.
```

G-10 dielectric break
t = 1/8 inch



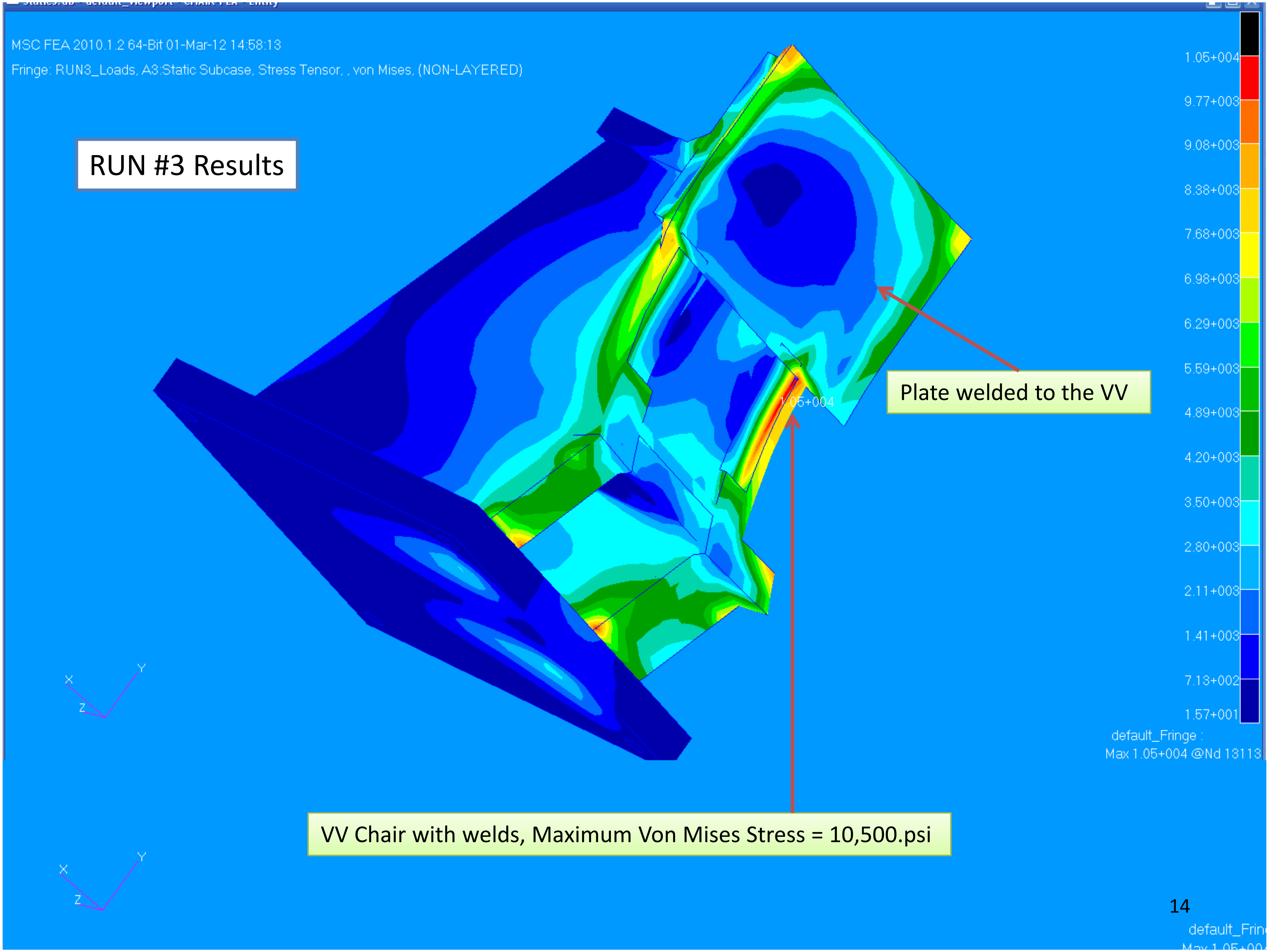
Between the cradle and Vertical support

RUN #3 Results



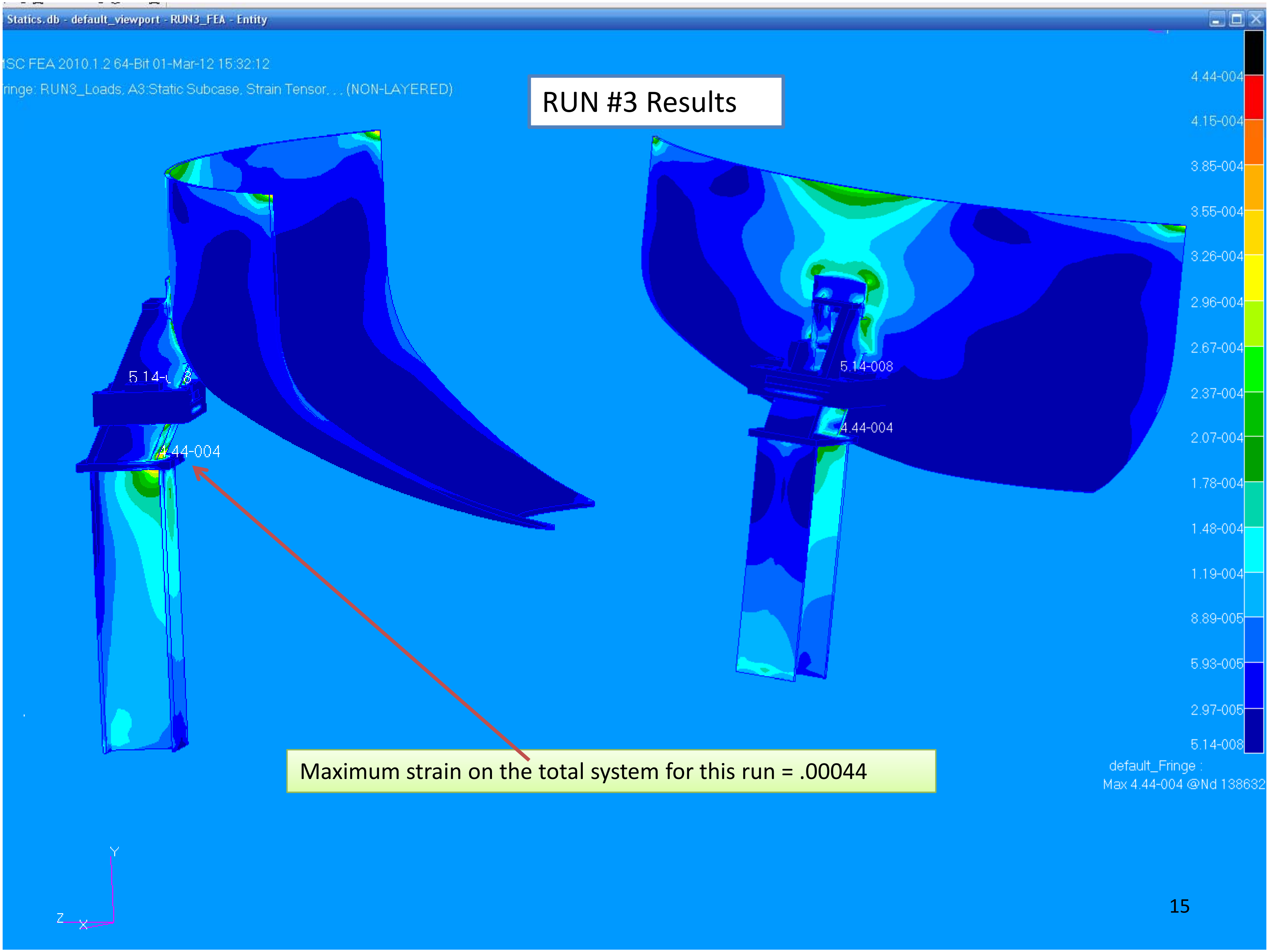
Maximum Von Mises stress on the total system for this run = 15,400. psi.

RUN #3 Results

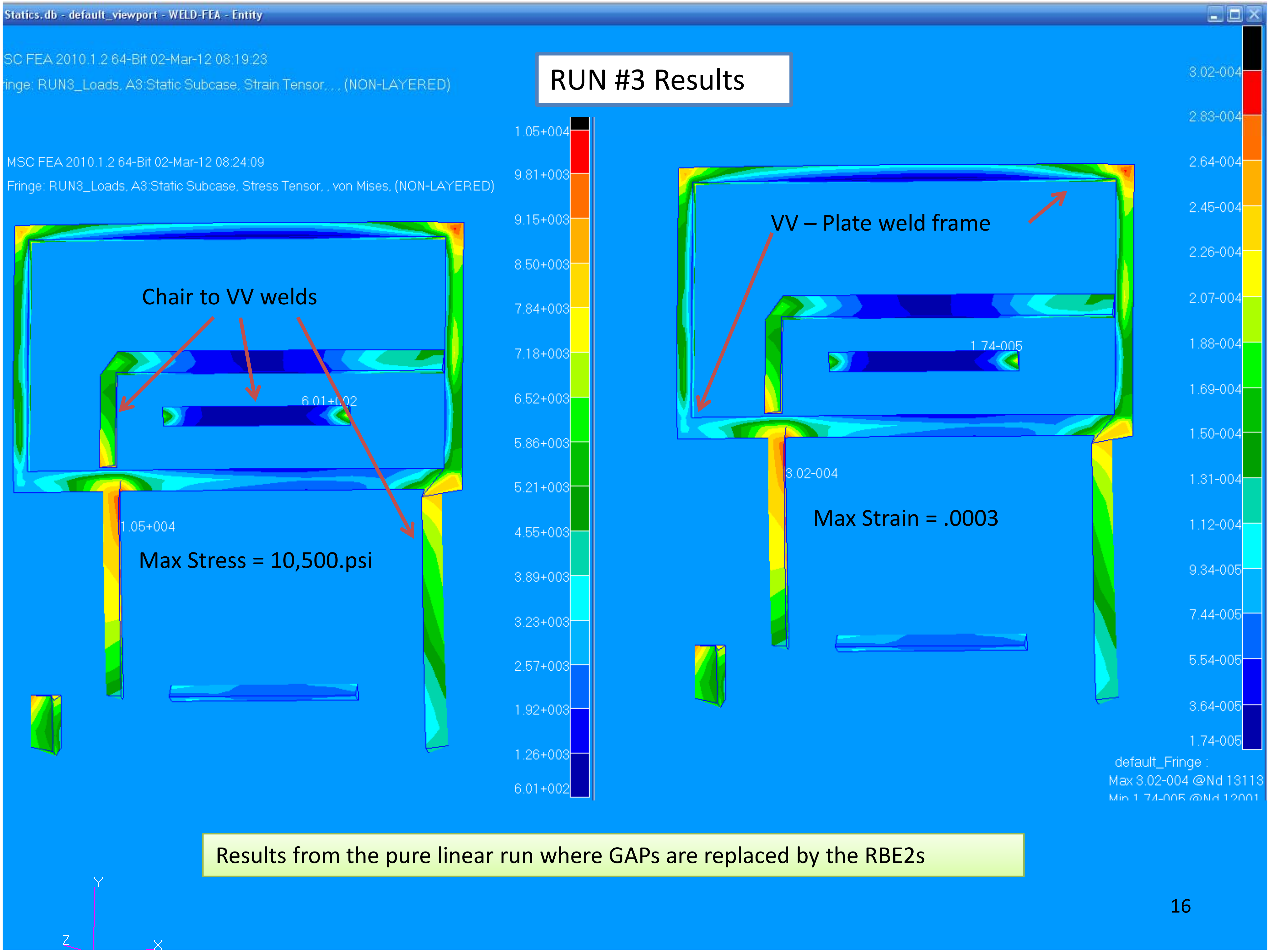


VV Chair with welds, Maximum Von Mises Stress = 10,500.psi

RUN #3 Results

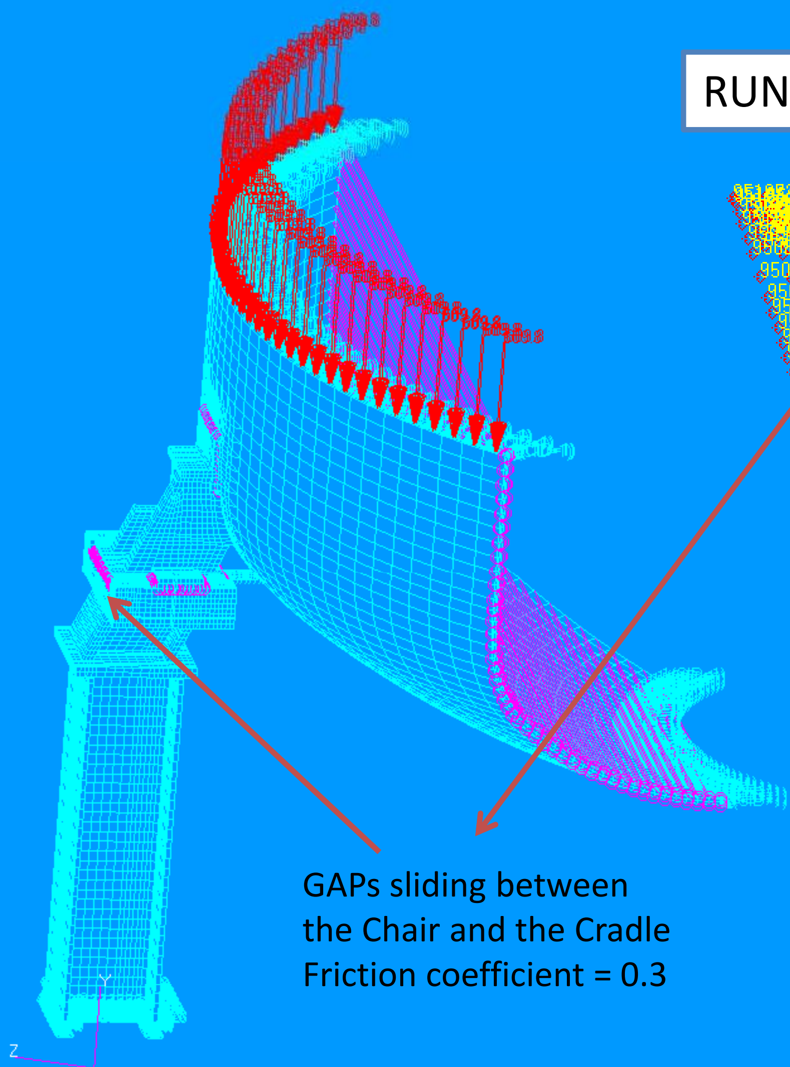


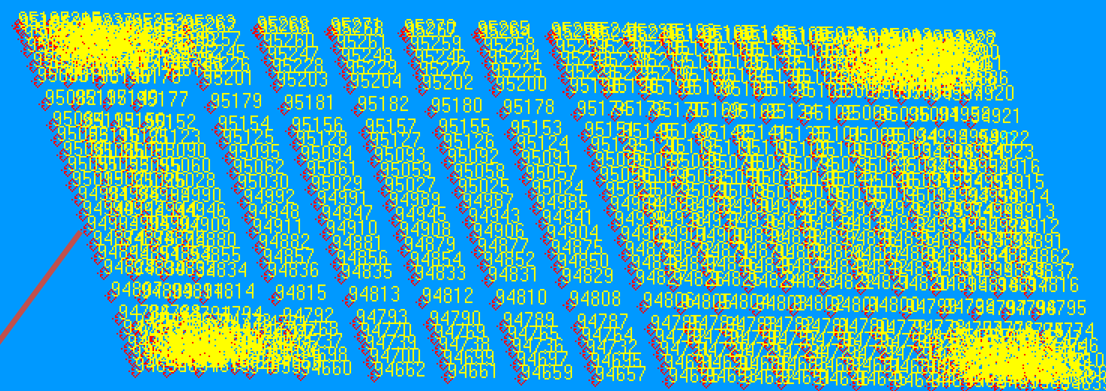
Maximum strain on the total system for this run = .00044


Statics.db - default_viewport - WELD-FEA - Entity
RUN #3 Results
SC FEA 2010.1.2 64-Bit 02-Mar-12 08:19:23
inge: RUN3_Loads, A3:Static Subcase, Strain Tensor, , , (NON-LAYERED)
MSC FEA 2010.1.2 64-Bit 02-Mar-12 08:24:09
Fringe: RUN3_Loads, A3:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)
Chair to VV welds
6.01+002
1.05+004
Max Stress = 10,500.psi
1.05+004
9.81+003
9.15+003
8.50+003
7.84+003
7.18+003
6.52+003
5.86+003
5.21+003
4.55+003
3.89+003
3.23+003
2.57+003
1.92+003
1.26+003
6.01+002
VV – Plate weld frame
1.74-005
3.02-004
Max Strain = .0003
3.02-004
2.83-004
2.64-004
2.45-004
2.26-004
2.07-004
1.88-004
1.69-004
1.50-004
1.31-004
1.12-004
9.34-005
7.44-005
5.54-005
3.64-005
1.74-005
default_Fringe :
Max 3.02-004 @Nd 13113
Results from the pure linear run where GAPs are replaced by the RBE2s
Y
Z
X

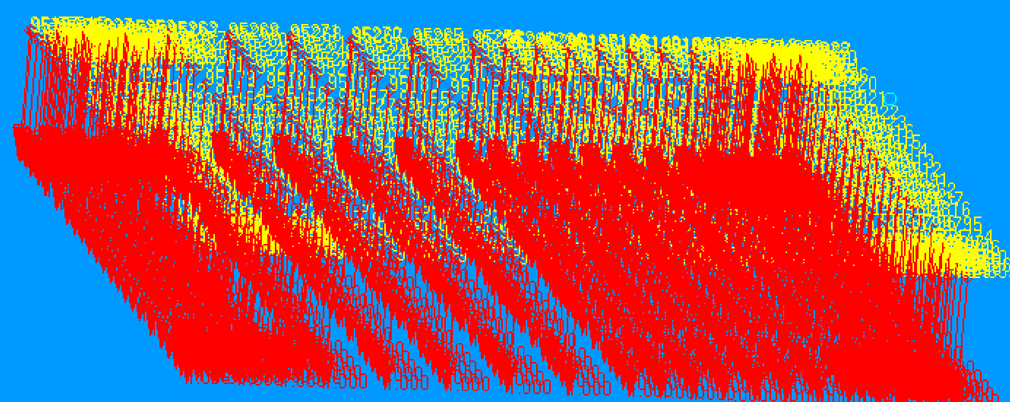
Statics.db - default_viewport - RUN4-106NL-FEA - Entity

RUN #4 Non Linear Static Solution (106)





GAPs sliding between
the Chair and the Cradle
Friction coefficient = 0.3



GAPs orientation in the -Y direction

	Type	Scale Factor	Priority
Slide_Fic_cen	Displacement	1.	Add
Support_Bot	Displacement	1.	Add
VV-Fixity	Displacement	1.	Add
VV-Fixity-Bot	Displacement	1.	Add
Pres40000	Pressure	1.	Add

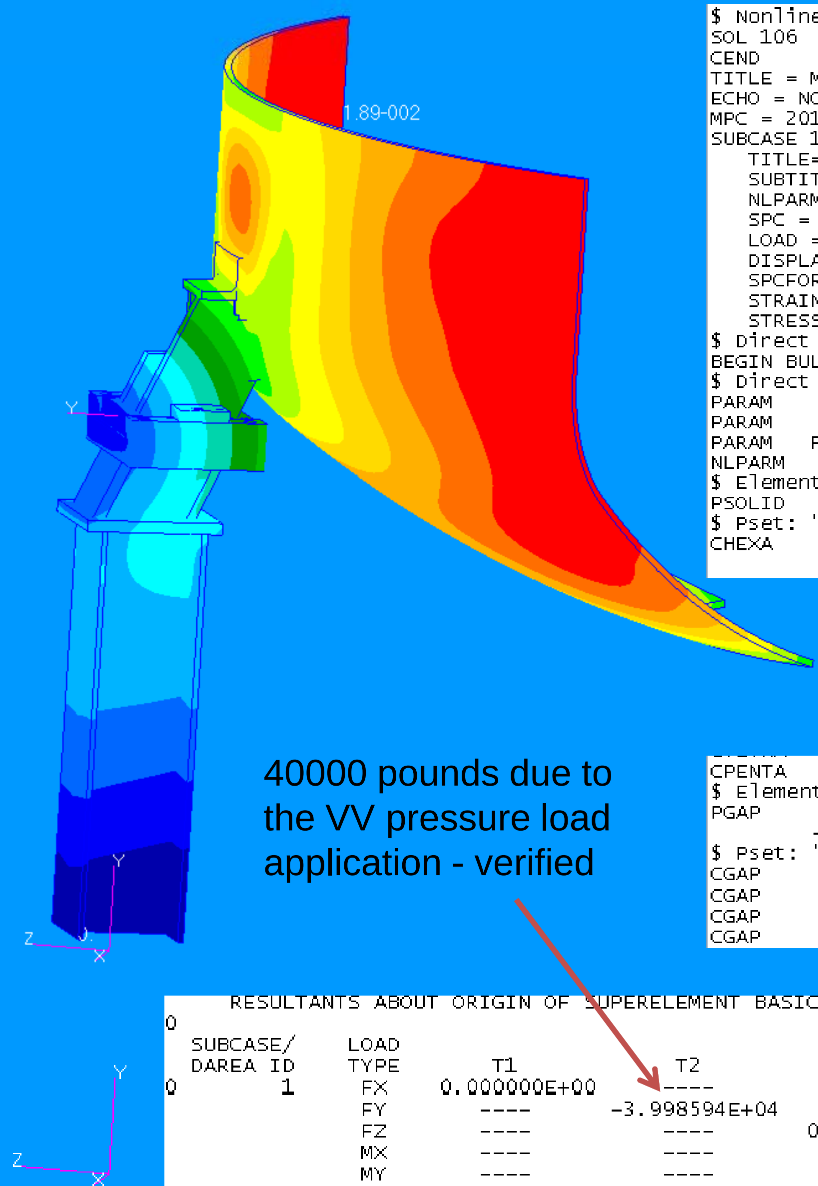
Load Case “RUN4-Loads”

Statics.db - default_viewport - RUN4-106NL-FEA - Entity

RUN #4 -Results

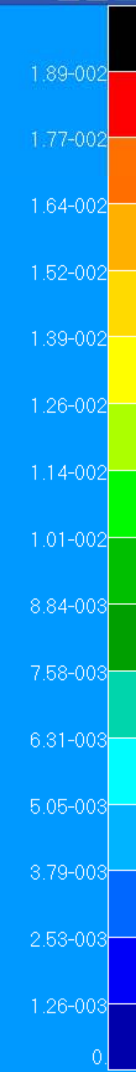
MSC FEA 2010.1.2 64-Bit 06-Mar-12 10:02:14

Fringe: RUN4-Loads, A4:Non-linear: 100. % of Load, Displacements, Translational, Magnitude, (NON-LAYERED)




```
$ Nonlinear Static Analysis, Database
SOL 106
CEND
TITLE = MSC.NASTRAN JOB CREATED ON 17-FEB-12 AT 08:48:38
ECHO = NONE
MPC = 20134
SUBCASE 1
   TITLE=Non Linear
   SUBTITLE=RUN4-Loads
   NLPARM = 1
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM    POST     0
PARAM    AUTOSPC  YES
PARAM   PRTMAXIM  YES
NLPARM   1        10               AUTO     5        25                NO
$ Elements and Element Properties for region : HexWedTet
PSOLID   1        1        0
$ Pset: "HexWedTet" will be imported as: "psolid.1"
CHEXA    2793     1        14407    14420    14419    14406    14606    14627
         14626    14605
```

Group – RUN4-106NL-FEA
Input – Run4-106NL-fea.bdf
Load -- RUN4-Loads
Output – Run4-106NL-fea.xdb

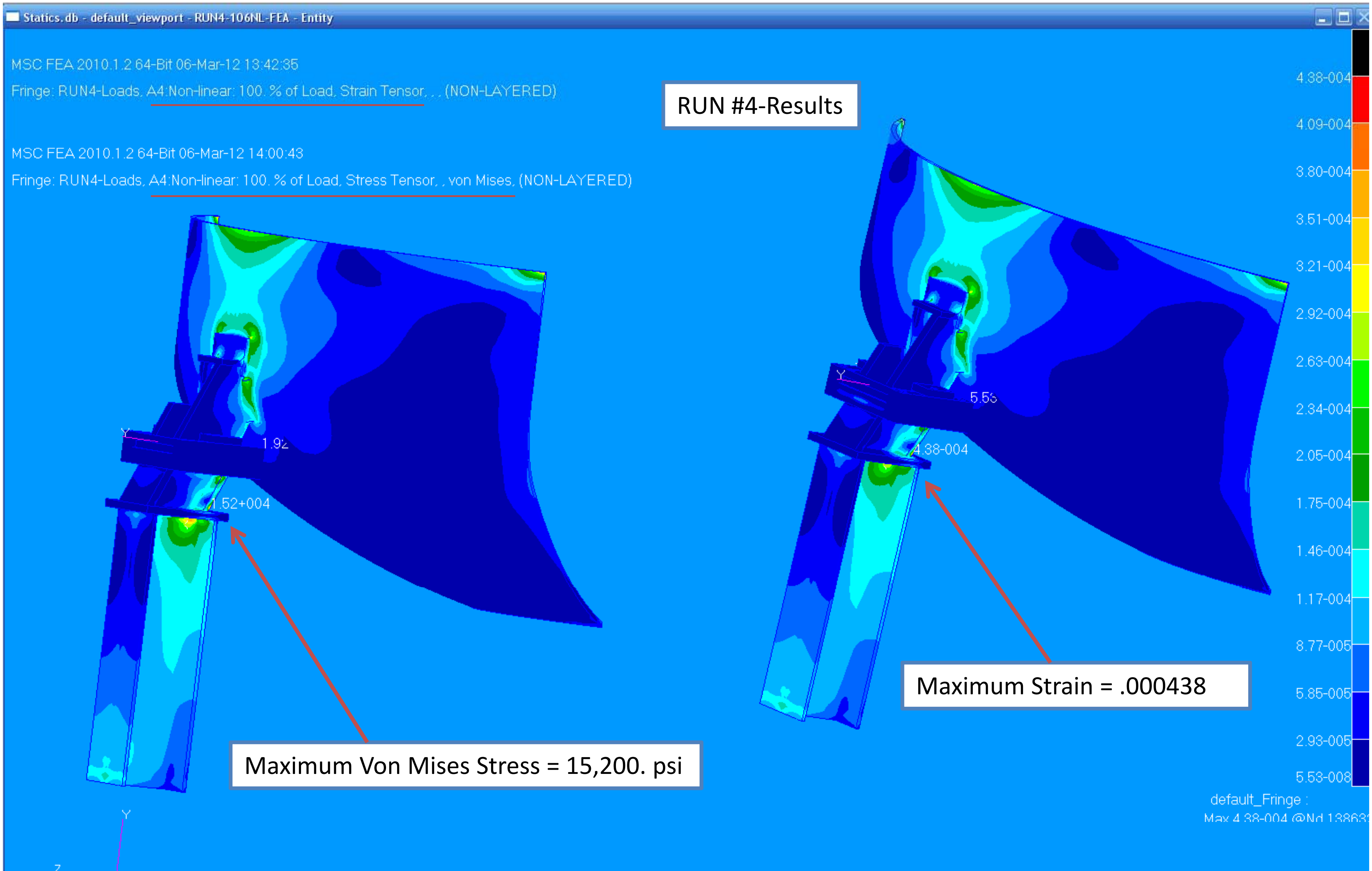



```
CPENTA   95277    1        16149    12736    12733    16152    12576    12573
$ Elements and Element Properties for region : Sliding-Gaps
PGAP     2        0.                3.+7     500.     1.+7     .3       .3
        -1.
$ Pset: "Sliding-Gaps" will be imported as: "pgap.2"
CGAP     94612    2        102192   127702                              2
CGAP     94613    2        102195   128013                              2
CGAP     94614    2        102199   128016                              2
CGAP     94615    2        102201   128015                              2
```

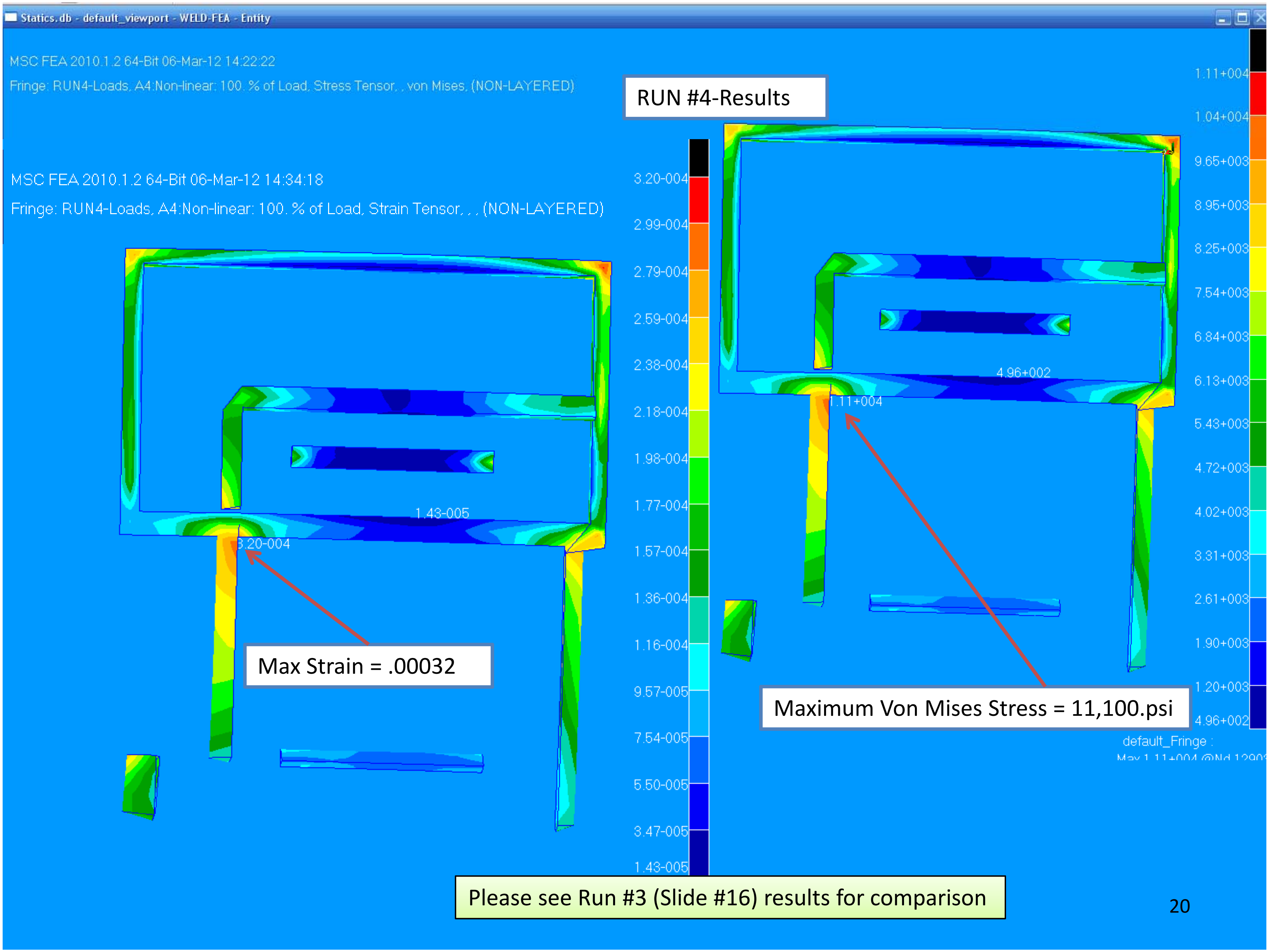
default_Fringe :
Max 1.89-002 @Nd 5909

40000 pounds due to the VV pressure load application - verified

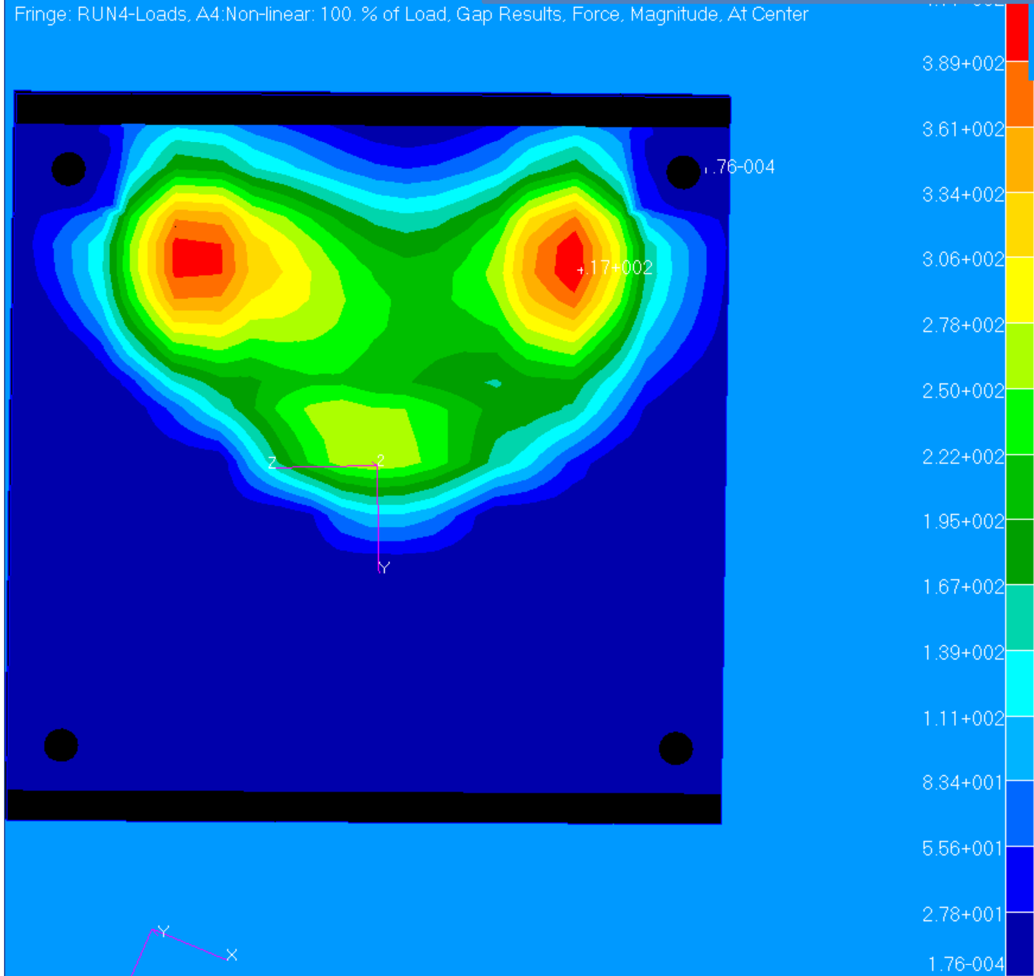
```
     RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                                    OLOAD    RESULTANT
  SUBCASE/      LOAD
  DAREA ID      TYPE       T1            T2            T3            R1            R2            R3
0        1       FX    0.000000E+00     ----          ----          ----      0.000000E+00  0.000000E+00
                 FY       ----      -3.998594E+04     ----      2.221206E+06     ----      -9.220052E+05
                 FZ       ----          ----      0.000000E+00  0.000000E+00  0.000000E+00     ----
                 MX       ----          ----          ----      0.000000E+00     ----          ----
                 MY       ----          ----          ----          ----      0.000000E+00     ----
                 MZ       ----          ----          ----          ----          ----      0.000000E+00
               TOTALS  0.000000E+00 -3.998594E+04  0.000000E+00  2.221206E+06  0.000000E+00 -9.220052E+05
1    MSC.NASTRAN JOB CREATED ON 17-FEB-12 AT 08:48:38                      MARCH   6, 2012  MSC.NASTRAN  6/19
```



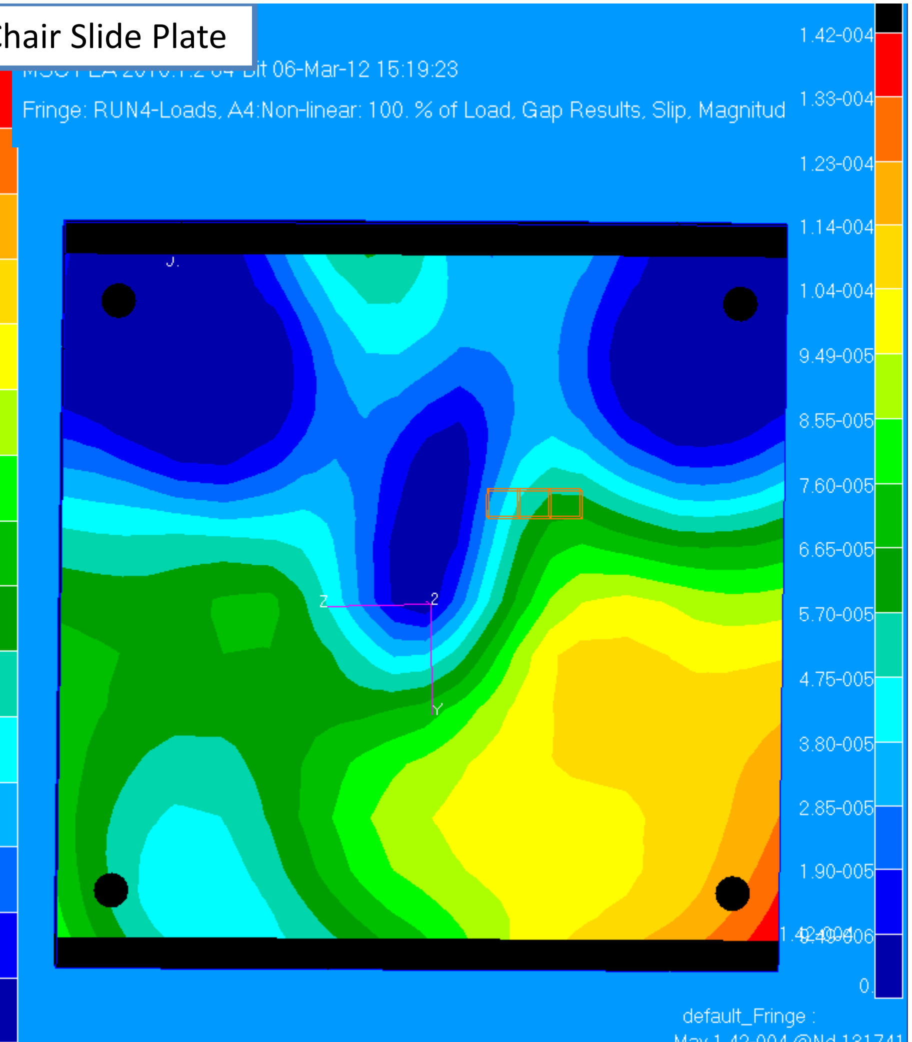
Note: These Non Linear results are very close to those calculated in linear RUN #3. This is actually expected because non-linear GAP sliding is small (See following slides). This confirms that a linear analysis can be used as effectively as a non-linear analysis for solving this calculation.


Statics.db - default_viewport - WELD-FEA - Entity
MSC FEA 2010.1.2 64-Bit 06-Mar-12 14:22:22
Fringe: RUN4-Loads, A4:Non-linear: 100. % of Load, Stress Tensor, , von Mises, (NON-LAYERED)
RUN #4-Results
MSC FEA 2010.1.2 64-Bit 06-Mar-12 14:34:18
Fringe: RUN4-Loads, A4:Non-linear: 100. % of Load, Strain Tensor, , , (NON-LAYERED)
1.43-005
3.20-004
Max Strain = .00032
3.20-004
2.99-004
2.79-004
2.59-004
2.38-004
2.18-004
1.98-004
1.77-004
1.57-004
1.36-004
1.16-004
9.57-005
7.54-005
5.50-005
3.47-005
1.43-005
4.96+002
1.11+004
Maximum Von Mises Stress = 11,100.psi
1.11+004
1.04+004
9.65+003
8.95+003
8.25+003
7.54+003
6.84+003
6.13+003
5.43+003
4.72+003
4.02+003
3.31+003
2.61+003
1.90+003
1.20+003
4.96+002
default_Fringe :
Please see Run #3 (Slide #16) results for comparison

RUN #4 – Load & Slip Results for Chair Slide Plate



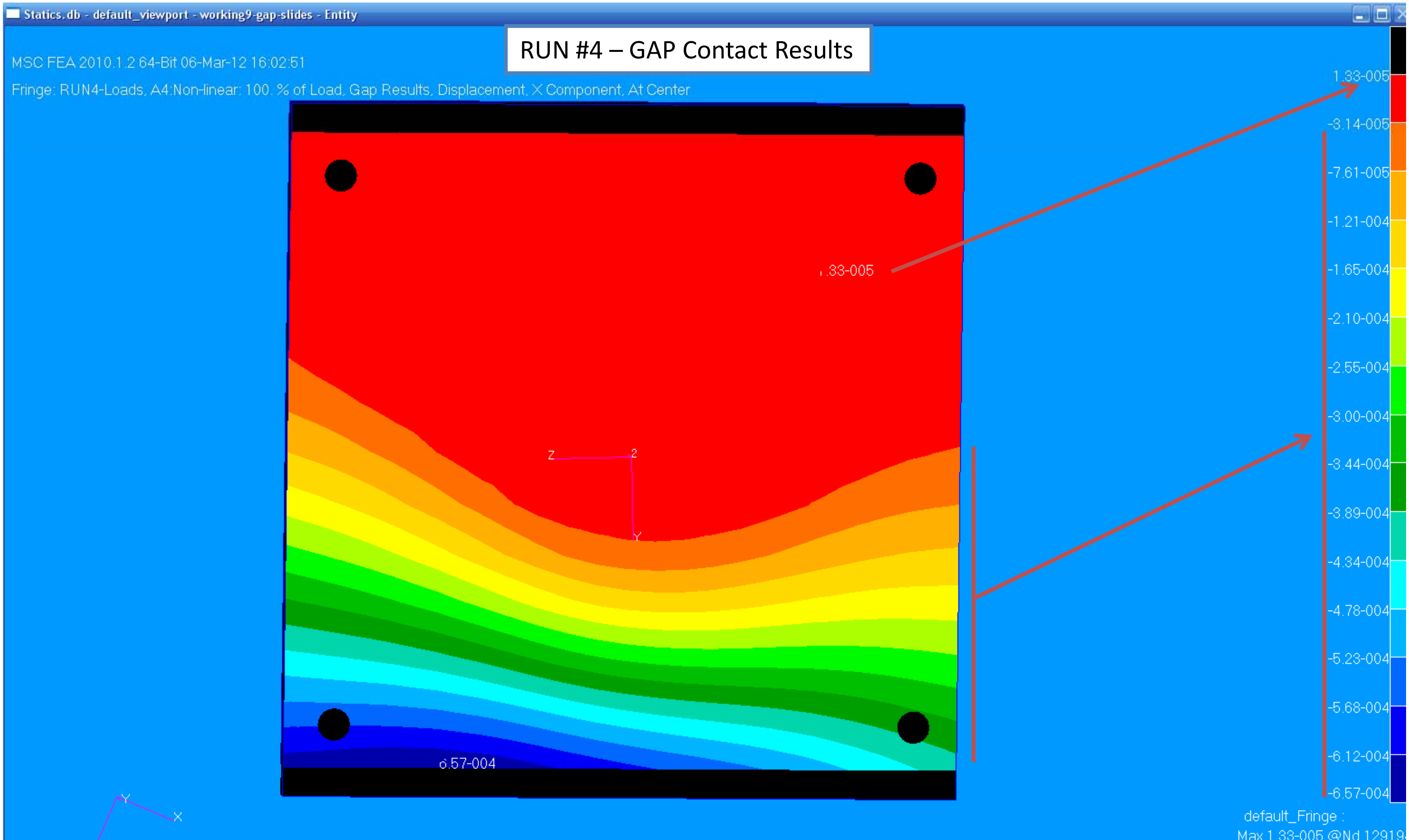
Gaps Load transfer distribution



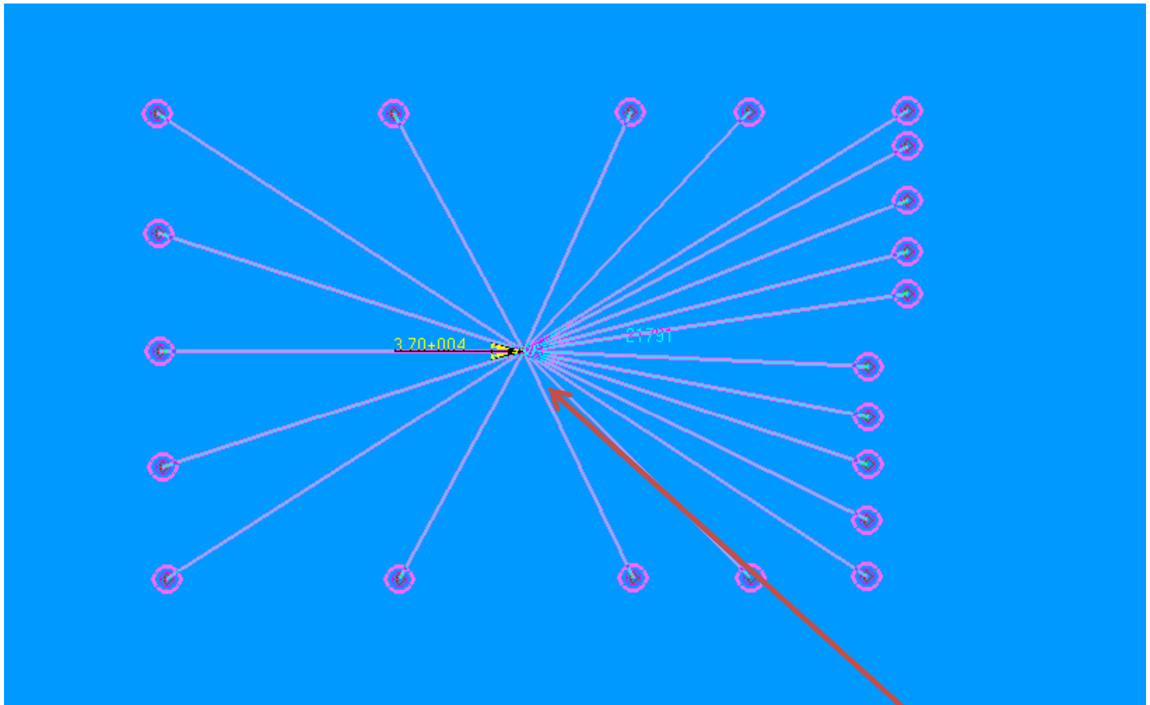
Gaps slipping distribution

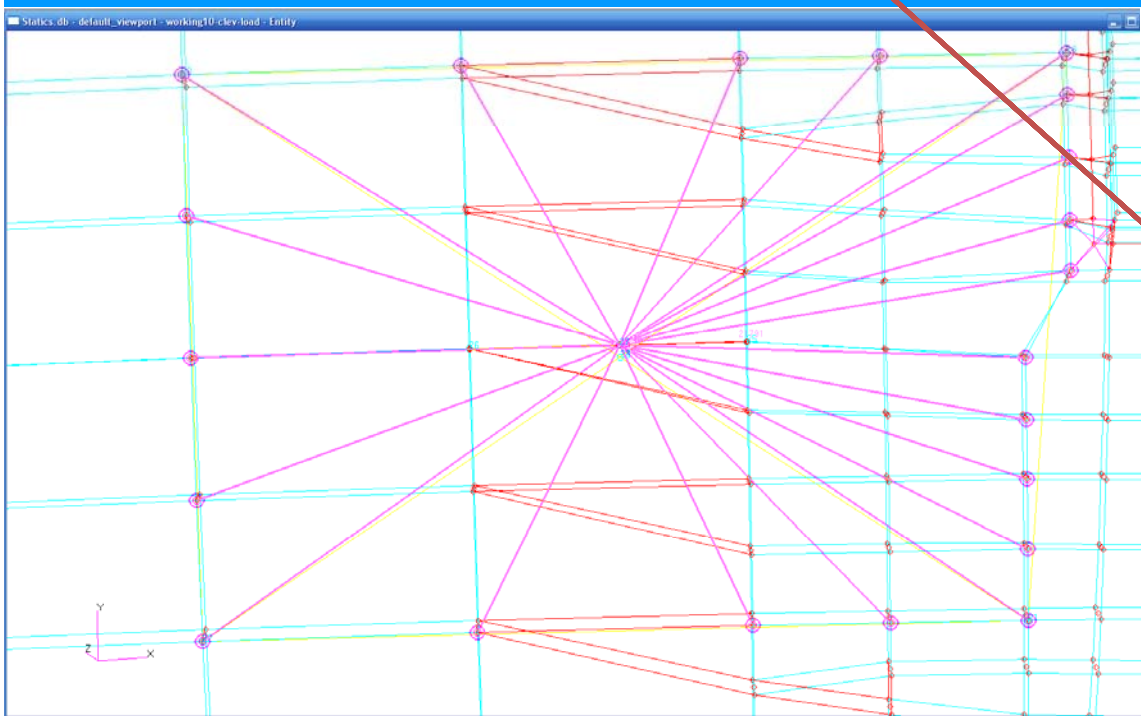
Note : These results are based on the coefficient of friction = 0.3 for GAP “Y” and “Z” transverse directions

RUN #4 – GAP Contact Results



Red area represents Gaps in contact (positive value X-direction of the Gaps) other values are negative and represent Gaps opening.



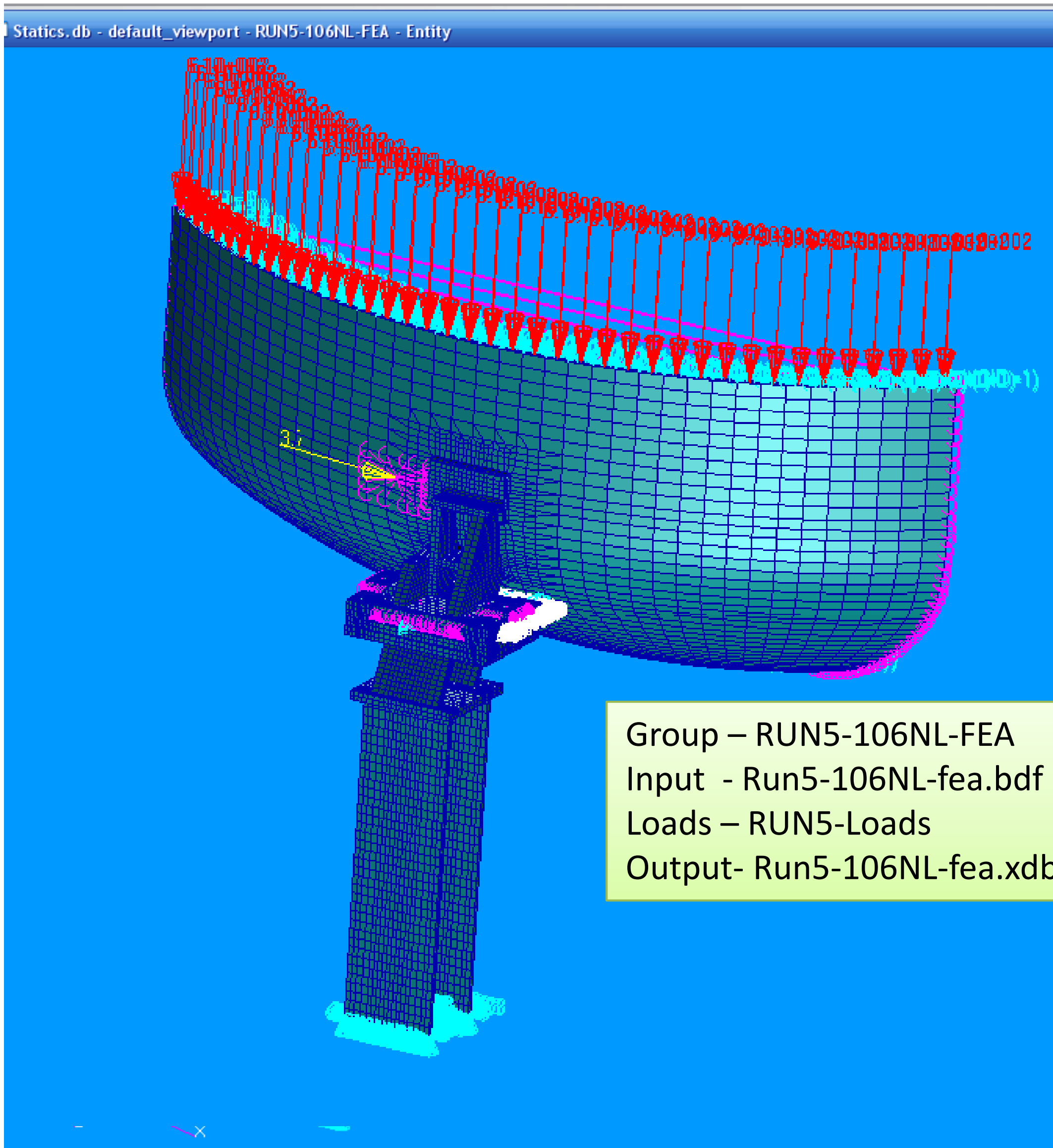


Changes for RUN #5

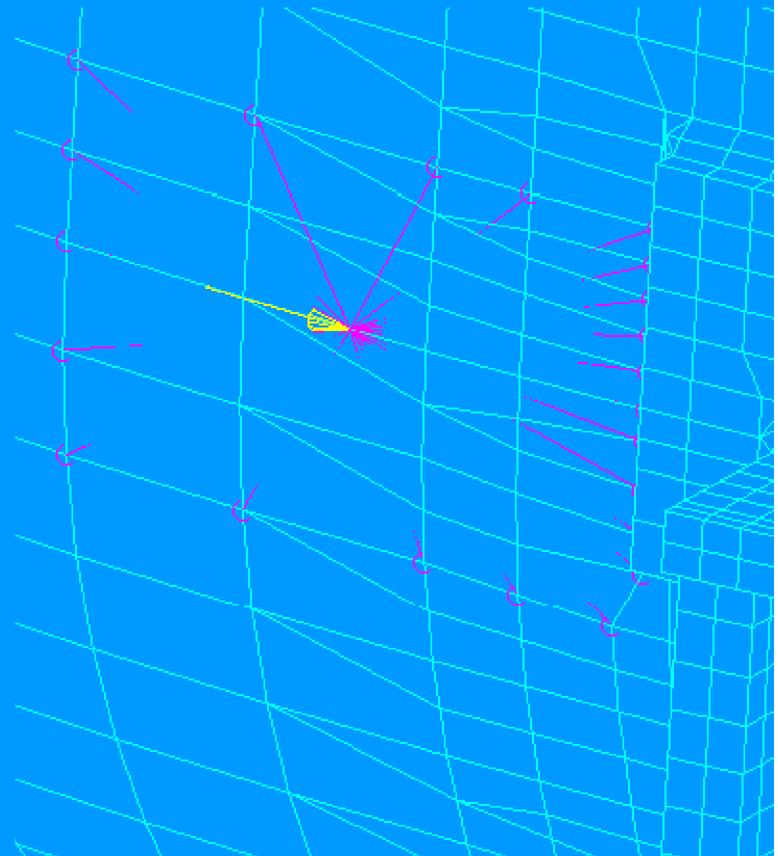
Introduction of the near-by Clevis force and its application is added to the Run #4 simulation.
The VV-chair was redesigned because of the interference created by the new TF coil link connections to the VV. Therefore, the clevis is placed very close to the VV-chair structure. The slides on the left show this closeness and the RIGID BODY formation to simulate this condition. This body consists of the central node to which outside nodes are connected rigidly.
Therefore, load applied at the central node distributes the force to the dependent nodes on the outside nodes which describe the maximum dimensions of the welded clevis.
A force equal to 37,000. lbs is applied at the central node using cylindrical Coordinate System #1 in the Theta – direction.

Central node is independent.

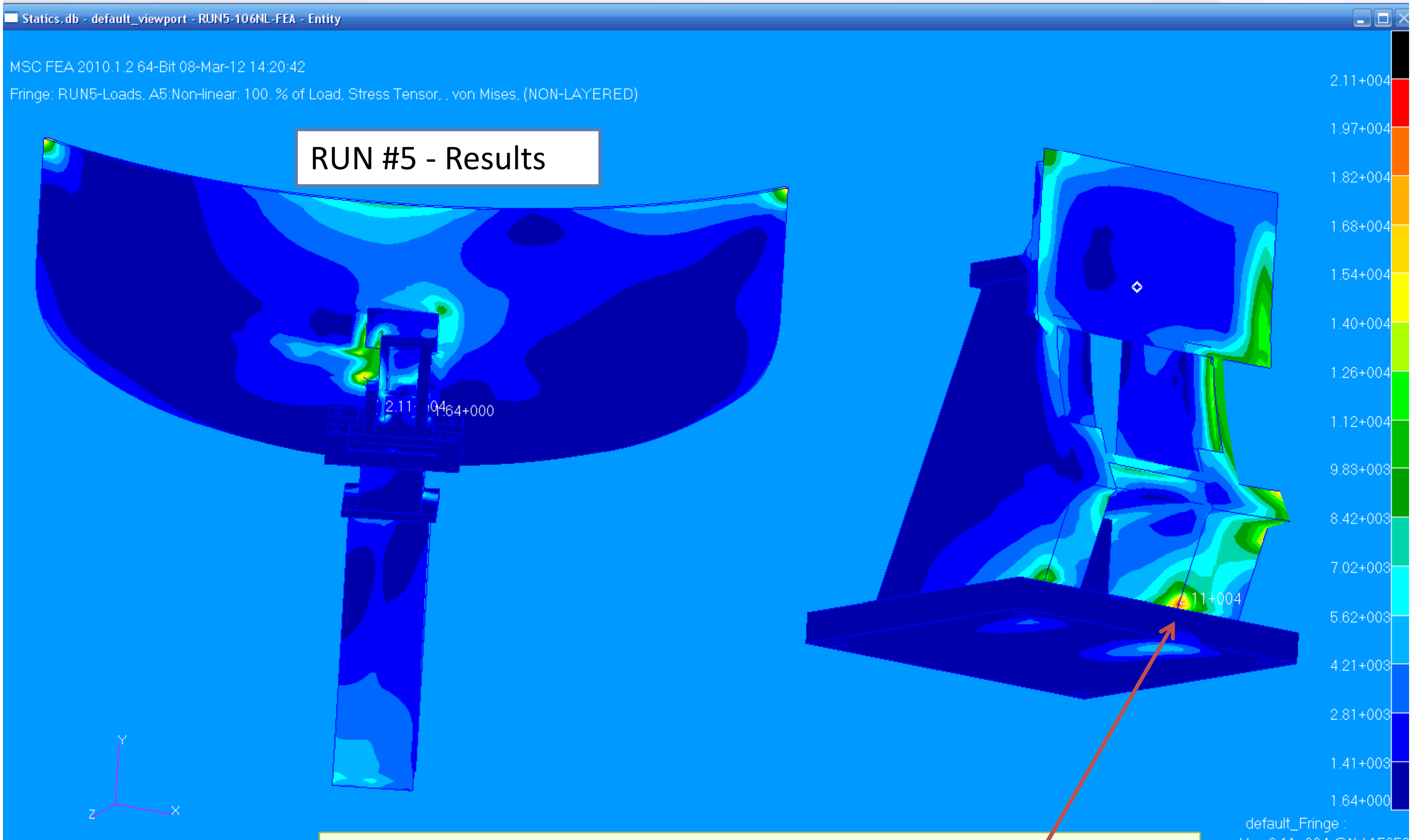
RUN #5 - Configuration



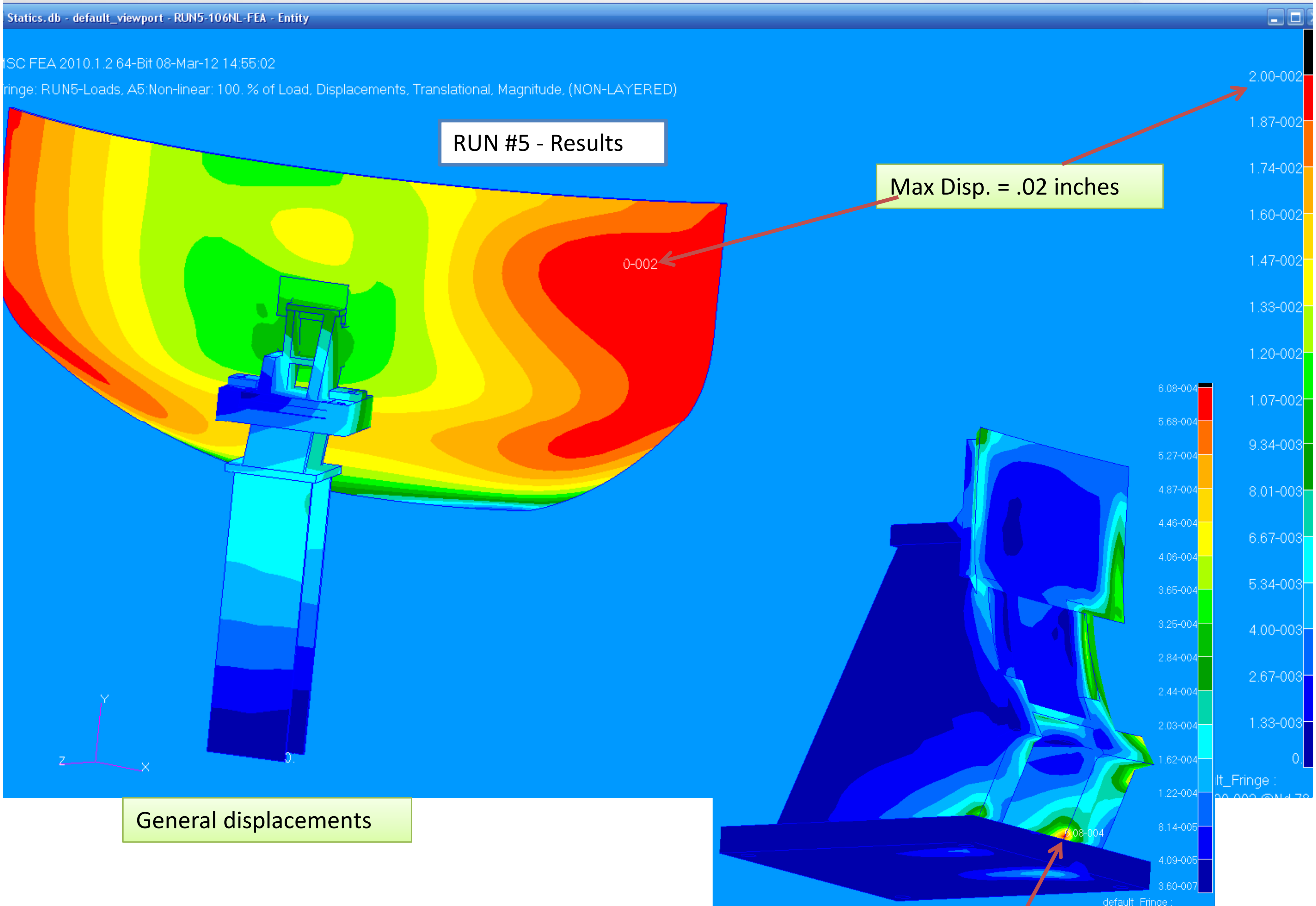
Group – RUN5-106NL-FEA
Input - Run5-106NL-fea.bdf
Loads – RUN5-Loads
Output- Run5-106NL-fea.xdb



RUN #5 is a Sol 106 (Non-Linear) run with B.C. and Load from Run #4 and additional rigid body simulation for the Clevis pad which resists the 37,000. lbs shear load parallel to the VV surface. (Yellow arrow)

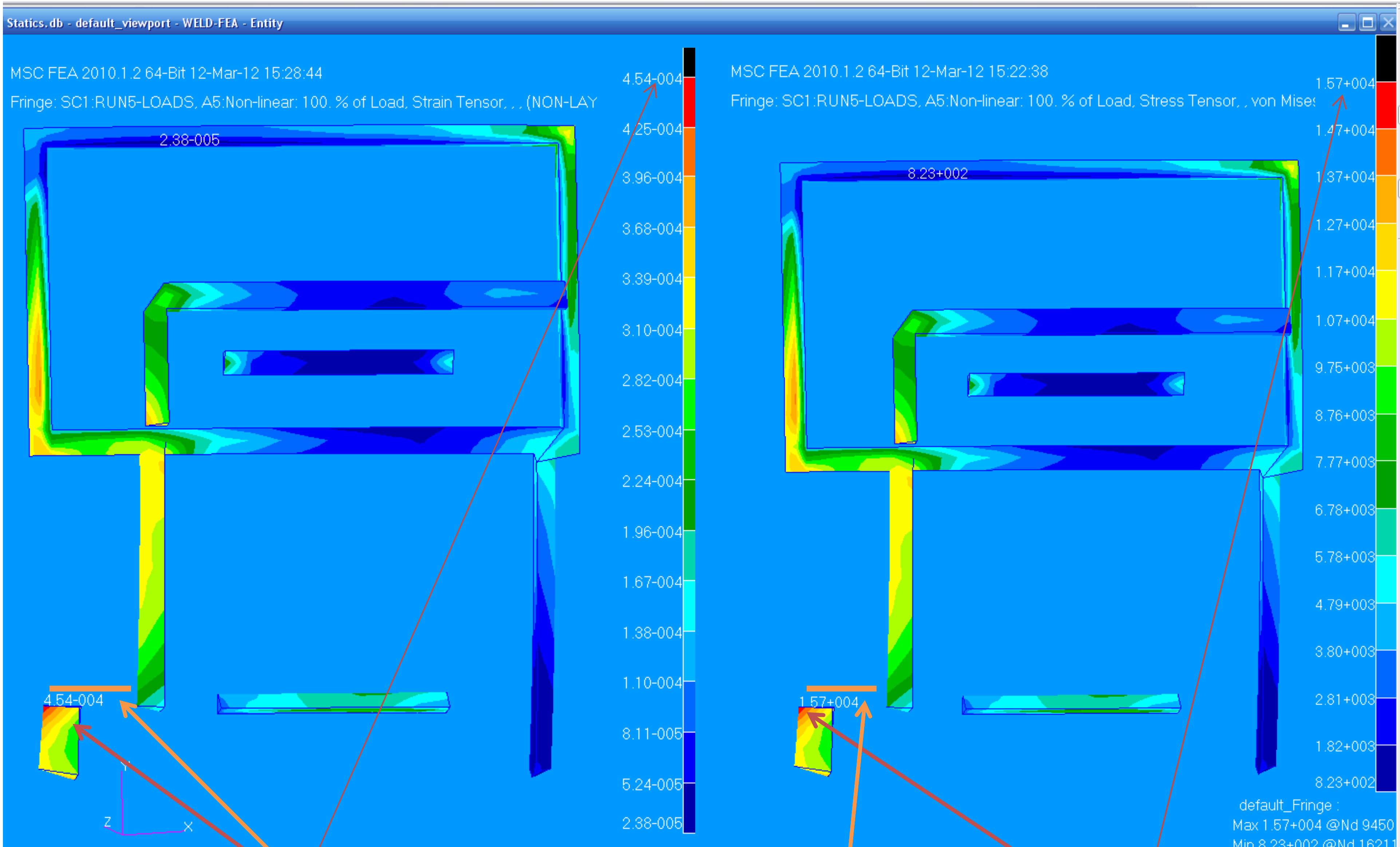


Maximum Von Mises Stresses around the VV-Chair connections = 21,100. psi at the Chair support. Welds seem to be O.K.



General displacements

Maximum strain = .000608



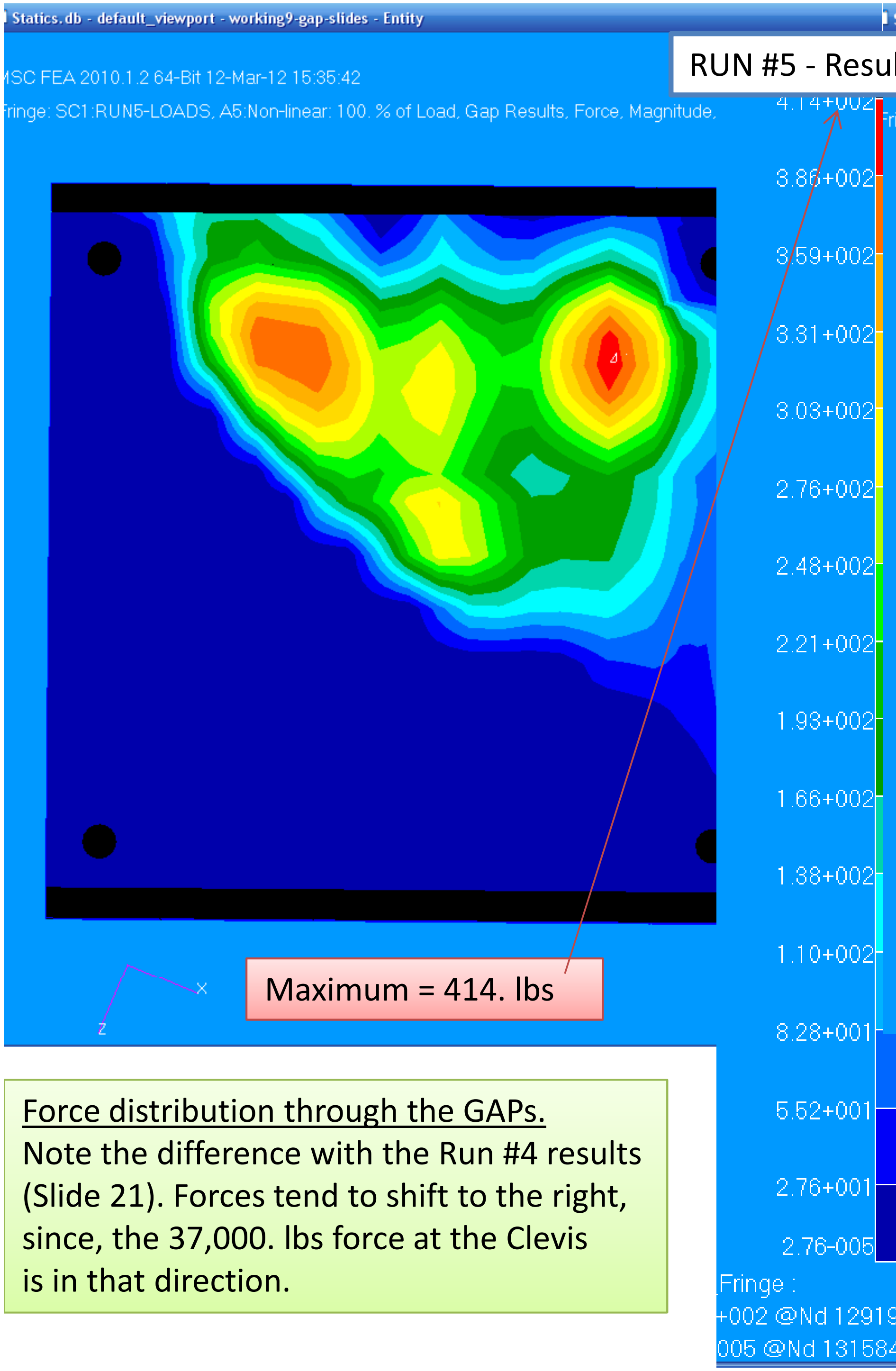
Maximum Strain = .00045

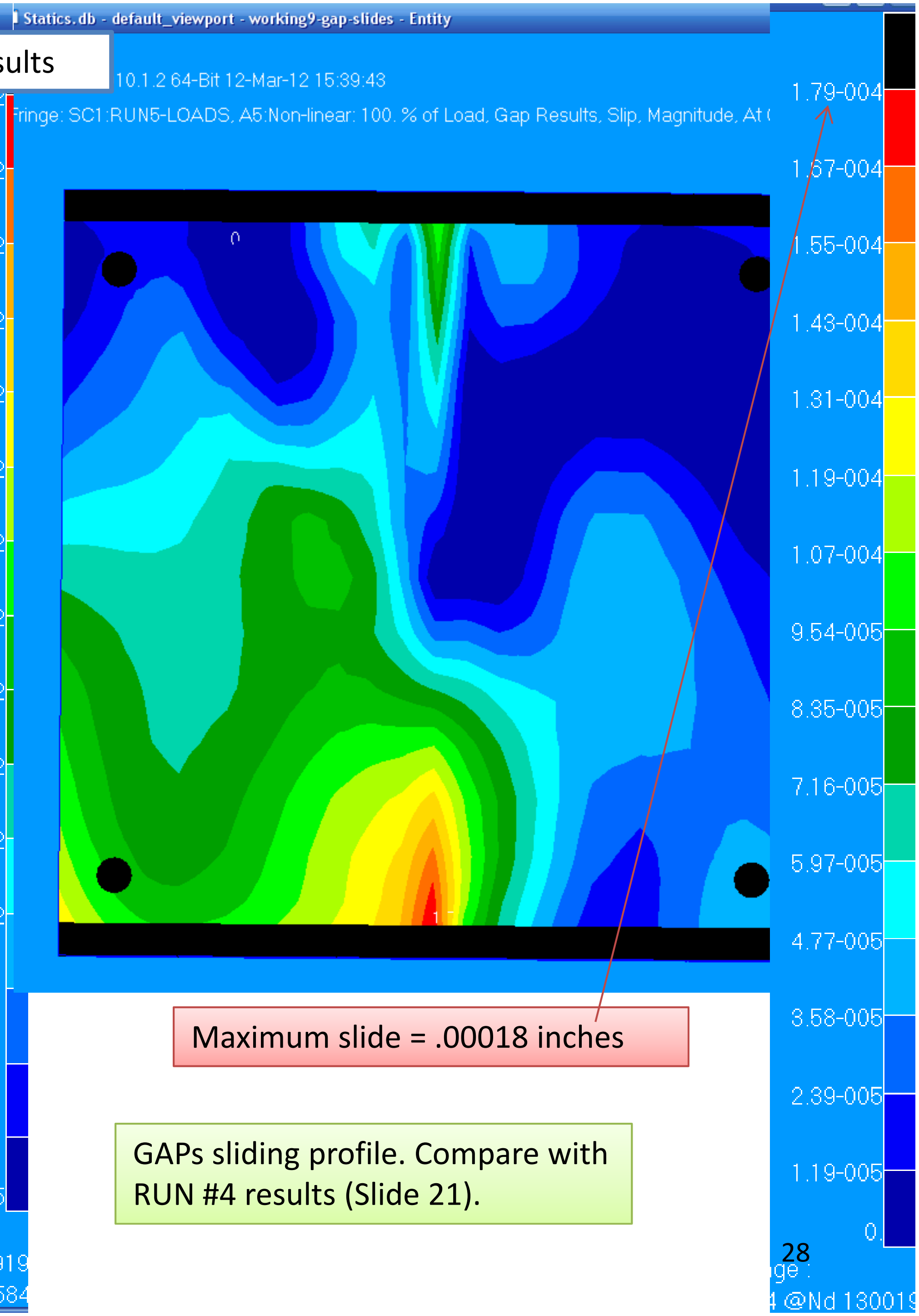
RUN #5 - Results

Maximum Von Mises Stress = 15,700. psi

Note: If this location is possible to weld, it should decrease the maximum stress and/or strain levels. Also see the welding slide below.

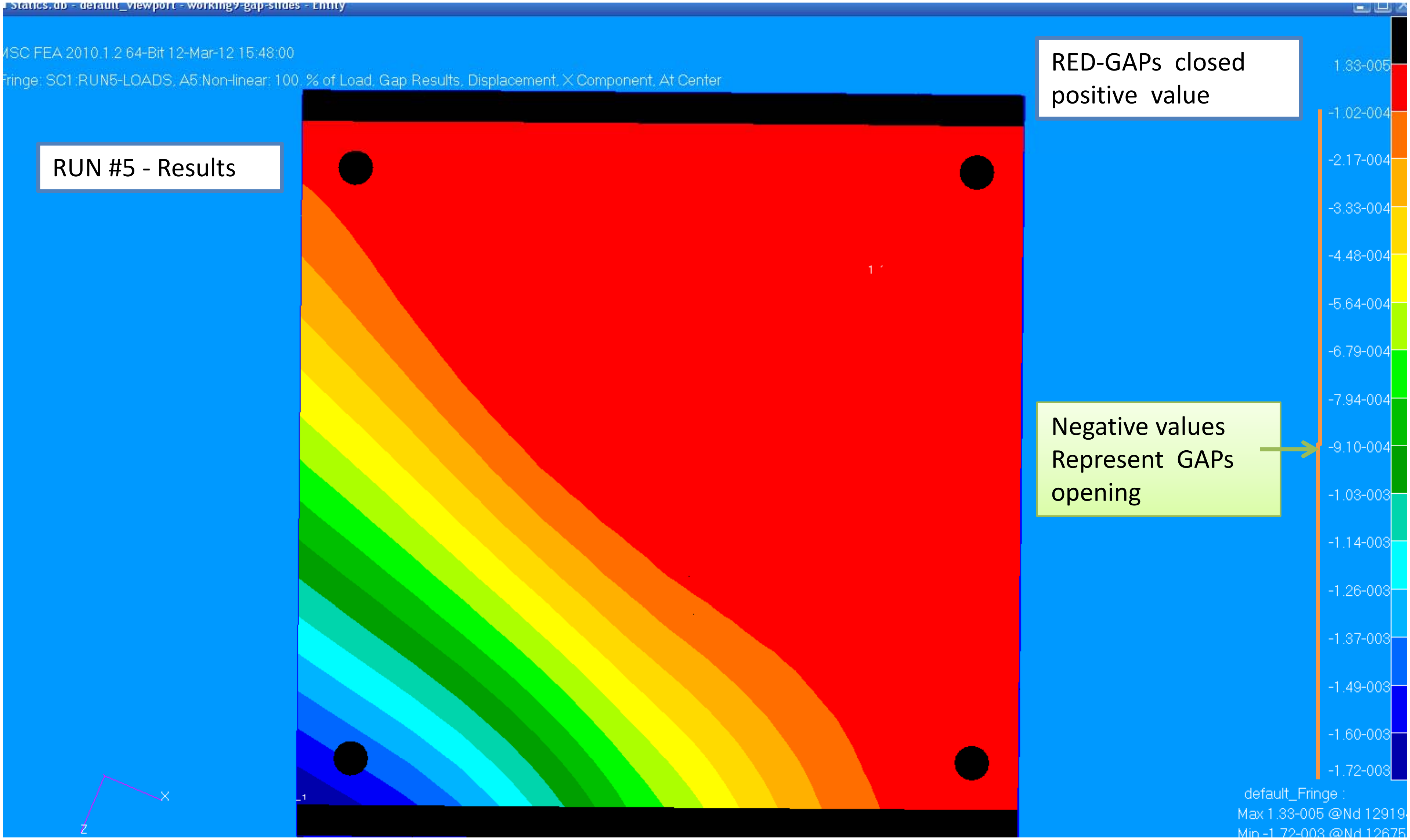
RUN #5 - Results



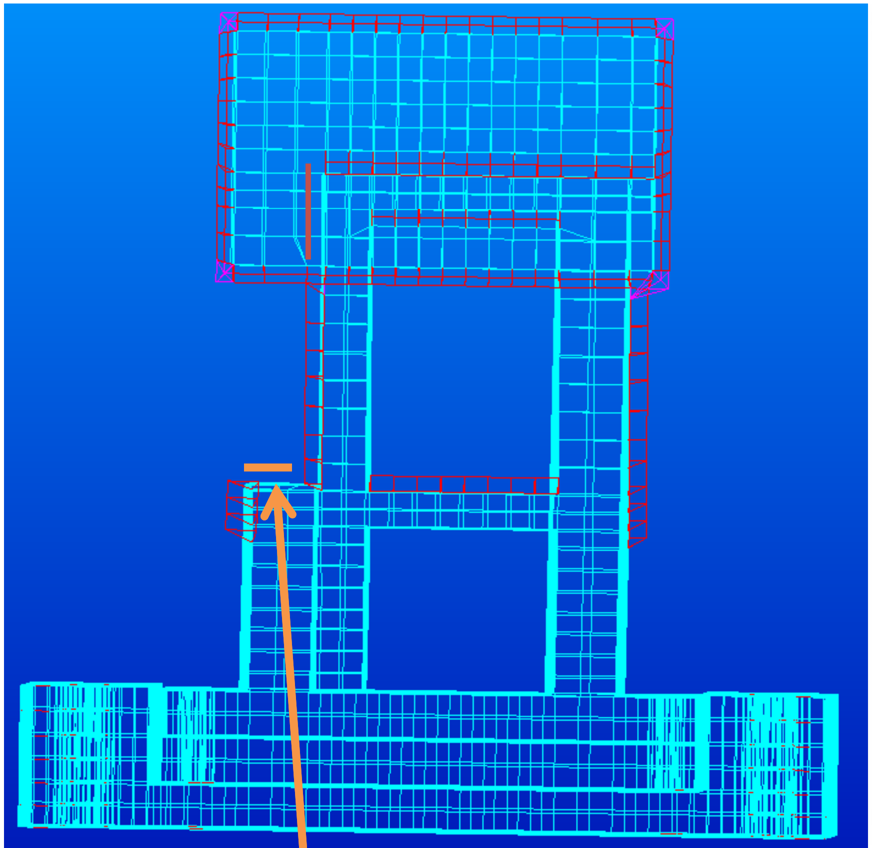


<u>Force distribution through the GAPs.</u>
Note the difference with the Run #4 results (Slide 21). Forces tend to shift to the right, since, the 37,000. lbs force at the Clevis is in that direction.

GAPs sliding profile. Compare with RUN #4 results (Slide 21).



Coordinate 2 controls (defines) the used GAPs. The X-direction (normal to page) is the principal gap direction which defines if the gap is open or closed.
Please see the ".bdf" input file for details.
(Compare with Run #4 results shown on Slide 22.)



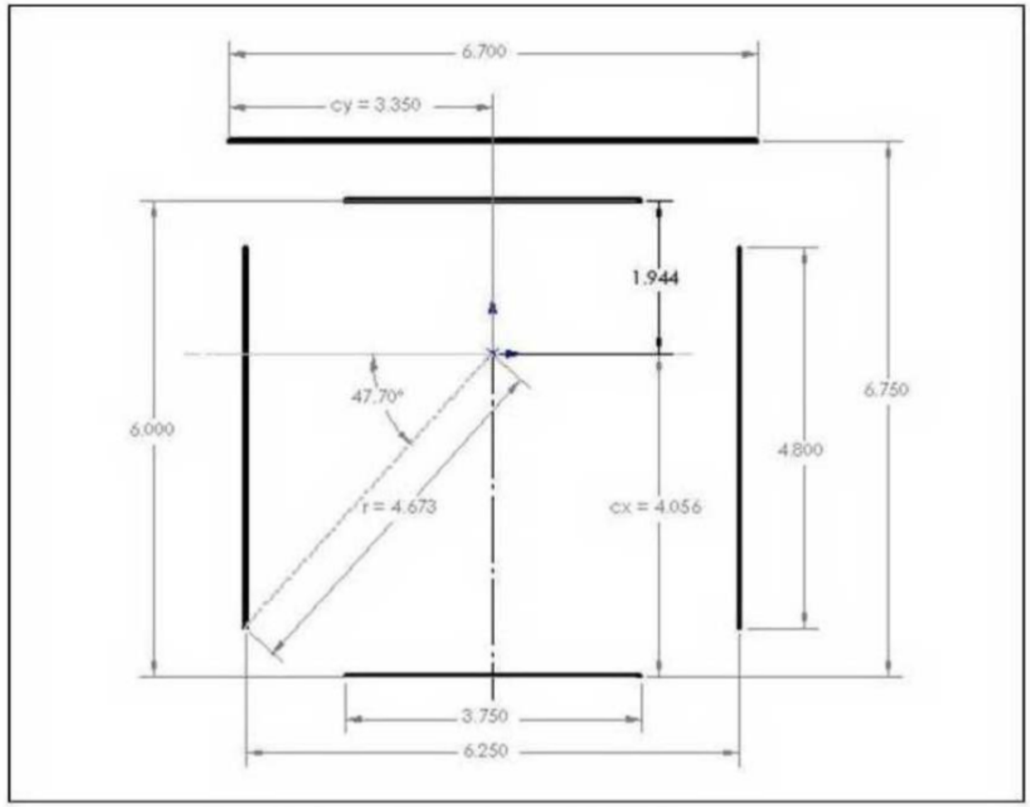


Figure 2 - Support Leg-to-Vessel Weld Dimensions

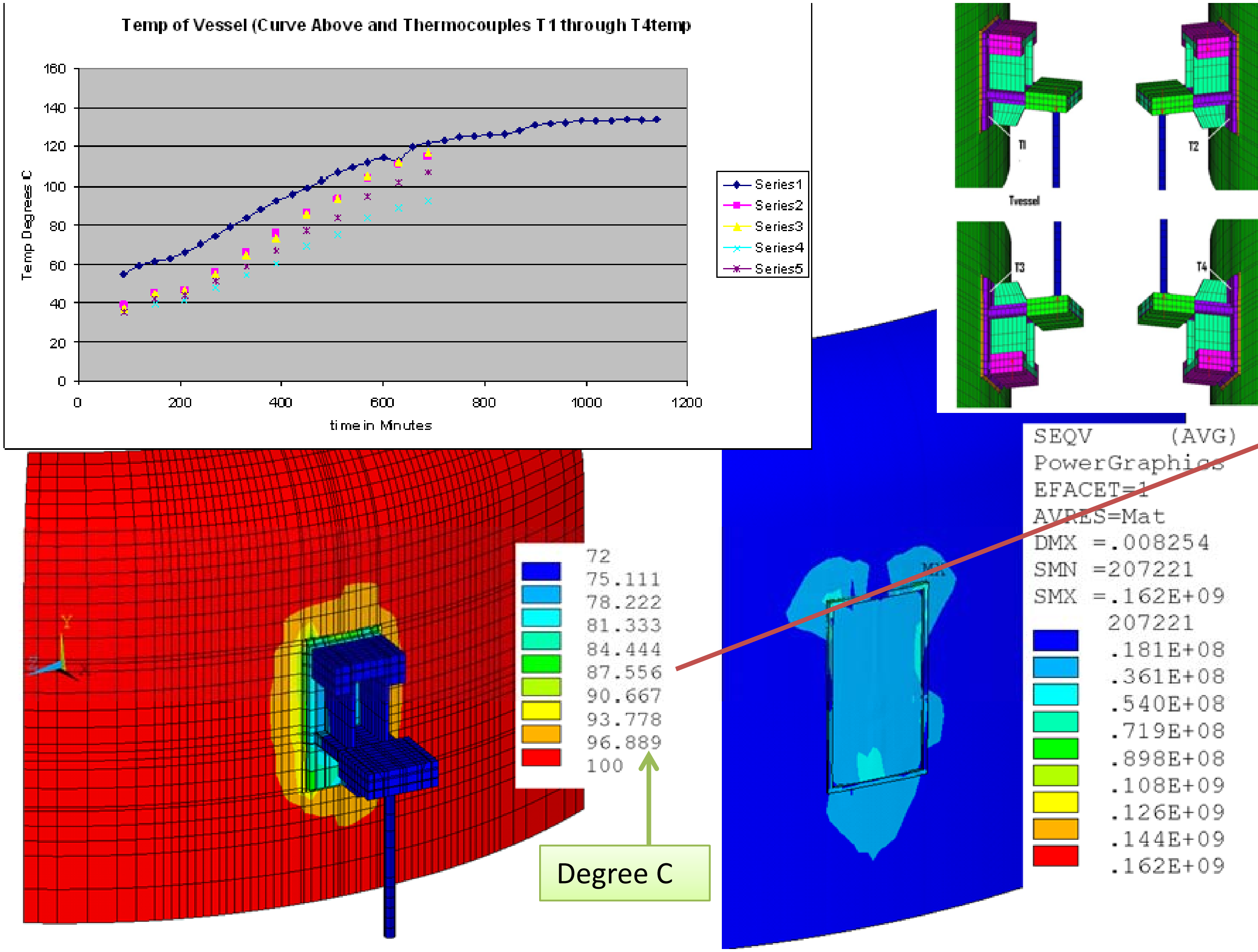
Area that would be <u>extremely desirable</u> to weld, but, may not be readily accessible

Original weld configuration – standard chair
Note: received from M. Smith (E-mail)

Red elements represent the actual welds

Welding scenario for the redesigned VV-Chair support

RUN #6 Bakeout



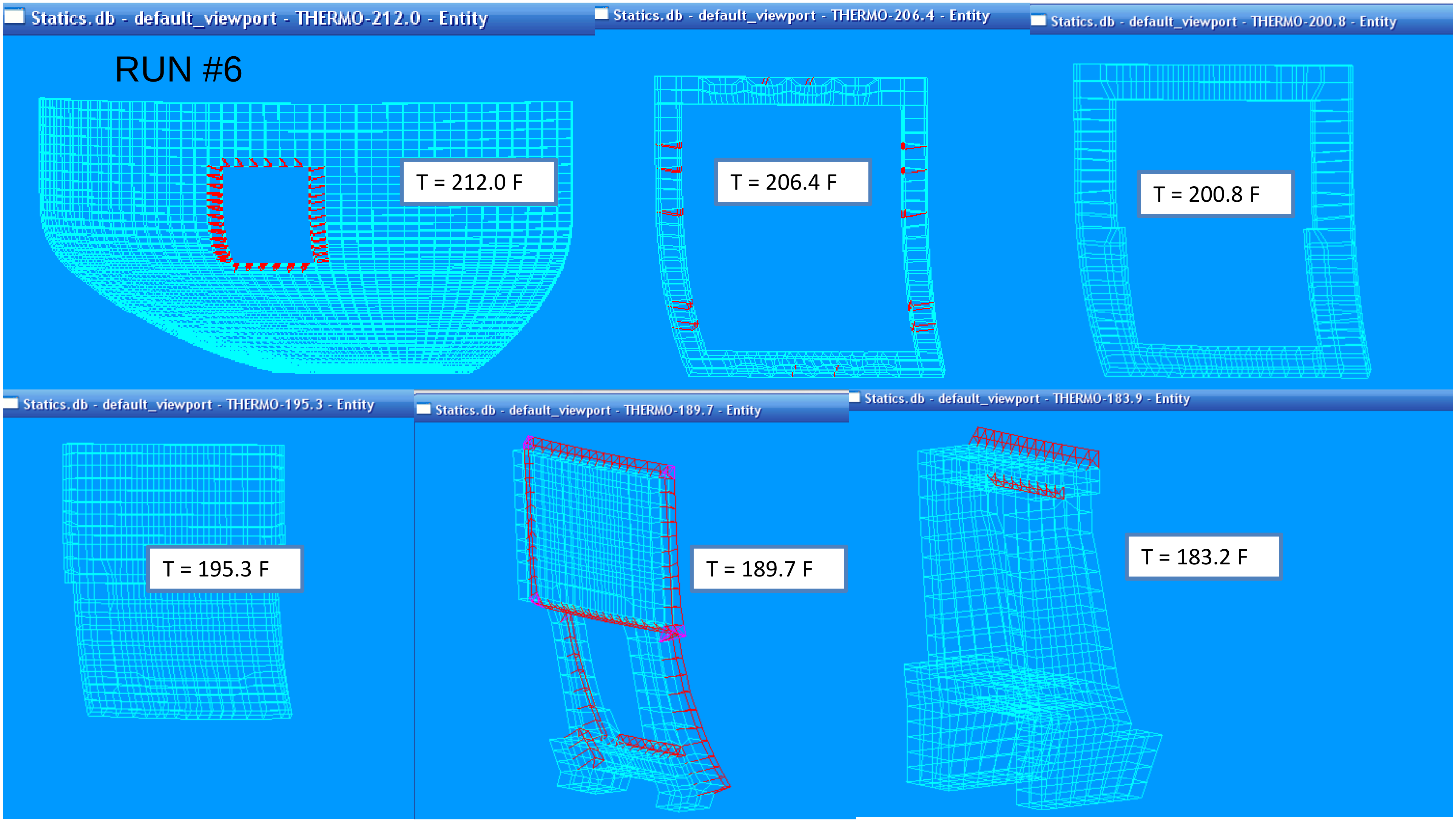
Degree C

Conversion to degrees F

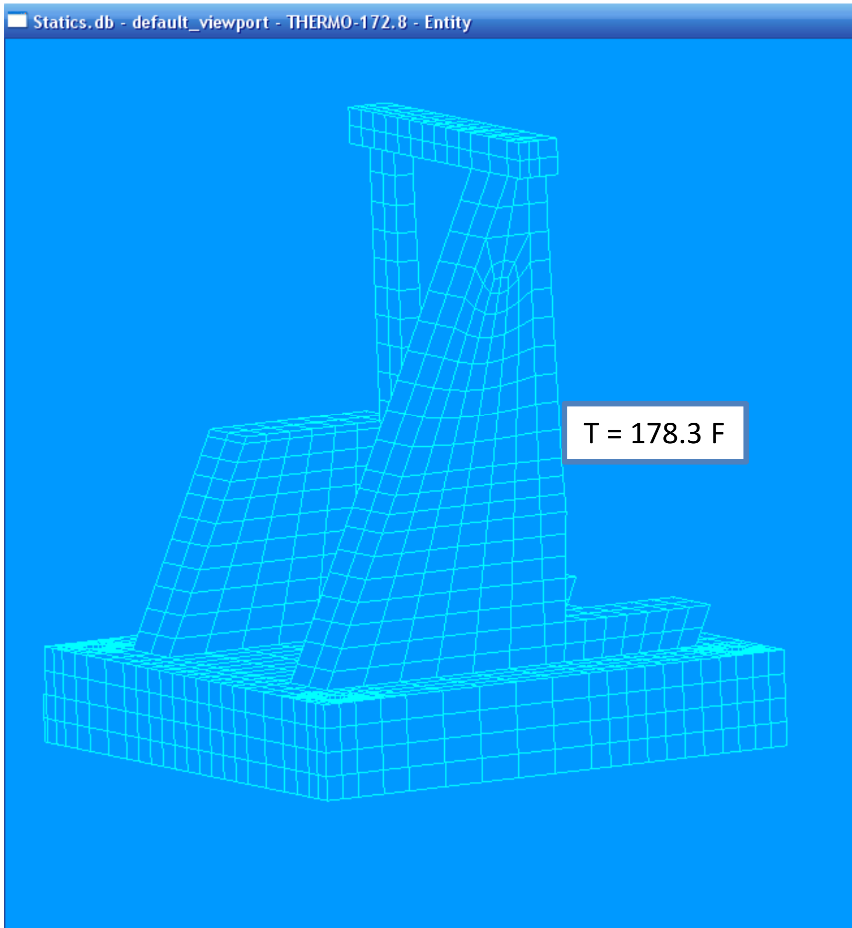
C	F
75.1	167.20
78.2	172.76
81.3	178.34
84.4	183.92
87.6	189.68
90.7	195.26
93.8	200.84
96.9	206.42
100.	212.0

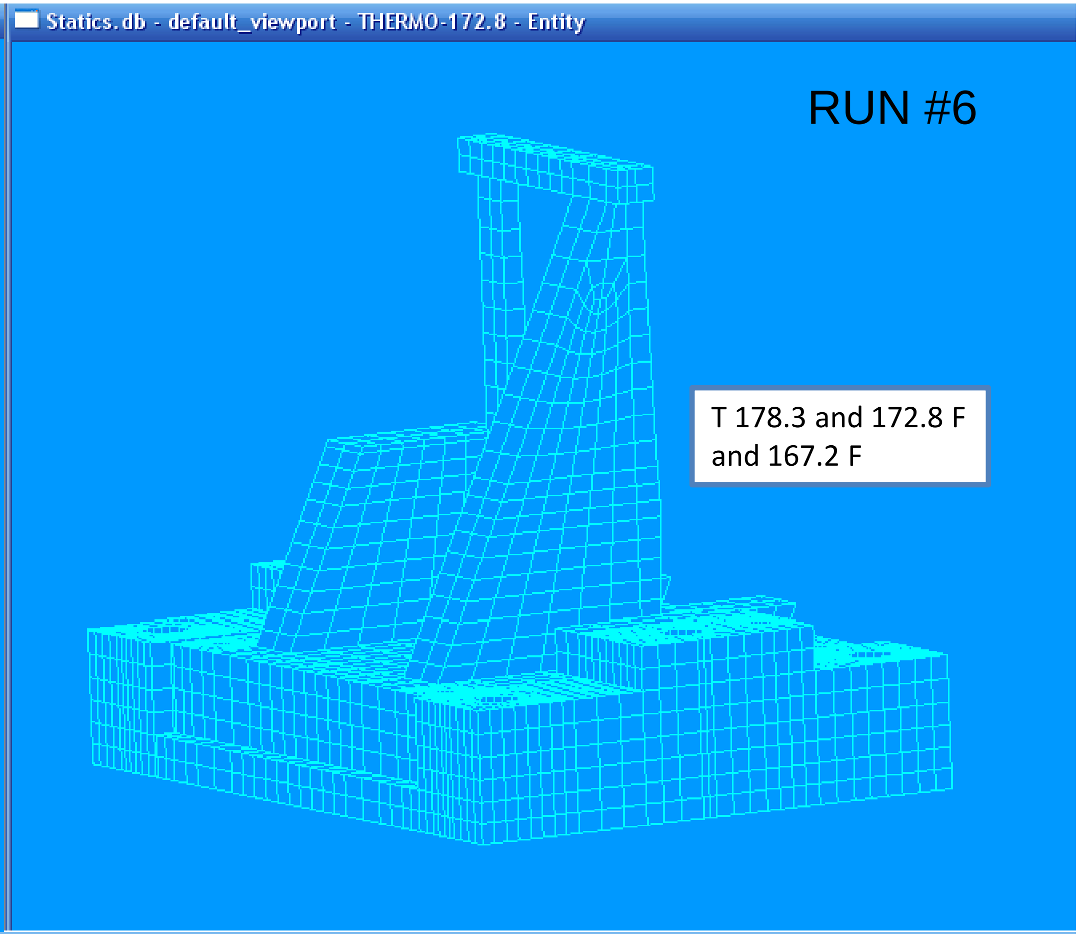
Use this for RUN #6

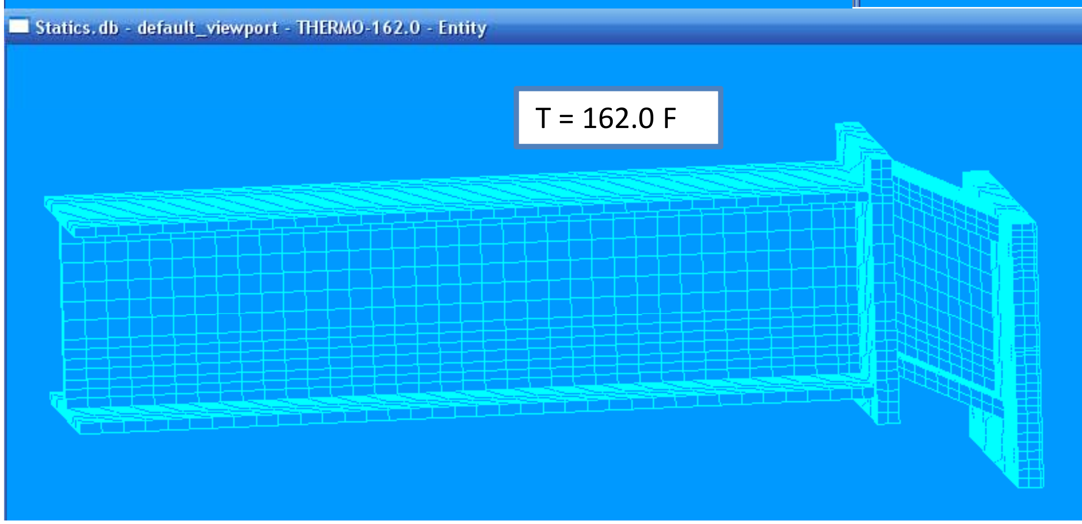
This data is taken from the previous PF 4 and 5 support hardware simulation. Similar temperature scenario exists in the area of contact between the VV and the main support Chair structure.
This temperature data has to be converted to degrees F in order to be used in the present RUN #6 simulation.



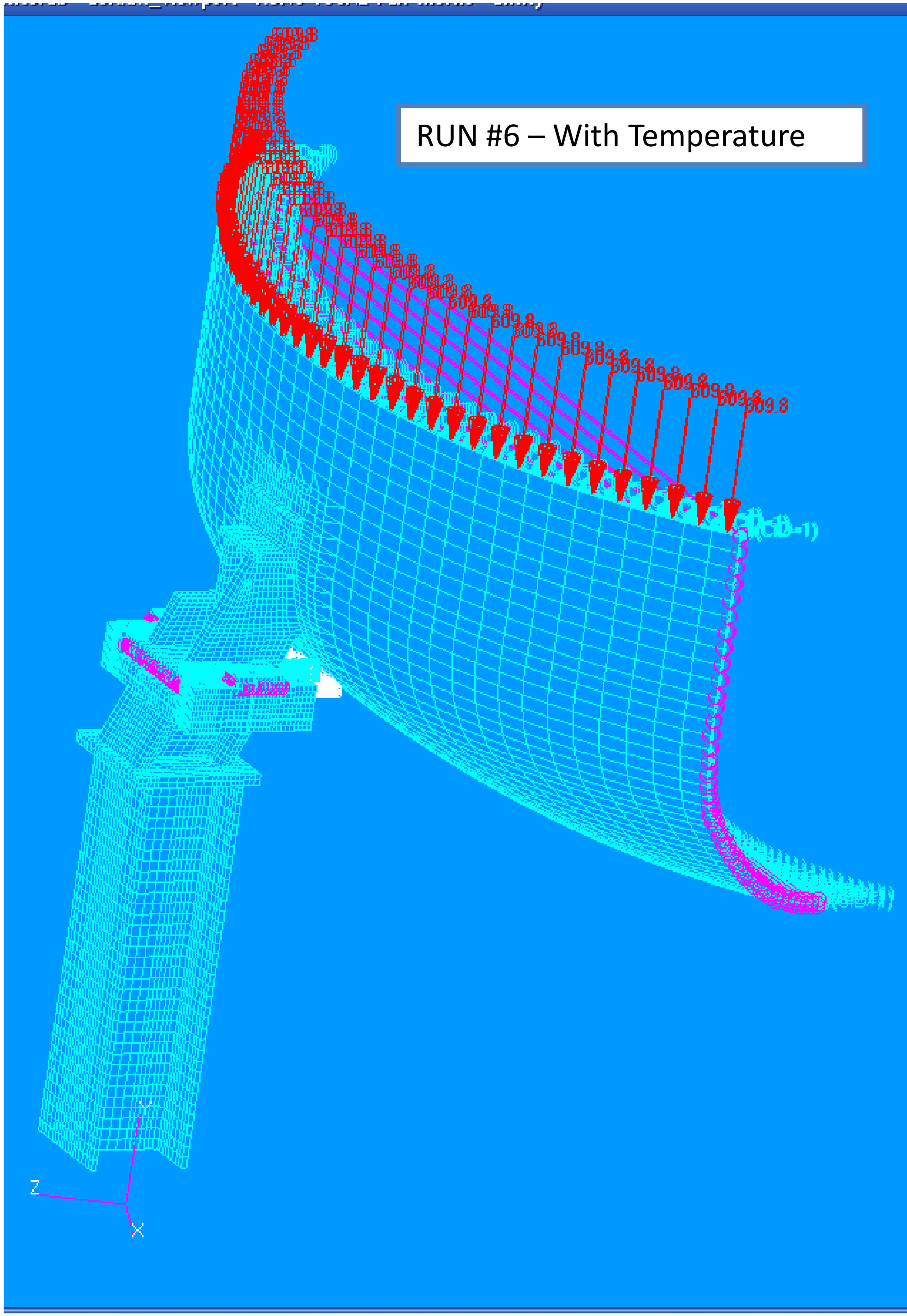
Applied temperature distribution according to the converted table from the previous slide. These groups are most critical in order to determine the VV bake out effects. This is a highly approximated temperature profile. The VV support structure is a very effective heat sink. Because of its total mass, the gradients are probably reasonable. If better simulation is necessary, then, a complete thermal analysis would be required to determine temperature at each nodal point (time consuming procedure).







Please Note: The described temperature distribution is the result of the author's assumed interpolation.




```
$ Nonlinear Static Analysis, Database
SOL 106
CEND
TITLE = MSC.NASTRAN JOB CREATED ON 17-FEB-12 AT 08:48:38
ECHO = NONE
MPC = 20134
TEMPERATURE(INITIAL) = 10
SUBCASE 1
   TITLE=106 with THERMO
   SUBTITLE=RUN6-Loads
   NLPARM = 1
   SPC = 2
   LOAD = 12
   TEMPERATURE(LOAD) = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM    POST    0
PARAM    AUTOSPC YES
PARAM   PRTMAXIM YES
NLPARM   1       10              AUTO    5       25              NO
$ Elements and Element Properties for region : HexWedTet
```

From the “RUN6.bdf” file

Group – RUN6-106HL-FEA-thermo
Input -- RUN6-NL-THLoads.bdf
Loads -- RUN6-Loads
Output – RUN6-NL-THLoads.xdb

RUN #6 Loads formulation is as follows:

Mechanical loads and B.C. are identical to those applied in RUN #4, plus the described temperature distribution scenarios on the previous slides.

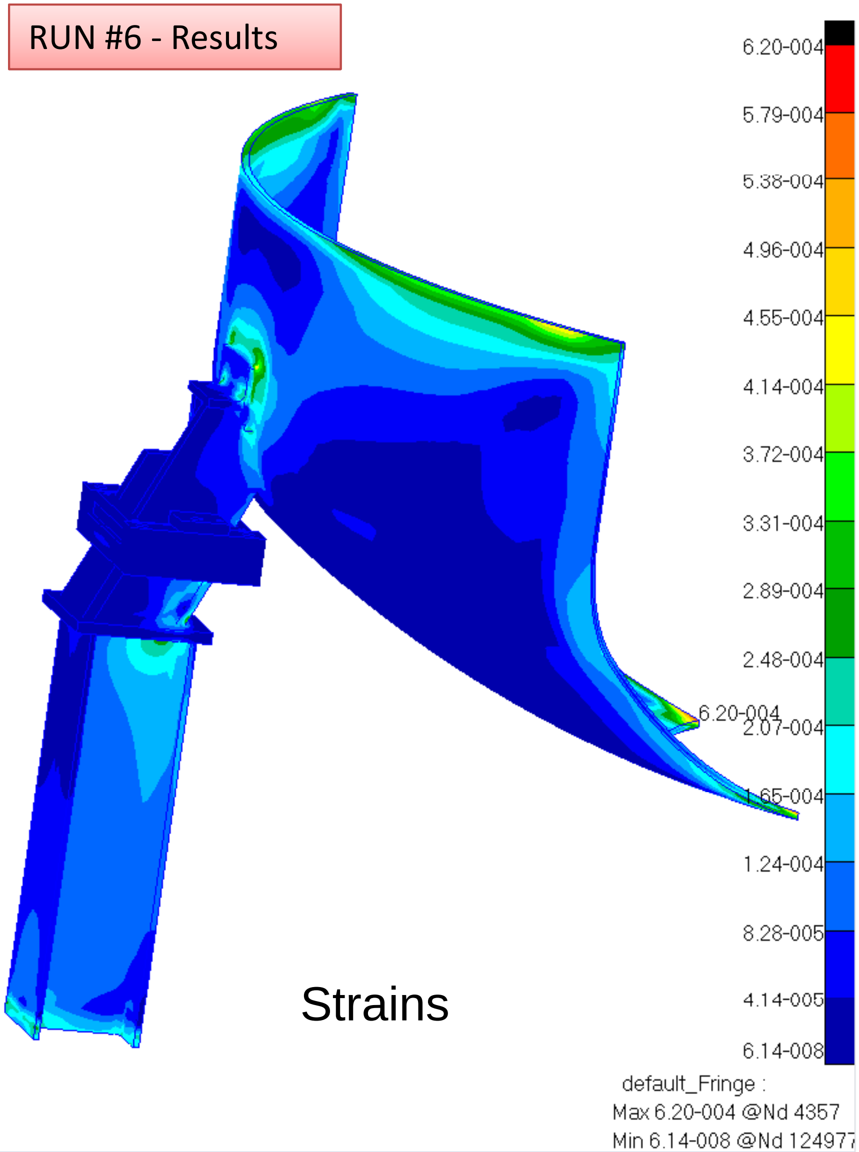
MSC FEA 2010.1.2 64-Bit 16-Mar-12 14:47:52

Fringe: RUN6-Loads, A7:Non-linear: 100. % of Load, Strain Tensor, , .

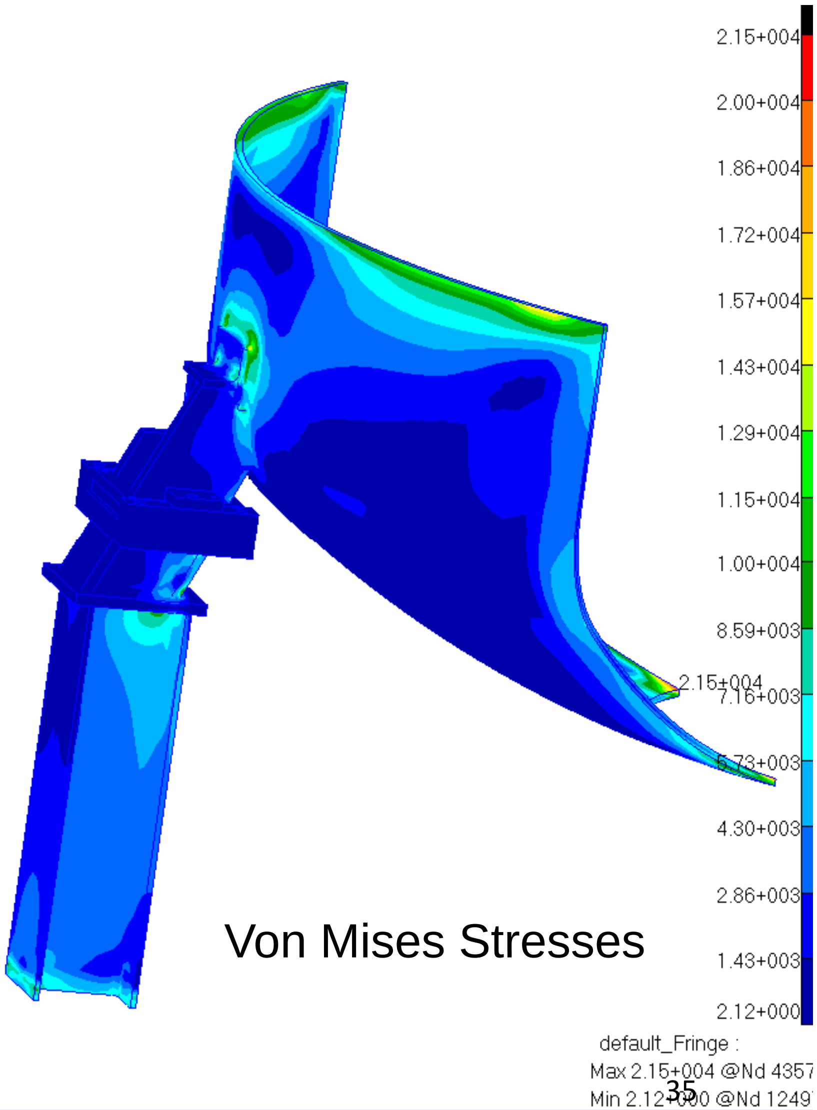
MSC FEA 2010.1.2 64-Bit 16-Mar-12 14:44:43

Fringe: RUN6-Loads, A7:Non-linear: 100. % of Load, Stress Tensor, , von Mises, (NON-LAYERED)

RUN #6 - Results

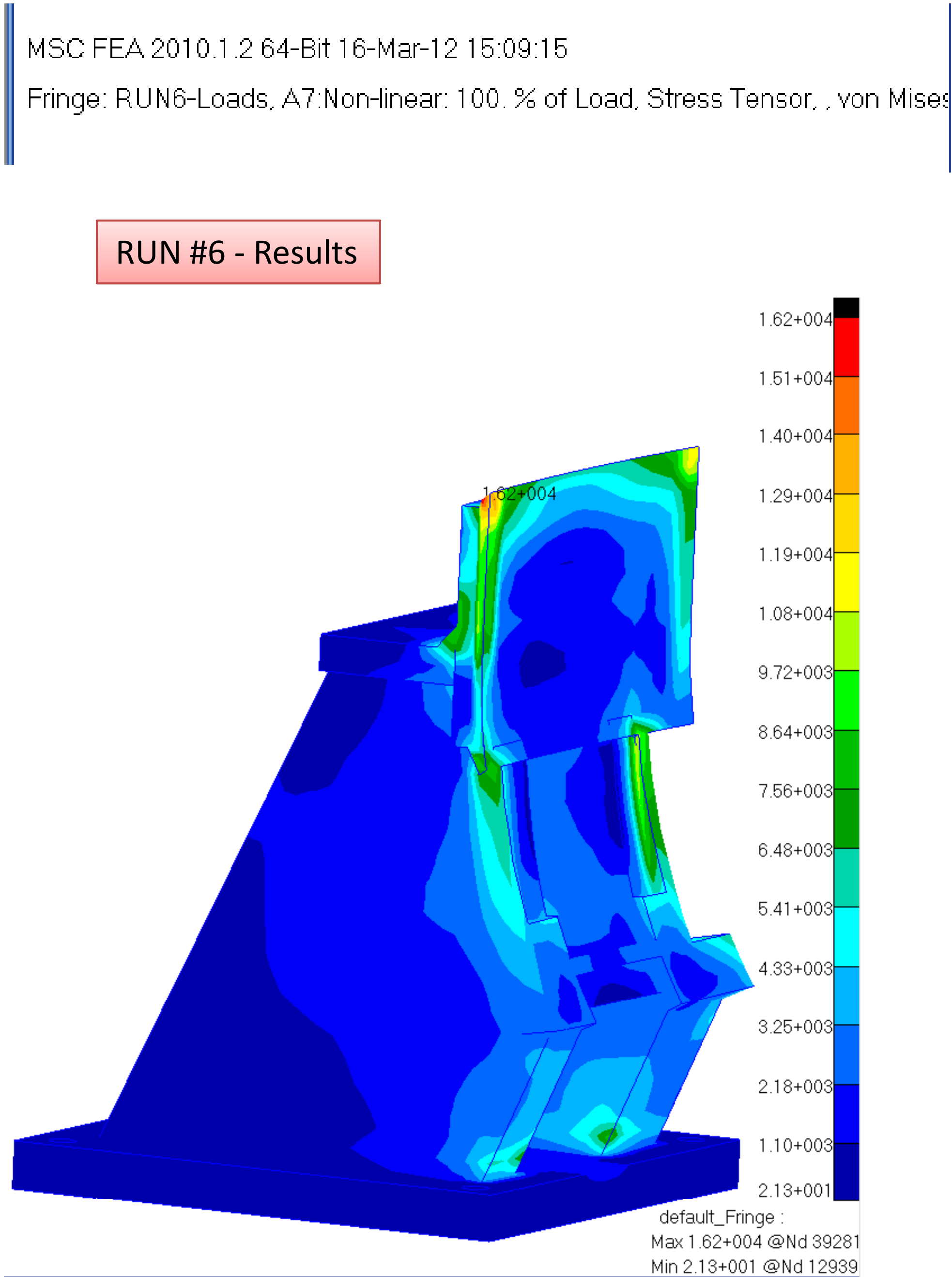


Strains

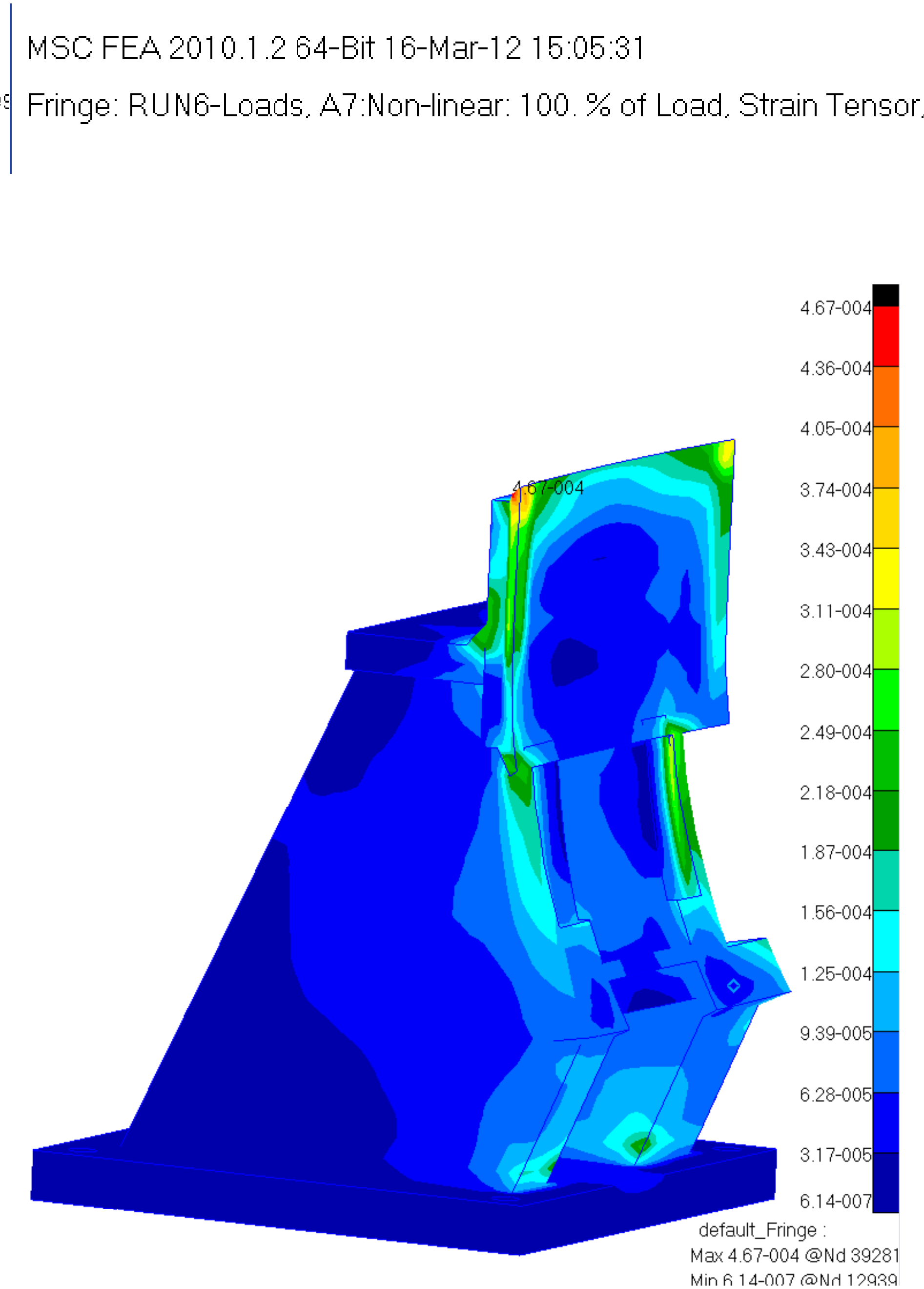


Von Mises Stresses

RUN #6 - Results

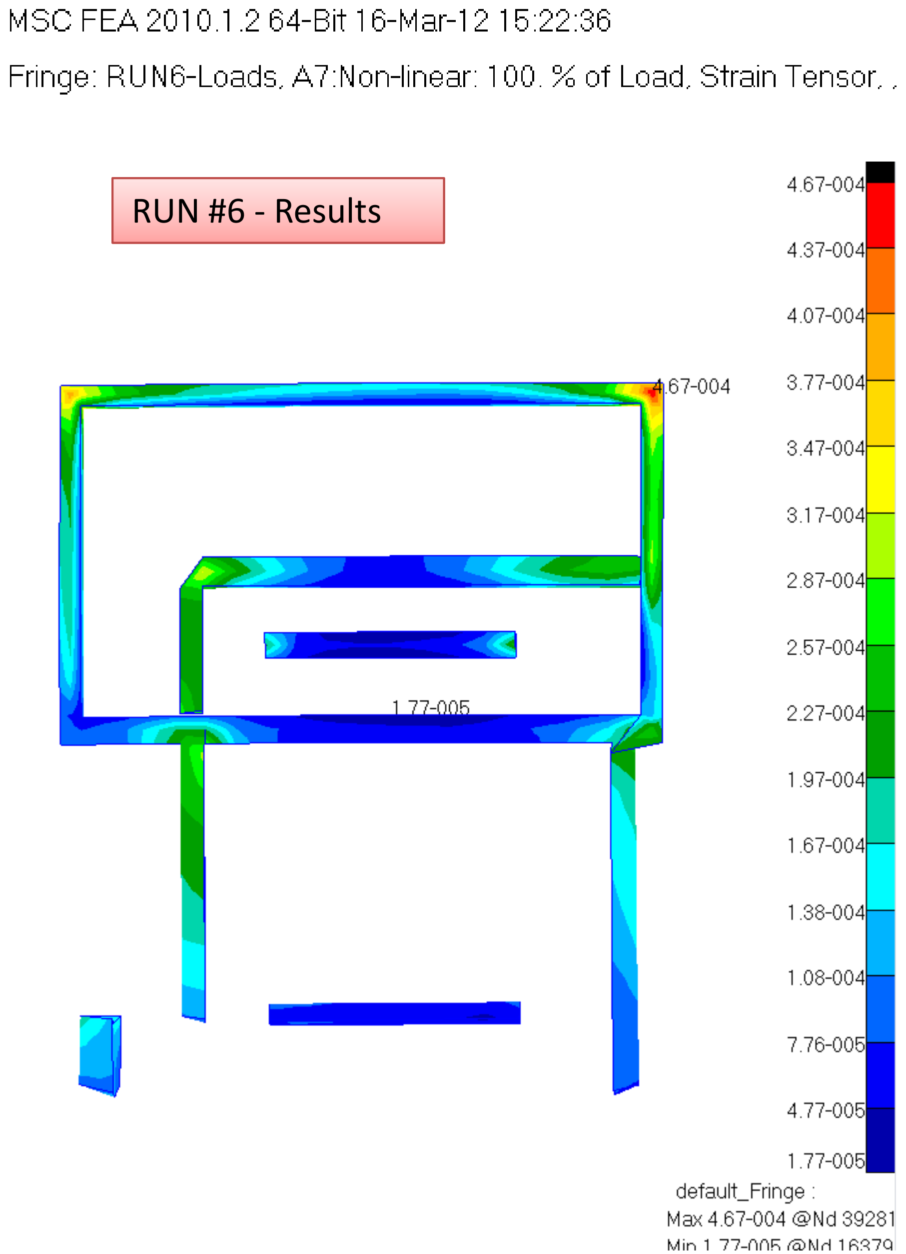


Von Mises Stresses

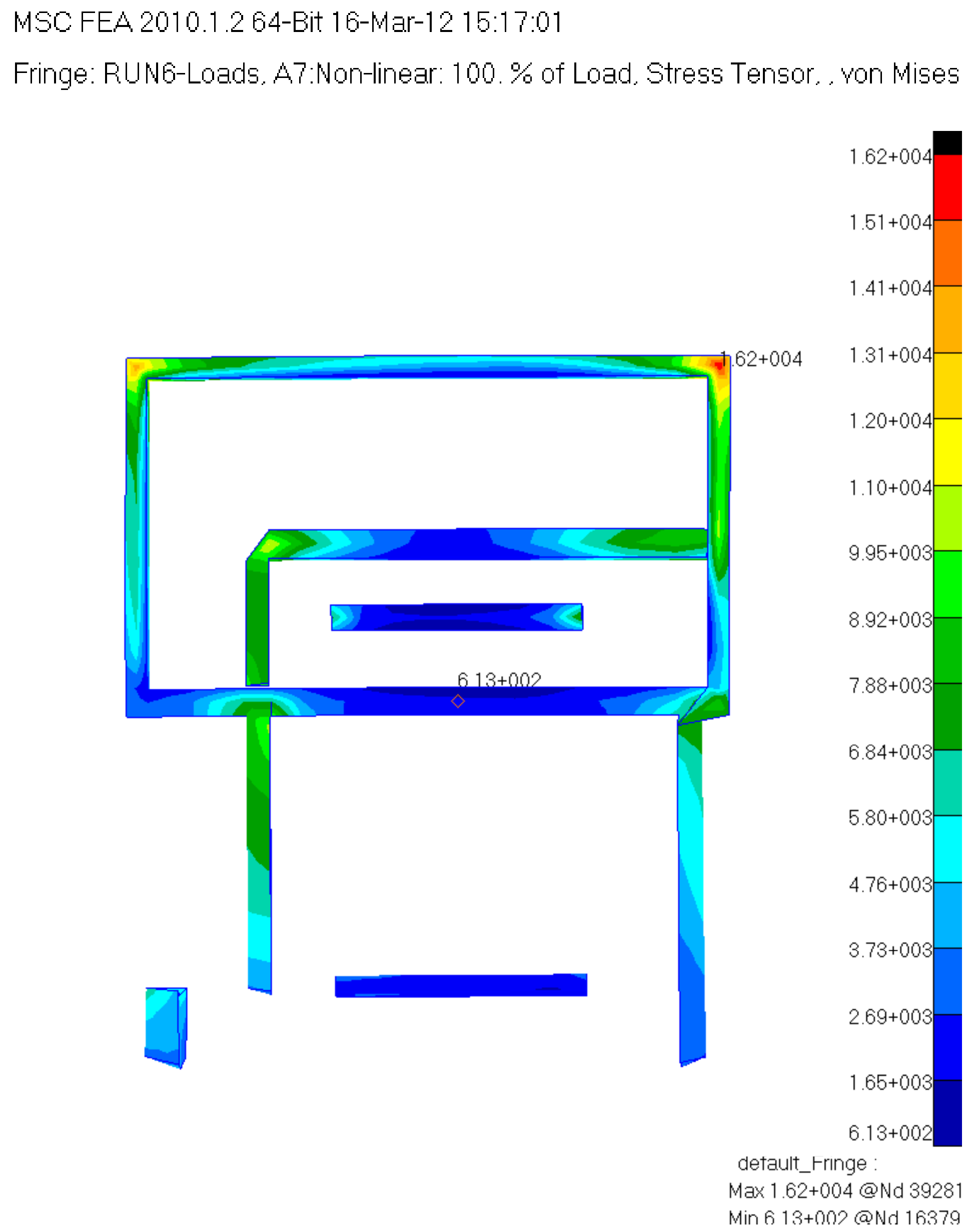


Strains

RUN #6 - Results

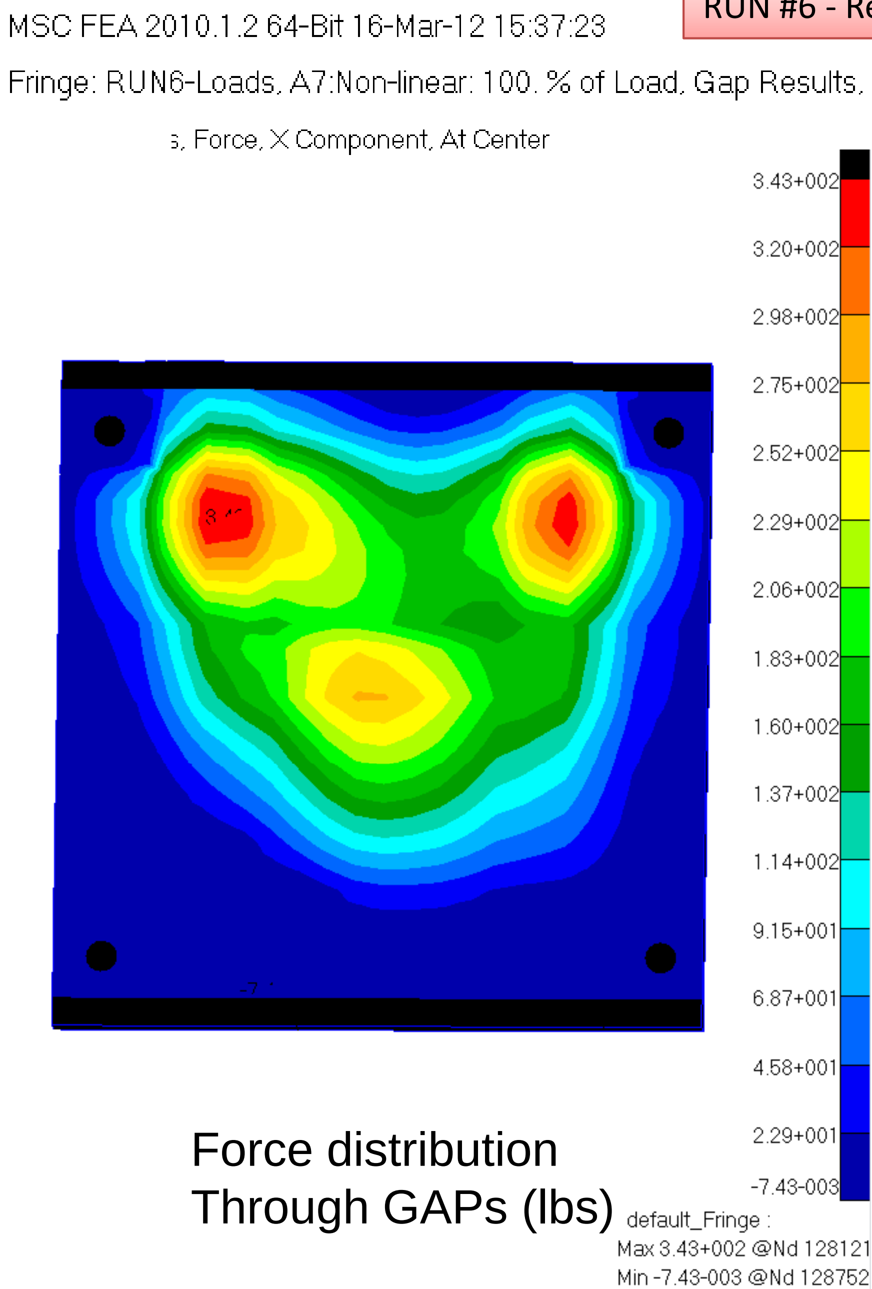


Strains

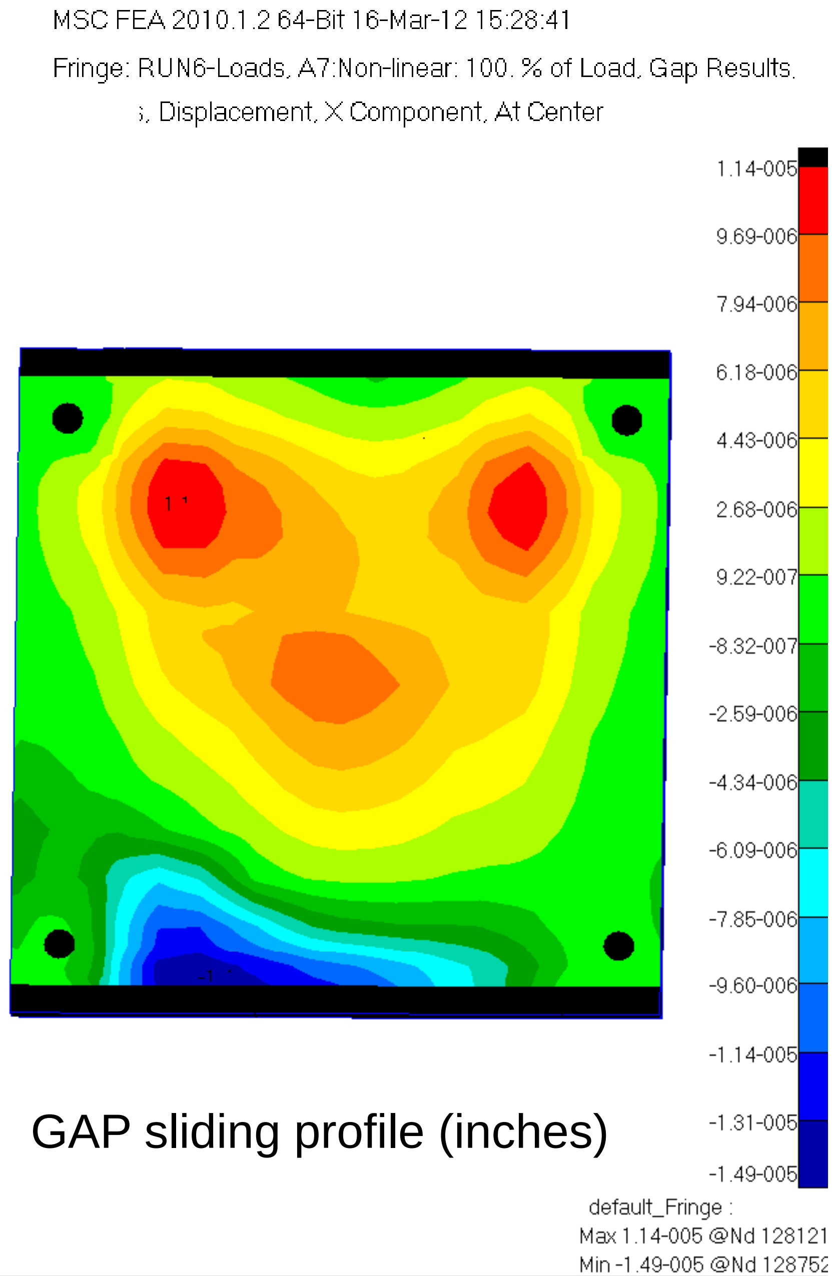


Von Mises Stresses

RUN #6 - Results



Force distribution Through GAPs (lbs)



GAP sliding profile (inches)

Complete Summary explanation for all Runs

From the examinations of the six different FEA simulations presented by the above slides, the following evaluation can be stated:

- Basic FEA geometry simulations (nodal points, elements), for all runs, are the identical.
- The differences are in the applications of the model support/boundary conditions (SPCs) and their locations for each specific run. This was implemented in order to fully investigate the worst possible magnitude and locations of stresses and strains in the critical areas (especially the welded locations between the VV and the support structure).
- Runs #1, #2 and #3 are fully linear (the sliding effects were not considered). GAPs were replaced with RBE2 rigid elements. Runs #4, #5, #6 are non linear (NASTRAN Solution 106) where GAPs were fully utilized to evaluate the contributions due to the VV-Chair sliding effects. For these runs, GAP element contours are included to evaluate the correctness of the load transmission at the sliding surface (see the appropriate slides).
- Runs #1 and #2 were included here only for model verification and to demonstrate the effects of some extreme boundary support conditions which are used to improve the total model simulations.

In general, the following can be concluded and recommended for minimizing stresses and strains: For Run #1 under the most severe boundary conditions, the maximum stresses in the weld locations are equal to 15,300. psi. For Run #2 (less severe boundary condition) maximum weld stress decrease to 10,200. psi. From these two runs, the maximum possible stresses are established for total VV dead weight effects. Inclusion of the additional static loads may change the locations as well as the stress/strain values.

Complete Summary explanation for all Runs (cont.)

	Welds Maximum		Chair Body Maximum	
Run #	VM Stress	Strain	VM Stress	Strain
#1	15,300. psi	.0004	56,700.*	.0002
#2	10,200.	.0005	12,500.	.0005
#3	10,500.	.0003	15,500.	.000444
#4	11,100.	.00032	15,200.	.000438
#5	15,700.	.00045	21,100.	.00061
#6	16,200.	.00047	21,500.	.00062

* This stress is high because of the conservative boundary conditions used and should be disregarded. (see RUN #1 results slides)

Final conclusion - Recommendations

Stainless steel with a yield stress of 42,000 psi was used for these analyses. Comparing to the above table, all the stress and strains are well within allowable limits (with the exception of Run #1 as noted above) and provides an adequate static margin for welds and the support structure alike.

Strong recommendation is to weld corner locations carefully in order to minimize the stress concentrations there (see Slide 30). It is advised that regular inspections of these locations should be performed.

Appendix A

All the data is saved in the following locations and it includes: the write up, analyses data base which includes all the analyses groups, input files for all completed runs (.bdf files), and the output results (.xdb files).

Folder name is "Vessel-Support-Chair-Data"

P:\public\Snap-srv\progoff

G:\Nastran P.R. - (external drive on ASALEHZ-64PC)

NSTX-U PROJECT

Vessel-Support-Chair As-Built evaluation and comparison to the original prediction as reported in the NSTXU-CALC-12-10-00 document

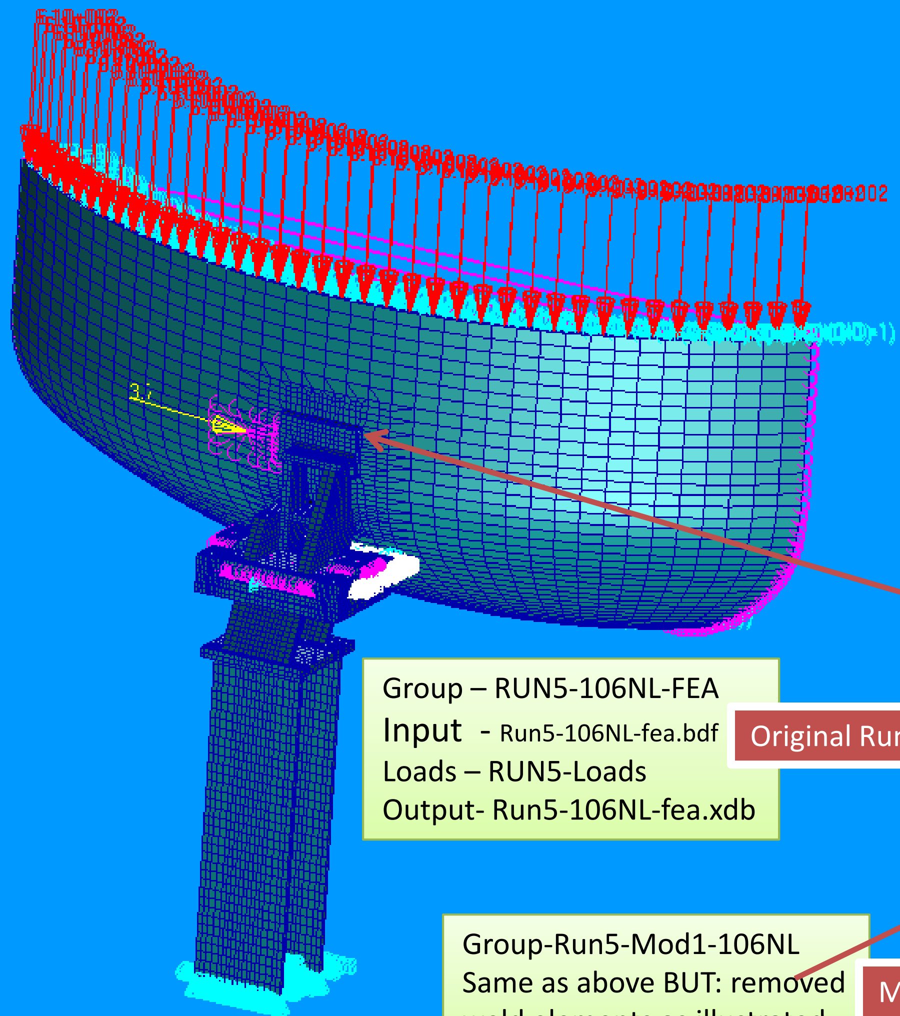
Methods of comparison:

1) Examination of the welds locations for the as-built configuration
2) Weld size increase from 3/8-inch (original) to 5/8-inch (as- built)
3) Removing weld elements from the upper location of the Chair-VV support plate, (see slide #6). Additional Run#5 configuration "Run5-Mod1_106NL" (Data stored in "Vessel-Support-Chair-Data" folder).
4) Examination of the as-built final pictures provided by the NSTX-U team.

Complete simulation justification

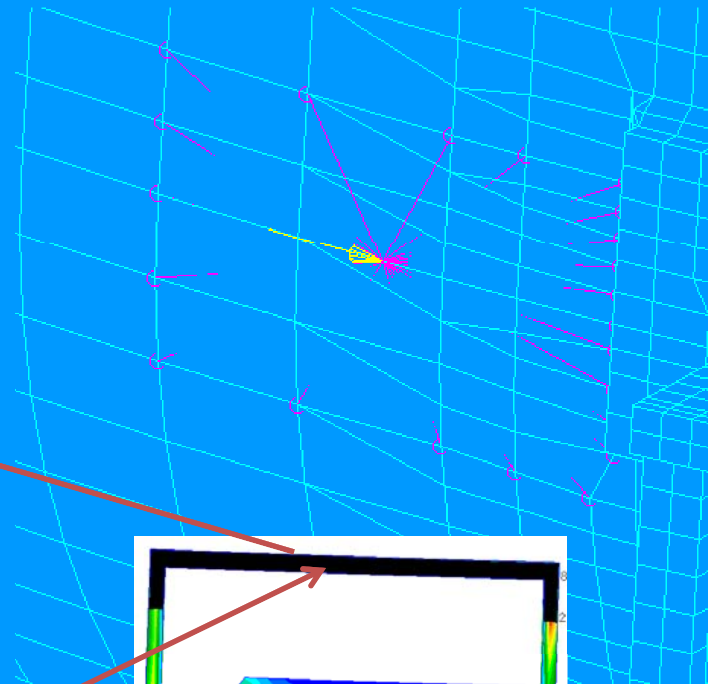
Peter Rogoff February 2013

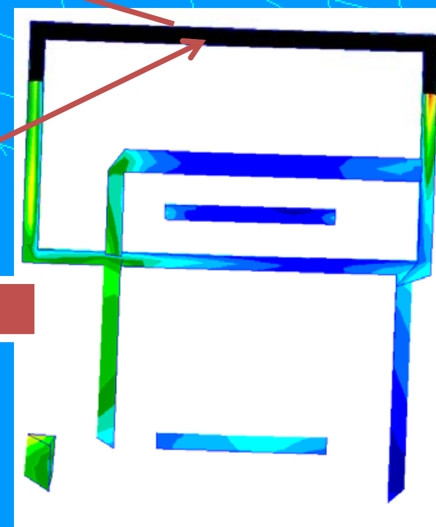
RUN #5 - Configurations



Group – RUN5-106NL-FEA
Input - Run5-106NL-fea.bdf
Loads – RUN5-Loads
Output- Run5-106NL-fea.xdb

Original Run





Group-Run5-Mod1-106NL
Same as above BUT: removed weld elements as illustrated

Modified Feb2013

RUN #5 is a Sol 106 (Non-Linear) run with B.C. and Load from Run #4 and additional "DUMMY" rigid body simulation for the Clevis pad which resists the 37,000. lbs shear load parallel to the VV surface. (Yellow arrow)

Original simulation configuration:

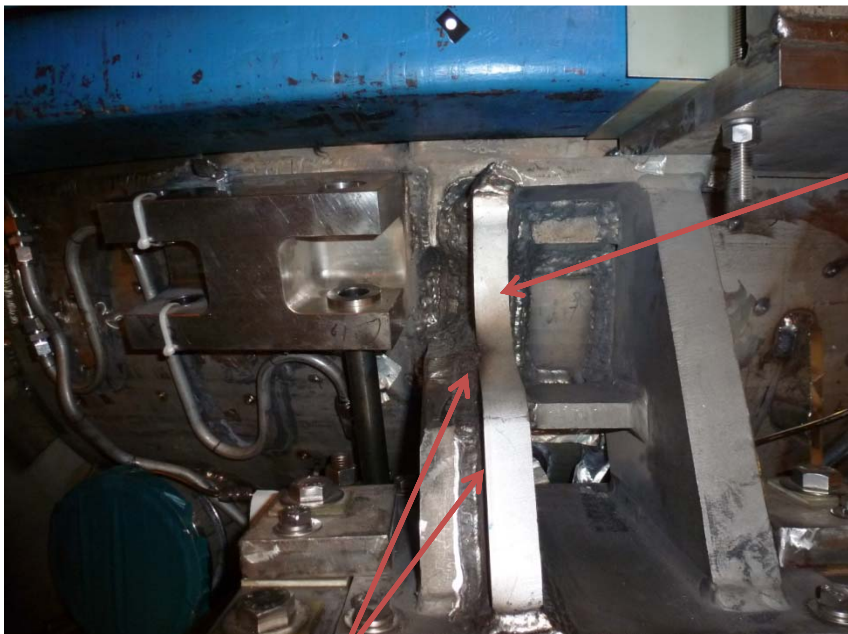
Slide#4 presents the original geometry as reported in the existing document (NSTXU-CALC-12-10-00), March 2012. The weld are 3/8 inch simulation applied only at the outside of Chair-VV contact points (original assumption - see the slide).

Much stronger then the original welding FEA assumptions

As – Built configuration:

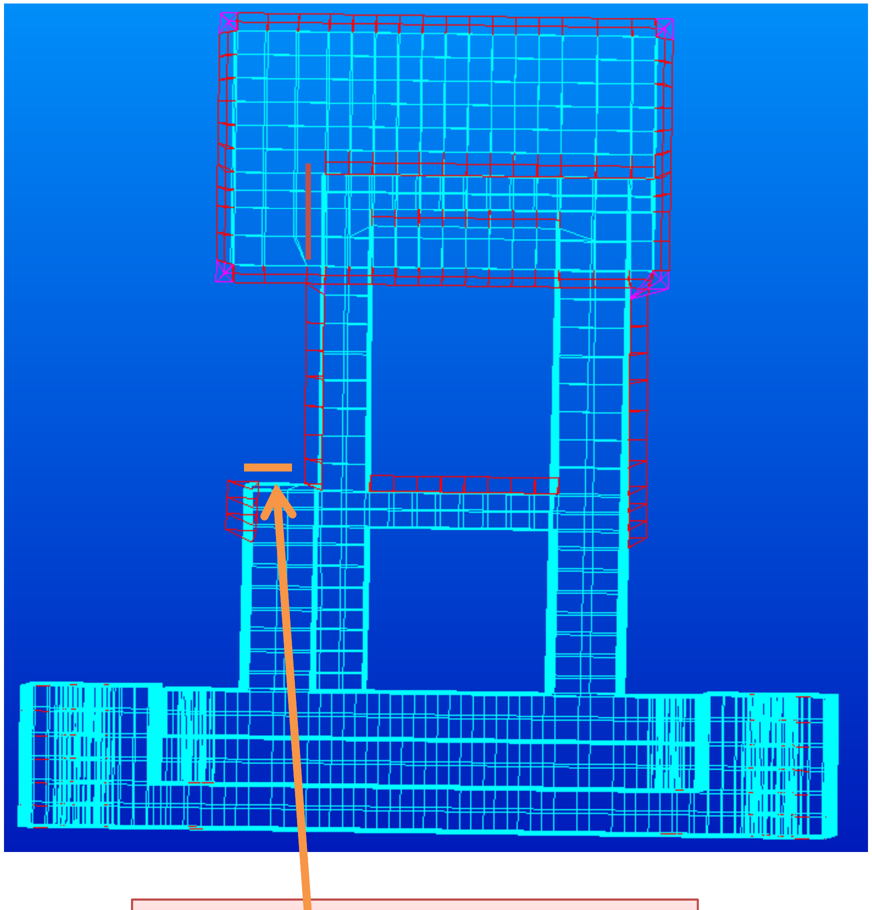
Weld increased to 5/8 inch minimum and totally applied at all contacts between the chair and the VV

New vertical member of the redesigned VV chair



Weld between partially cut out old vertical to the new insert member

Welded inside contact areas



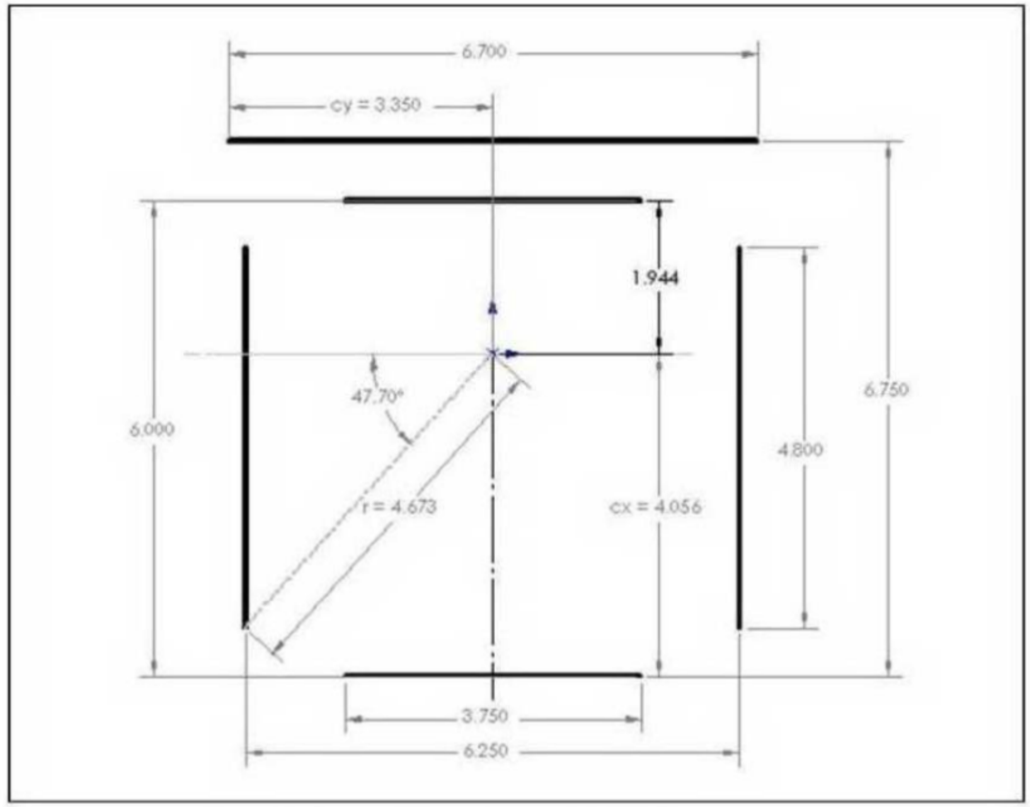


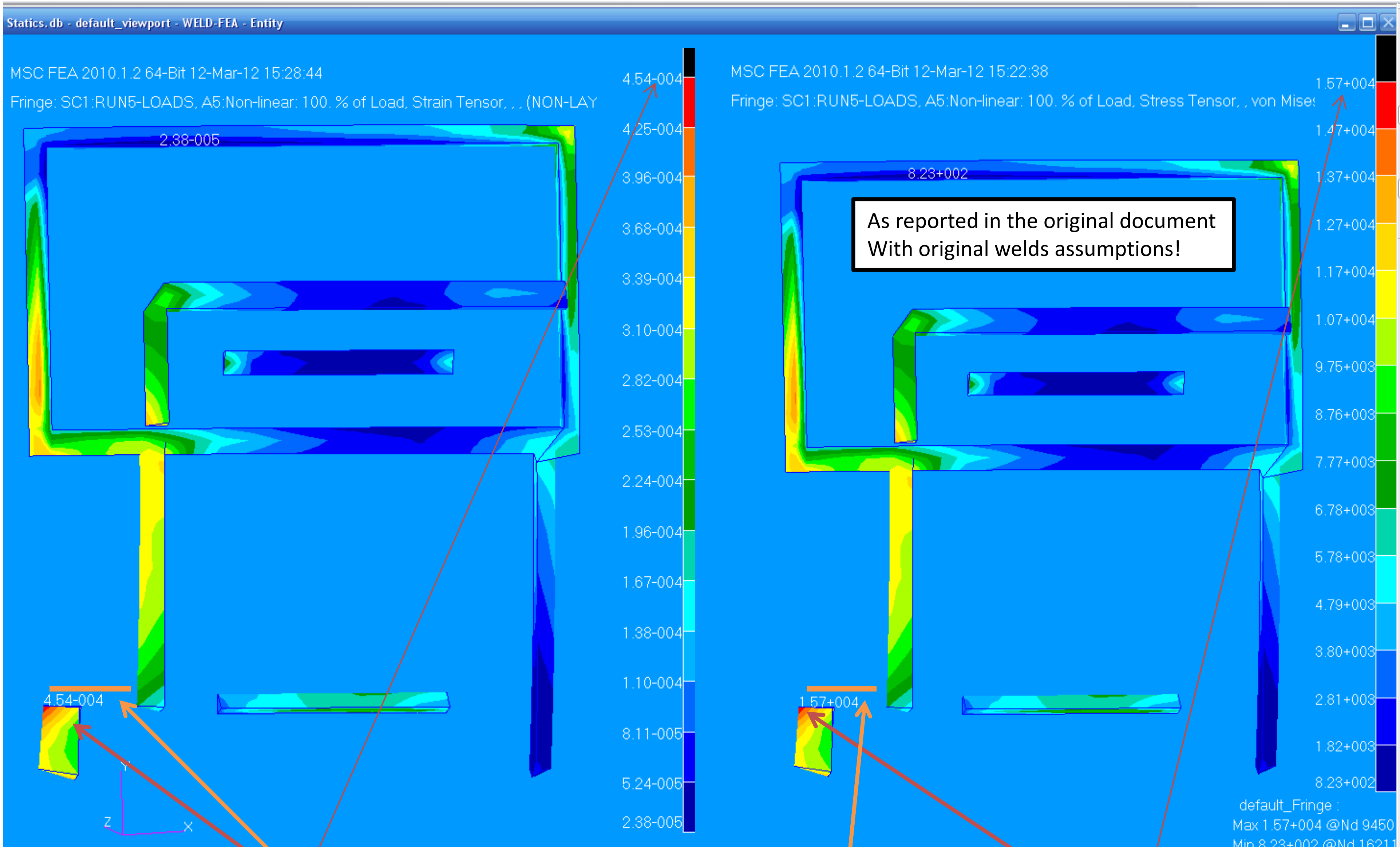
Figure 2 - Support Leg-to-Vessel Weld Dimensions

Area, Extremely desirable to weld, but, may not be readily accessible

Original weld configuration – standard chair
Note: received from M. Smith (E-mail)

Red elements represent the actual welds

Welding scenario for the redesigned VV-Chair support

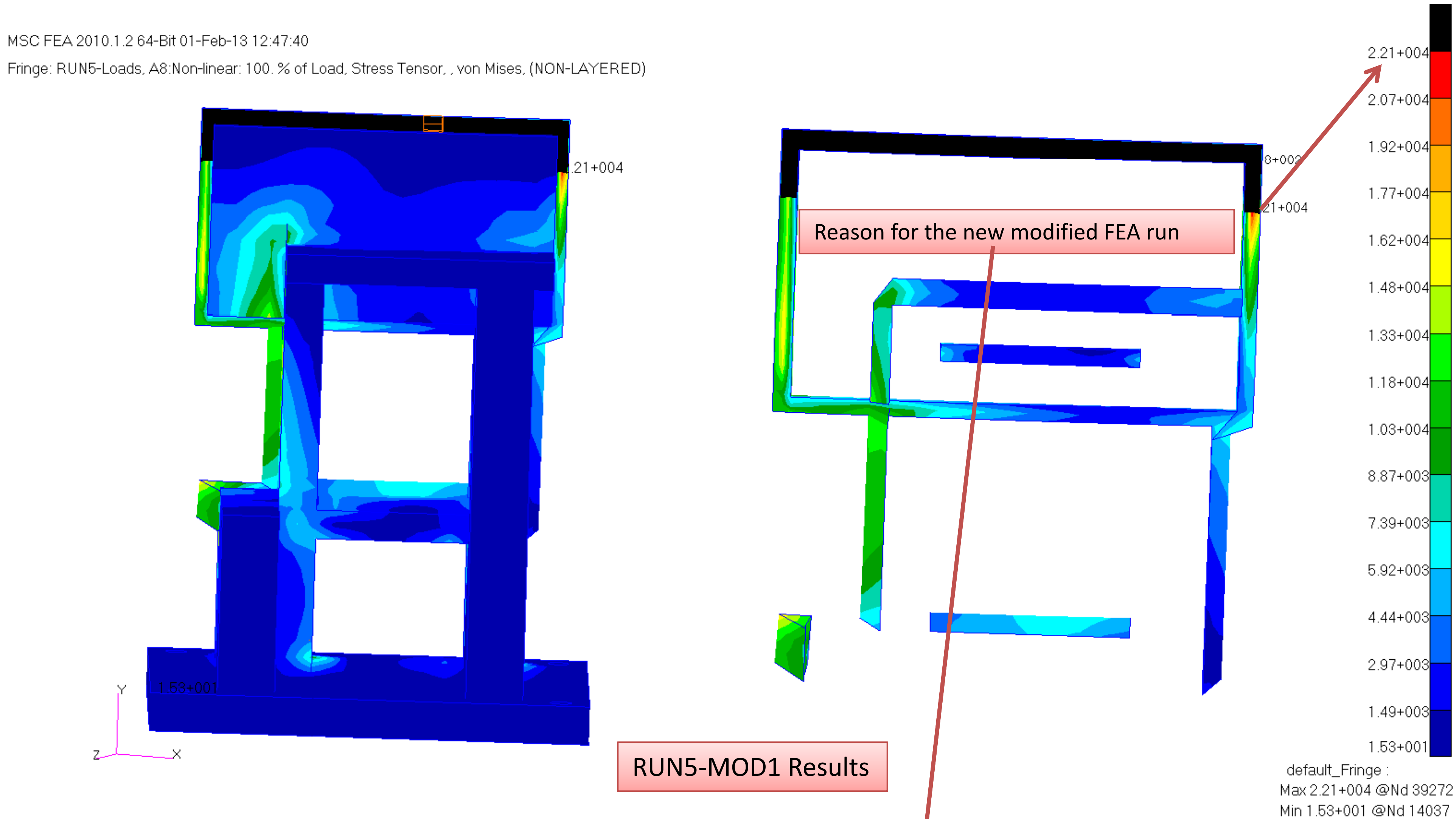


Maximum Strain = .00045

RUN #5 - Results

Maximum Stress = 15,700. psi

Note: If this location is possible to weld, it should decrease the Max stress and/or strain.
Also see the welding slide below!



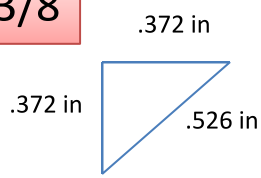
MOD1 consists of removing the weld simulation elements (black area above) as specified by Steve Raftopoulos, that this area may not be welded as originally assumed! RUN5 loads and B.C. have been used exactly as in the original run, so that there is a direct comparison. Findings: weld stress increases in the demonstrated location!

Welds comparative calculations

Comment from M. Smith: Existing welds are 5/8 inch minimum

3/8

Original simulation weld = .526 inch (FEA diagonal)

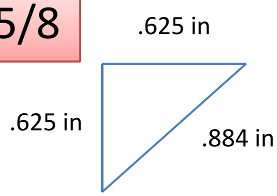


A1 = (.372 X .372)/2 = .0692 in^2

Calculated stress in Run5-Mod1 is = 22,100 psi

5/8

Actual weld dimensions = .884 inch (Increasing diagonal)



A2 = (.625 X .625)/2 = .195 in^2

Equivalent stress based on the increase of the weld area, decreases accordingly

Therefore, equivalent stress =22,100 psi X (.0692 / .195) = 7,842.7 psi

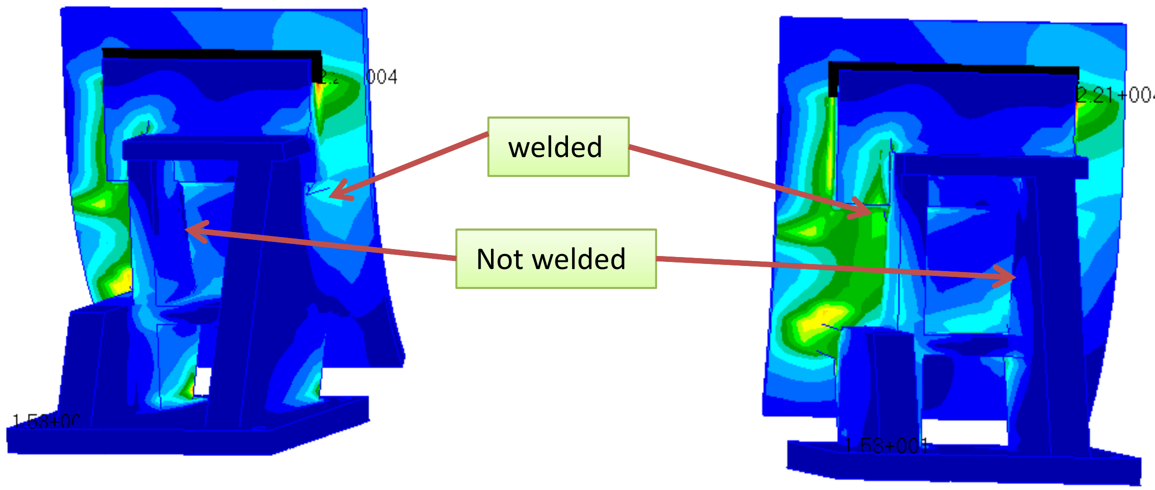
Note: Based on the stainless steel yield (28,000 psi) and 2/3 allowable, this is okay, if there are no other changes of the VV chair structure.

This is an ideal weld simulation as calculated by the FEA technique. The real world may be completely different. Examining some of the actual photos received from Steve Raftopoulos, some of these welds are extremely porous and full of potential stress concentration points which are impossible to simulate in FEA. It is recommended to examine these new welds and smooth them out as much as possible to eliminate the stress concentration points which may exist.

MSC FEA 2010.1.2 64-Bit 11-Feb-13 08:35:08

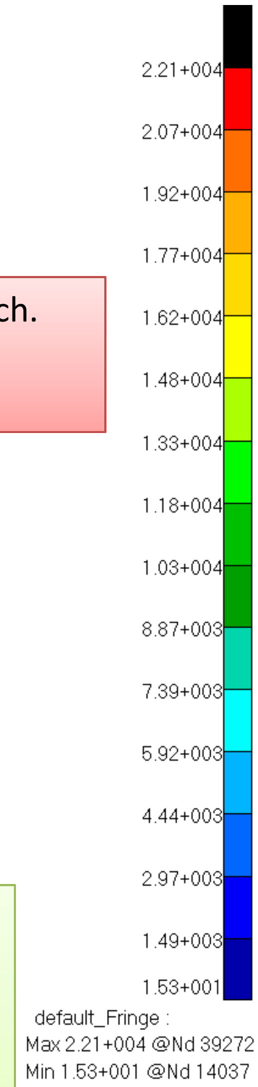
Fringe: RUN5-Loads, A8:Non-linear: 100. % of Load, Stress Tensor, , von Mises, (NON-LAYERED)

Results from Run#5-Mod1



Simulation welds are = 3/8 inch.

As- built welds are = 5/8 inch.



Welds connectivity in the original “Run#5” and “Run5- Mod1” non-linear FEA simulation, (Contour visual check).

Based on the two Run#5 FEA simulations, Run#5-Mod1 predicts the highest welds stresses as indicated above. This clearly changes and increases the weld stress profile when compared to the completely welded VV-plate-chair support as was simulated in the “original” calculation. However, since the “as-built” welds are increased to 5/8 inch and included at all VV Chair contact edges, the as-built stresses decrease even more.

Conclusions

Based on the discussed two Nastran FEA simulations and the visual inspection of the as-built pictures, the general conclusion of the investigation, is that this modification is completely adequate to effectively support the total VV through the 5/8 inch welds as presented in the enclosed pictures and slides.

This investigation demonstrates that creating (changing) new FEA weld geometry was completely unnecessary and probably would have demonstrated the same conclusion. However, if it becomes necessary, the Run#5-Mod1 simulation can be adjusted accordingly.

Appendix

Simulation data for the additional Run#5-Mod1 is stored at the following locations and folder: “Vessel-Support_Chair-Data”

This includes the additional (.bdf and .xdb files). Simply added to the original calculations as reported by the “XSTXU-CALC-12-10-00”.

See: P:\public\Snap-svr\progoff

G:\Nastran P.R. – on ASALEHZ-64PC