NSTX-U

S-FLIP Port Stress Analysis

NSTX-CALC-24-06-00

Sept 25, 2013

Prepared By:

Han Zhang Digitally signed by Han Zhang
DN: cn=Han Zhang, o, ou=PPPL,
email=hzhang@pppl.gov, c=US
Date: 2013.09.26 09:51:36 -04'00'

Han Zhang, PPPL Mechanical Engineering

Reviewed By:

Peter H. Titus Digitally signed by Peter H. Titus
DN: cn=Peter H. Titus, o=Princeton
University, ou=PPPL,
email=ptitus@pppl.gov, c=US
Date: 2013.09.26 10:05:09 -04'00'

Peter Titus, Branch head, Engineering Analysis Division

PPPL Calculation Form

Calculation # **NSTX-CALC-24-06-00** (ENG-032) Revision # **00** WP #, **1672**

Purpose of Calculation: (Define why the calculation is being performed.)
A S-FLIP Diagnostic port will be added to the vacuum vessel at the midplane between bay J and bay I, where has a stress level at 200MPa. Neway Atnafu added this port into his ANSYS workbench model and his simulation shows a peak stress of 39ksi at the edges and stress in other areas below 22ksi. He proposes adding fillet welds at the edges between the tube and the vessel in order to reduce the peak stress but his model doesn't have this weld detail. I have a detailed local model of bay K and J cap which also included bay L and J areas, So I add the s-flip port to this model and compare the results with Neway's results.
Deformation of the vessel may effect alignment of diagnostics. This calculation is not part of s flip analysis. We added as an appendix to document the magnitude of the possible deformations and optical mis-alignments.

References (List any source of design information including computer program titles and revision levels.)
Included in the body of the calculation

Assumptions (Identify all assumptions made as part of this calculation.)
Loads are mapped from the global TF outer leg analysis (http://nstx-upgrade.pppl.gov/Engineering/Calculations/index_Calcs.htm) and equilibrium number 79 loads are used which have maximal TF outer leg OOP force and torque.

Calculation (Calculation is either documented here or attached)
See the attached document.

Conclusion (Specify whether or not the purpose of the calculation was accomplished.)
In my detailed model, three cases, including no welding, seam welding and stitch welding, are compared. Loads are mapped from the global TF outer leg analysis (http://nstx-upgrade.pppl.gov/Engineering/Calculations/index_Calcs.htm) and equilibrium 79 loads are used which have maximal TF outer leg OOP force and torque. For these three cases, the peak stress is reduced from 86ksi to 60ksi but membrane and bending stresses are not very different. Then I added a flange to this s flip. Peak stress is reduced more to 54.9ksi and membrane and bending stresses are also reduced a little. However, the stress is still higher than heat affected zone allowable**.** Then the elasto-plastic analysis was performed which shows that peak stresses are highly localized, yield areas are very small and most of the material is safe enough to take the load. For fatigue requirement, it remains a problem. And the welding details should be added into the inspection list. The loads were further increased to 2x, the large displacement solution envoked, and the model doesn't collapse. The safety factor of 2 for the limit analysis [3] can be satisfied.
In appendix 1, two diagnostic signals are selected, which are bay J to centerstack and bay G to B. Maximal tip deflection is from bay J to CS: about 7.8mm (0.31"). The other one, from bay G to B, is very small, less than 1mm. Considering other scenarios with different deflections and different orientations, it is better that the sensor size to be bigger than 16mm (0.62").

Cognizant Engineer's printed name, signature, and date Neway D. Atnafu

Digitally signed by Neway D. Atnafu
DN: cn=Neway D. Atnafu, o, ou, email=natnafu@pppl.gov, c=US
Date: 2013.09.30 08:10:50 -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 Peter H. Titus

Digitally signed by Peter H. Titus
DN: cn=Peter H. Titus, o=Princeton University, ou=PPPL, email=ptitus@pppl.gov, c=US
Date: 2013.09.26 10:05:41 -04'00'

Table of Contents

List of References

[1] H. Zhang, "Analysis of TF Outer Leg", http://nstx-upgrade.pppl.gov/Engineering/Calculations/1_Torus_Systems/1_3_Magnets/Toroidal_Field_Magnets-WBS132/CALC-132-04/NSTXU-CALC-132-04-01.pdf.

[2] N. ATNAFY, "NSTX UPGRADE Stress Analysis of the Bay L and 2nd Neutral Beam Upgrades", http://nstx-upgrade.pppl.gov/Engineering/Calculations/2_Heating_Current_Drive/2_4_NBI/CALC-24-05/NSTXU-CALC-24-05-00%20(BayL_2nd%20NB%20Upgrade).pdf

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

Executive Summary

The S flip port will be added to the vessel at the midplane between bay J and bay I, where has the stress level is at 200MPa. Neway Atnafu added this port into his ANSYS workbench model and the simulation shows a peak stress of 39ksi at the corner and stress in other areas below 22ksi. He has proposed welding the port to vessel which will help to reduce the peak stress but his model doesn't have this weld detail. I have a detailed local model of bay K and J cap which also included bay L and J areas. So I add the s flip port to this model and I compare with Neway's results.

In my detailed model, three cases, including no welding, seam welding and stitch welding, are compared. Loads are mapped from the global TF outer leg analysis (http://nstx-upgrade.pppl.gov/Engineering/Calculations/index_Calcs.htm) and equilibrium 79 loads are used which has maximal TF outer leg OOP force and torque. For these three cases, the peak stress is reduced from 86ksi to 60ksi but membrane and bending stresses are not very different. Then I added flange to this s flip, peak stress is reduced more to 54.9ksi and membrane and bending stresses are also reduced a little. However, the stress is still higher than heat affected zone allowables. Then the elasto-plastic analysis shows that peak stresses are highly localized, yield areas are very small and most of the material is safe enough to take the load. For fatigue requirement, it remains a problem. And the welding should be added into the inspection list. The loads were further increased to 2x and the model doesn't collapse. The safety factor of 2 for limit analysis required by the NSTX criteria [3] can be satisfied.

Deformation of the vessel may effect alignment of diagnostics. This calculation is not part of s flip analysis. We added as an appendix to document the magnitude of the possible deformations and optical mis-alignments. Since the ports and vessel deform upon the load from TF outer legs, the deformation may affect the diagnostic signal and thus we did this simulation. Two diagnostic signals are selected, which are bay J to centerstack and bay G to B. Maximal tip deflection is from bay J to CS: about 7.8mm (0.31"). The other one, from bay G to B, is very small, less than 1mm. Maximal deflection with equilibrium 79 loads is 7.8mm (0.31"). Considering other scenarios with different deflections and different orientations, it is better that the sensor size to be bigger than 16mm (0.62").

Neway's model and results

Neway uses his workbench model shown in figure 1 and figure 2. Thise figures show the reinforcement he included. He applied a fine mesh to the s flip area. He gave 3E7 lbf-in torque to the top and bottom of vessel. He mapped the PF 4 and 5 loads from Maxwell. Vacuum pressure is also applied. His simulation shows the peak stress is 46ksi at the corner and stress of all the other area is below 22ksi (figure 3).

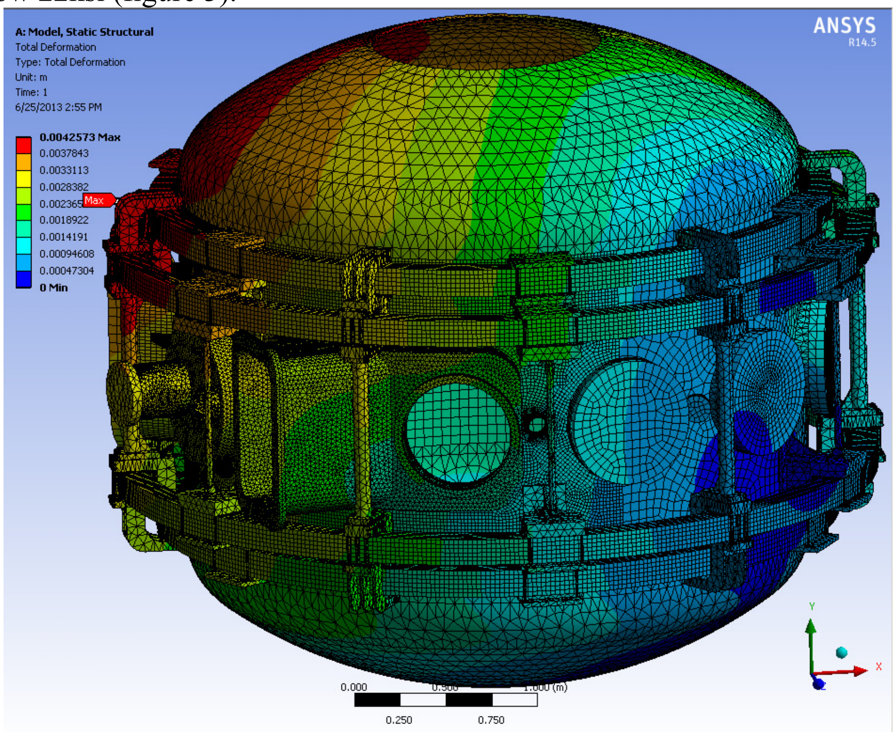

Figure 1: Neway's workbench model.

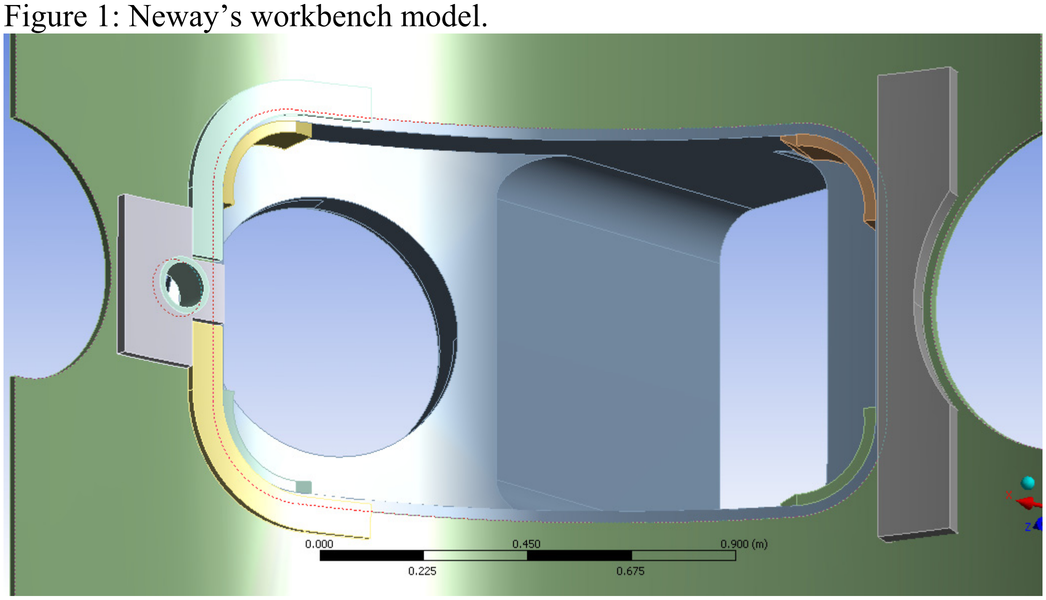

Figure 2: Reinforcements inside the vessel.

Neway's result using workbench, with torque, vacuum pressure and PF4,5 load.
Yellow area stress is 16~22 ksi, red area 22~46 ksi.

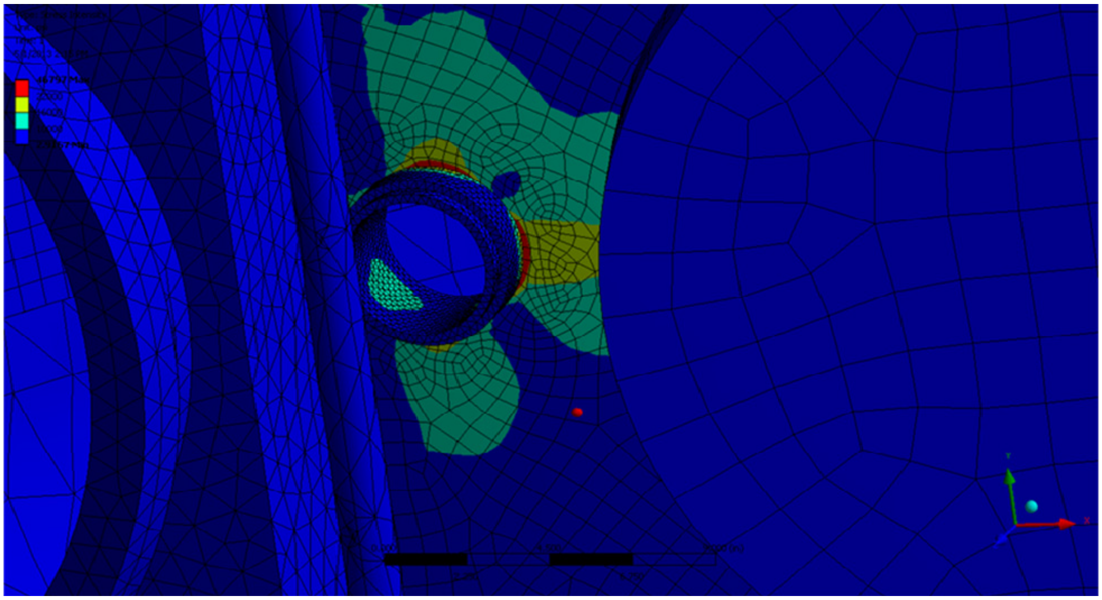

Figure 3: Neway's result.

Han's model and results

My detailed model is shown in figure 4. S flip was added and reinforcements inside the vessel are same as Neway's. The loads are from my NSTX full model (equilibrium 79, ref 1) and vacuum pressure, but no PF 4, 5 loads. Three cases (figure 5) are simulated, which are no welding, seam welding and stitch welding.

Using my partial model and add s flip. BCs are displacements from my NSTX full model (scenario 79 loads) and vacuum pressure, no PF 4,5 loads.

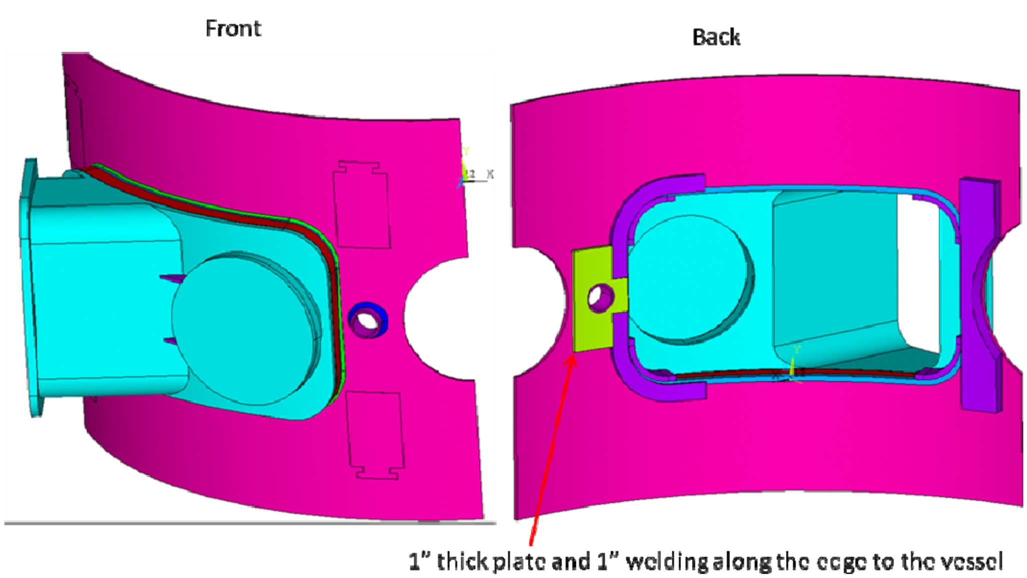

Figure 4: Han's detailed model.

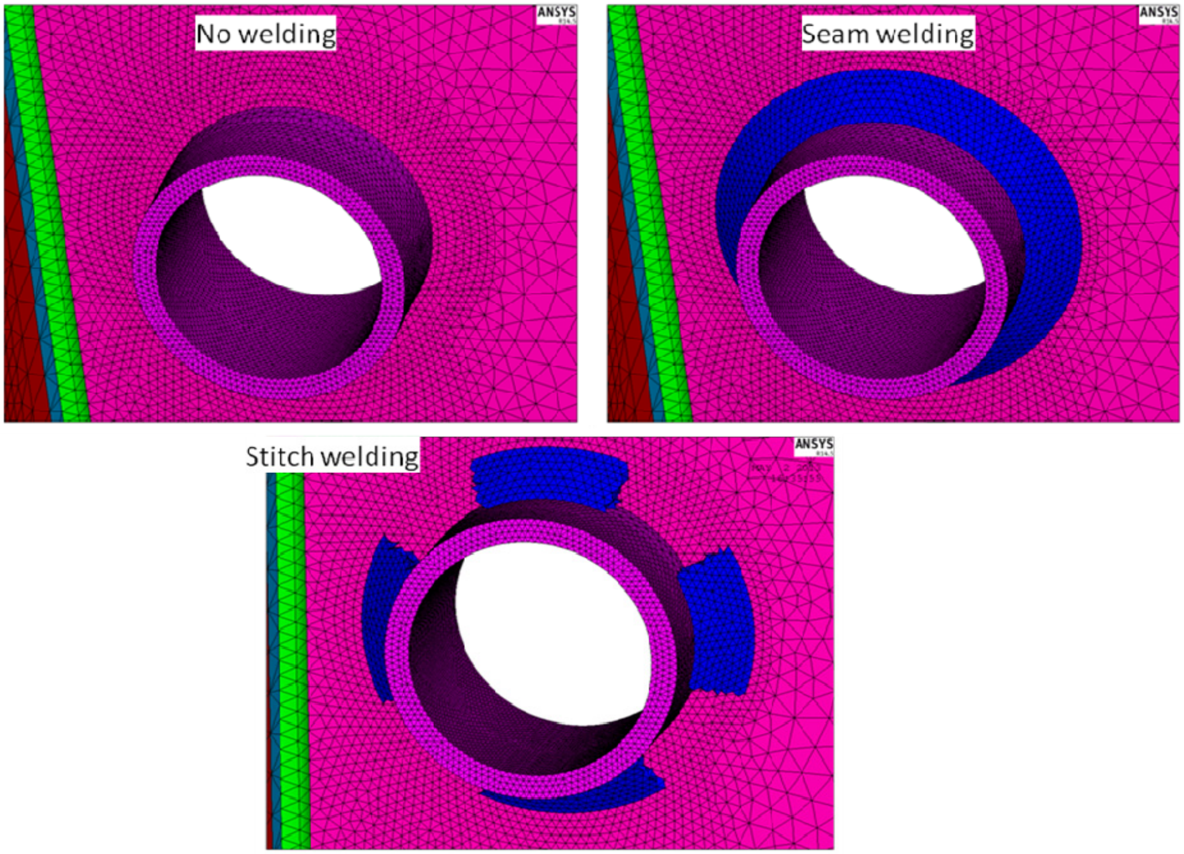

Figure 5: Three cases, including no welding, seam welding and stitch welding, are simulated.

Stress results of these cases are shown in figure 6. As Neway expected, welds can reduce the peak stress significantly. There is always stress concentration at geometrically discontinuous areas, e.g. the corner, but these peak stresses are acceptable because they are highly localized. In reality such stress peaks would not occur due to localized yielding. In this static analysis, they can be ignored. (But for fatigue analysis, they cannot.). When looking at the linearized stress of these three cases (figure 7), they are not very different: membrane stress is ~200 MPa (29 ksi) and membrane+bending is ~320 MPa (46 ksi).

Notice that Neway is using workbench and I am using ANSYS classic. Recalling our previous analysis results, my model always has higher deformations and stresses than Neway's, which we don't know why. See the Appendix 1 of ref 2 for previous comparison. The displacements around the new neutral beam port reinforcement will be measured as a part of the structural benchmarking effort intended to substantiate DCPS algorithms. This should resolve the discrepancy. My partial detailed model has a little higher stress than full model because it has vacuum pressure and also does not model some port covers. Figure 8-10 compares the radial displacement, theta displacement and stress of the three models.

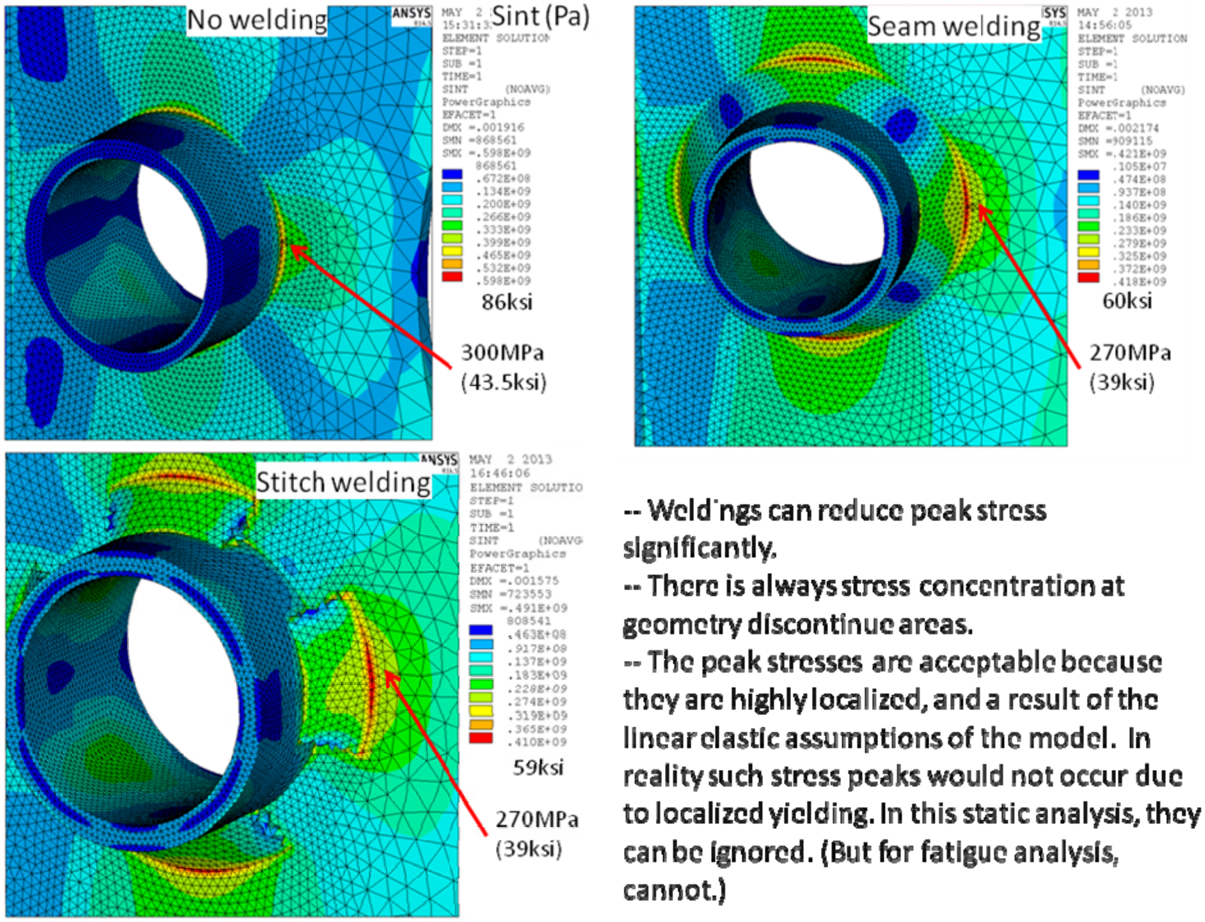

Figure 6: Stresses of these three cases.

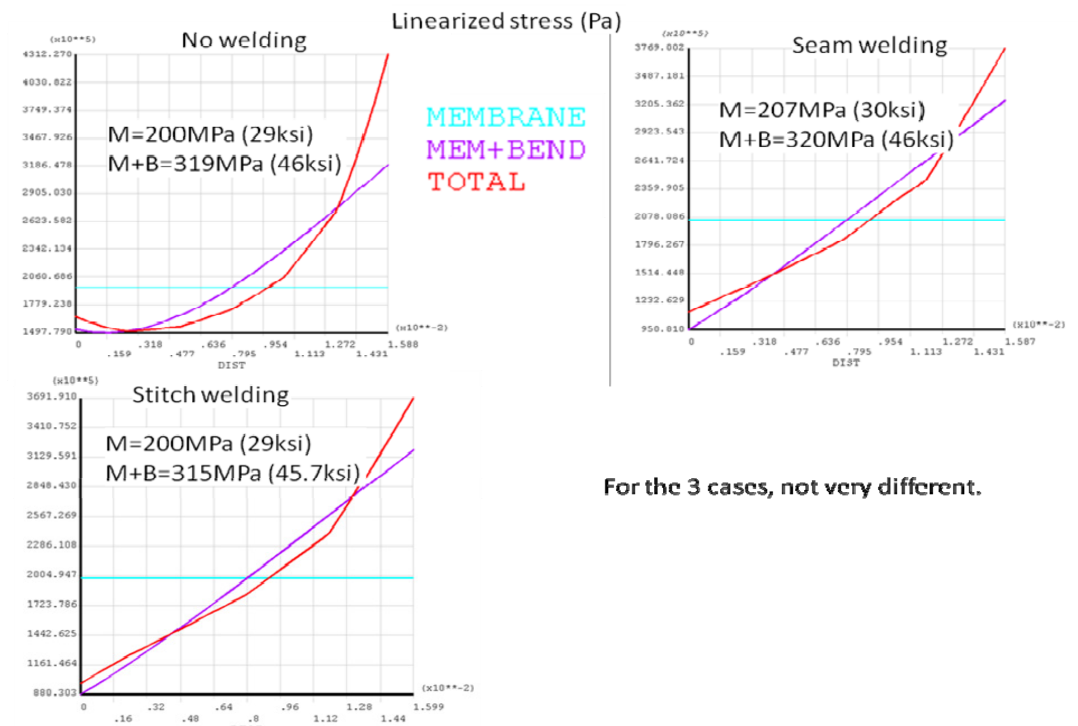

Figure 7: Linearized stress (Pa).

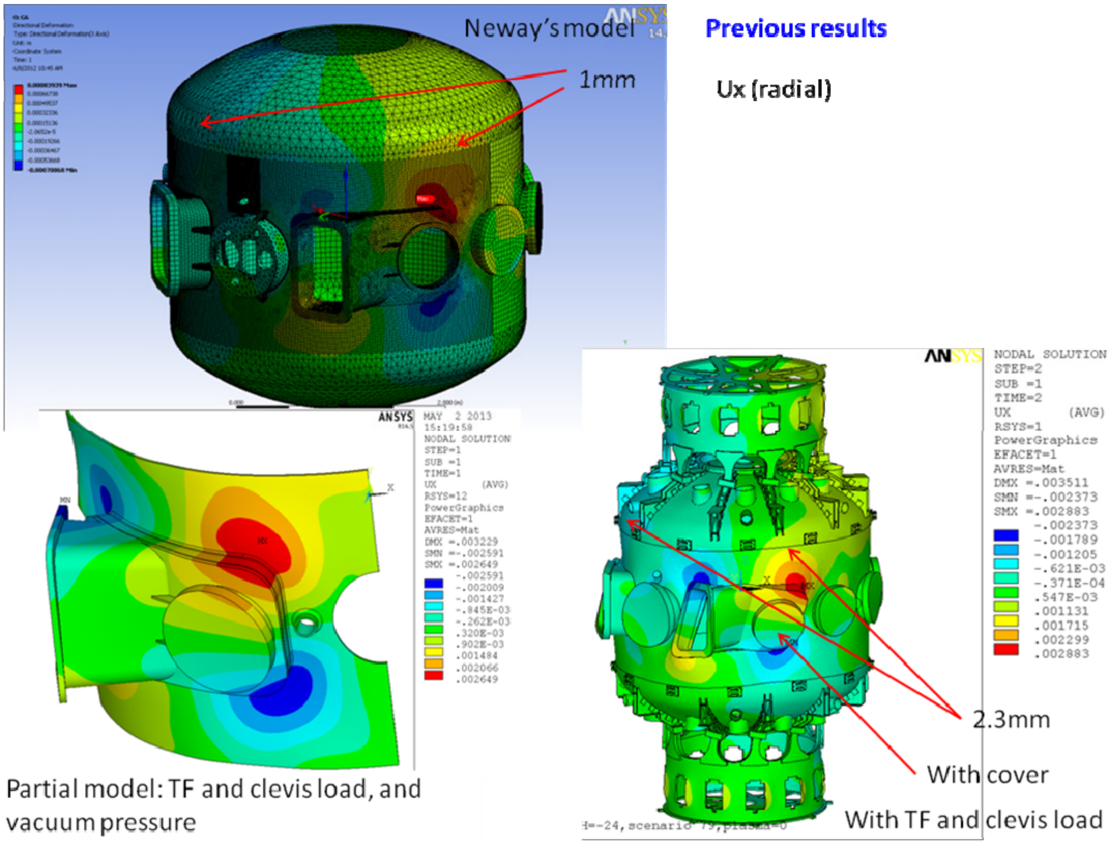

Figure 8: previous results: radial displacement.

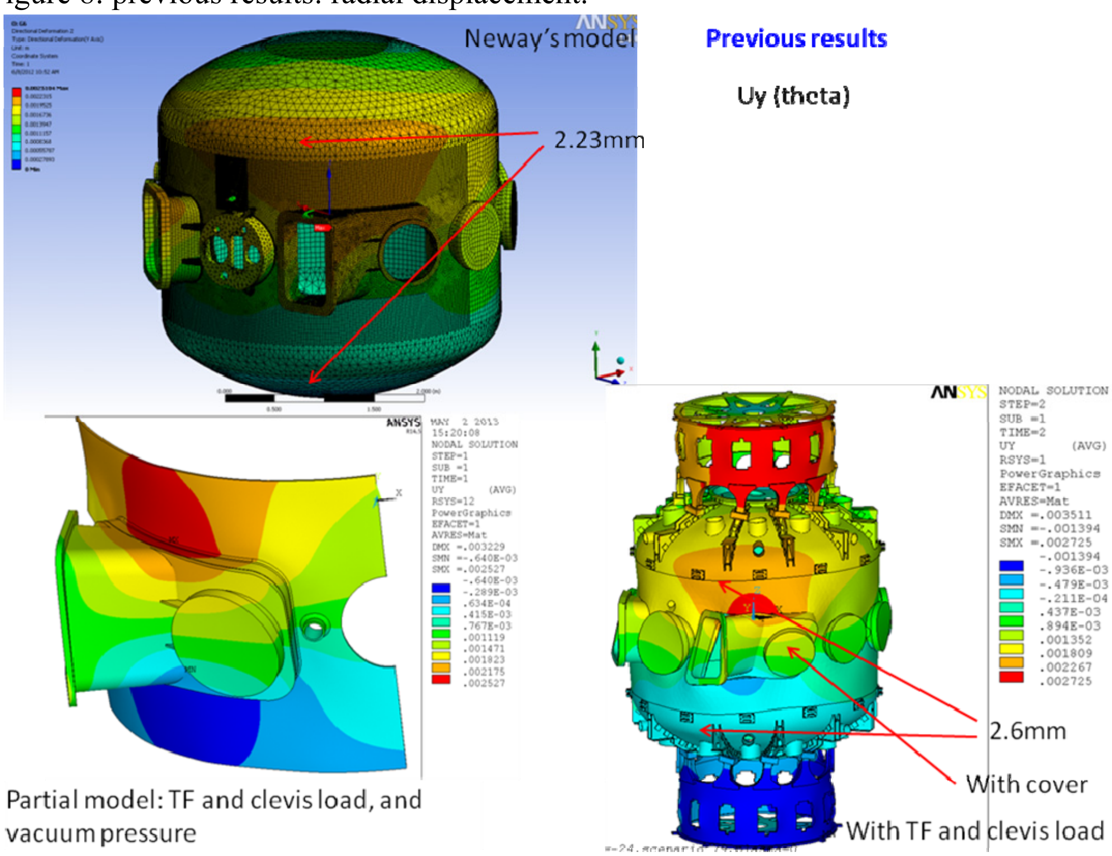

Figure 9: previous results: theta displacement.

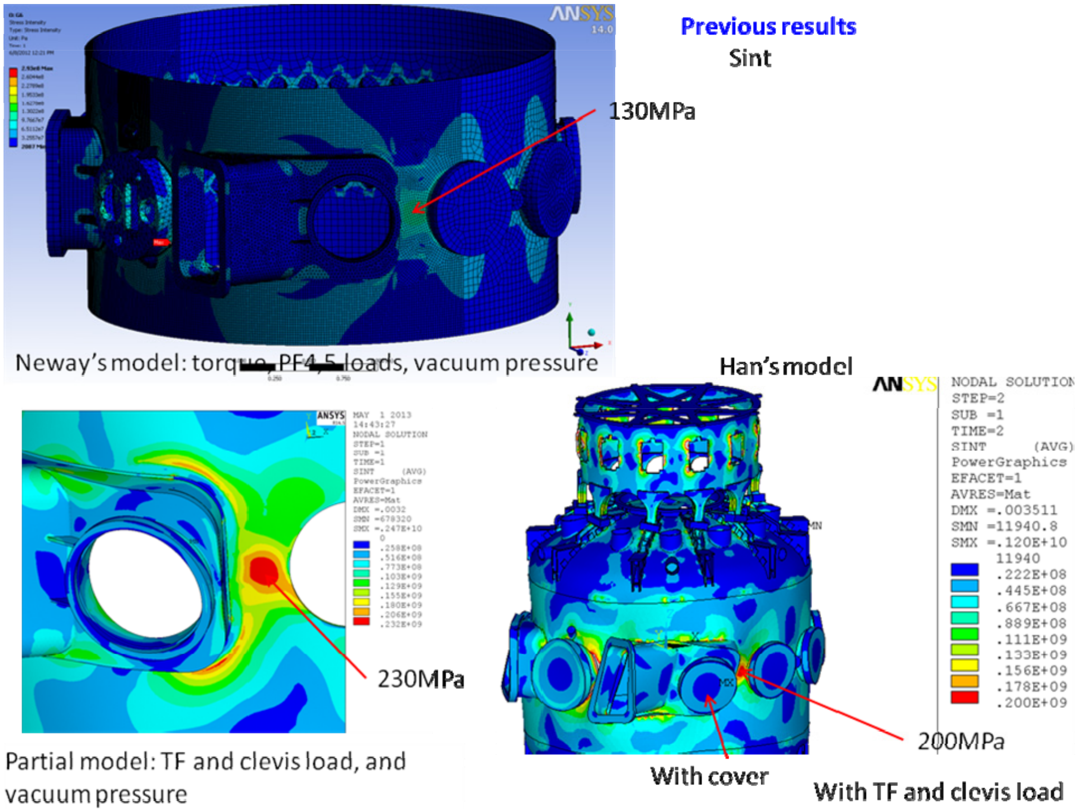

Figure 10: previous results: stress.

Then the model was updated with latest design (figure 11). However, results are similar to what they were before. Then by adding a flange to the model (figure 12), peak stress is reduced (from 61.3ksi to 54.9ksi), the membrane stress is reduced (from 30ksi to 27.5ksi), and membrane+bending stress is reduced (from 45.8ksi to 41.3ksi) are further reduce a little (figure 13).

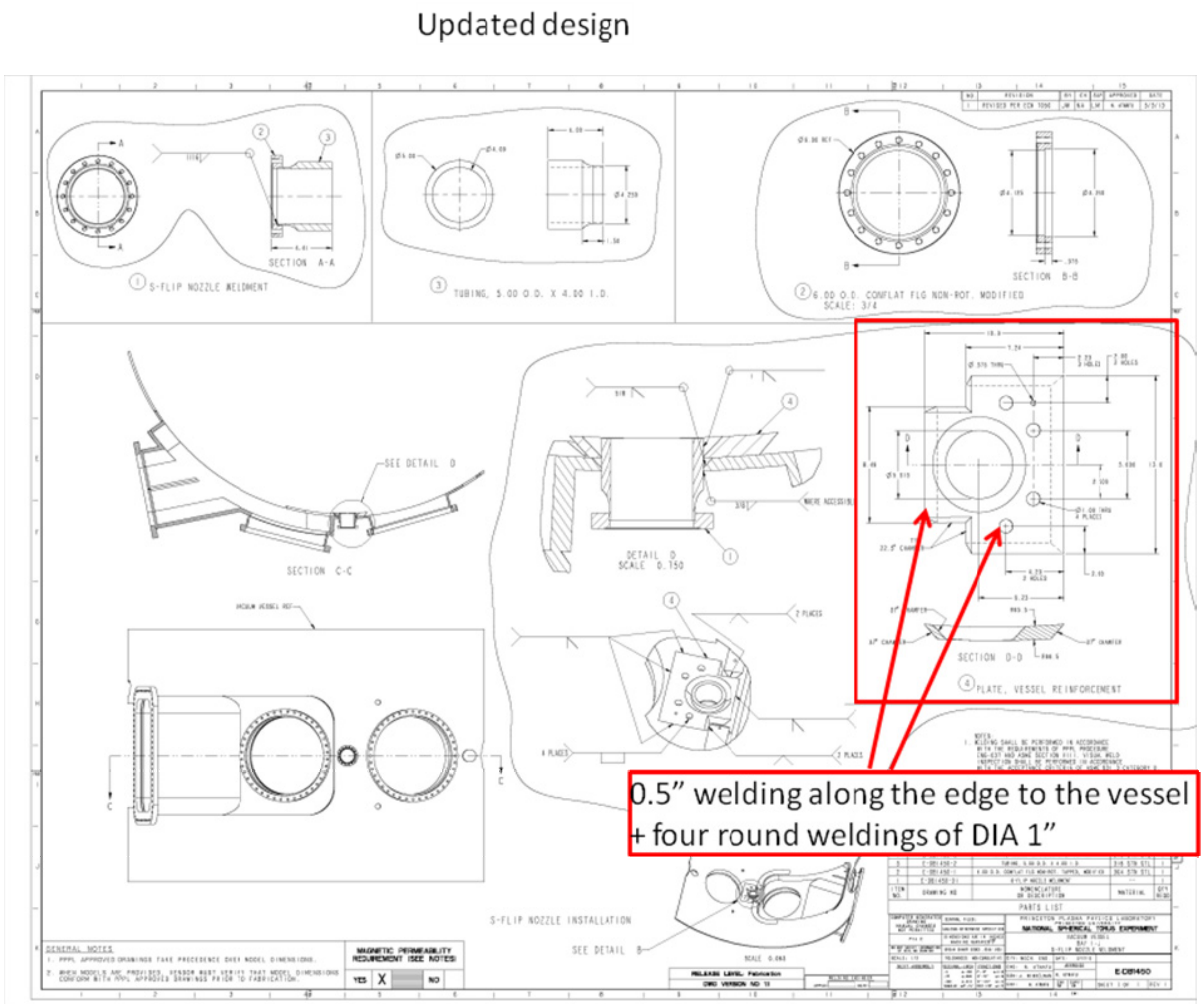

Figure 11: updated design.

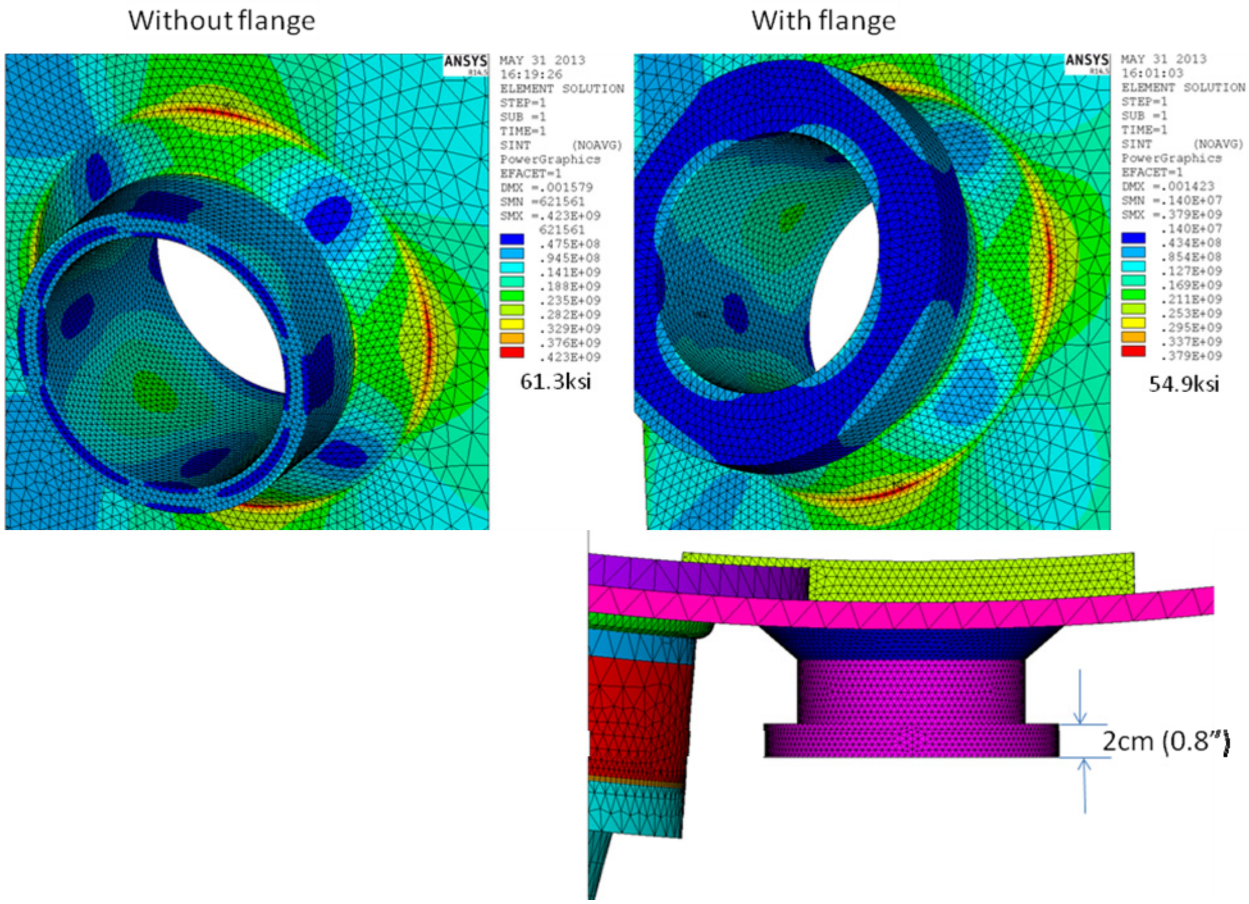

Figure 12: model of cases, without flange and with flange.

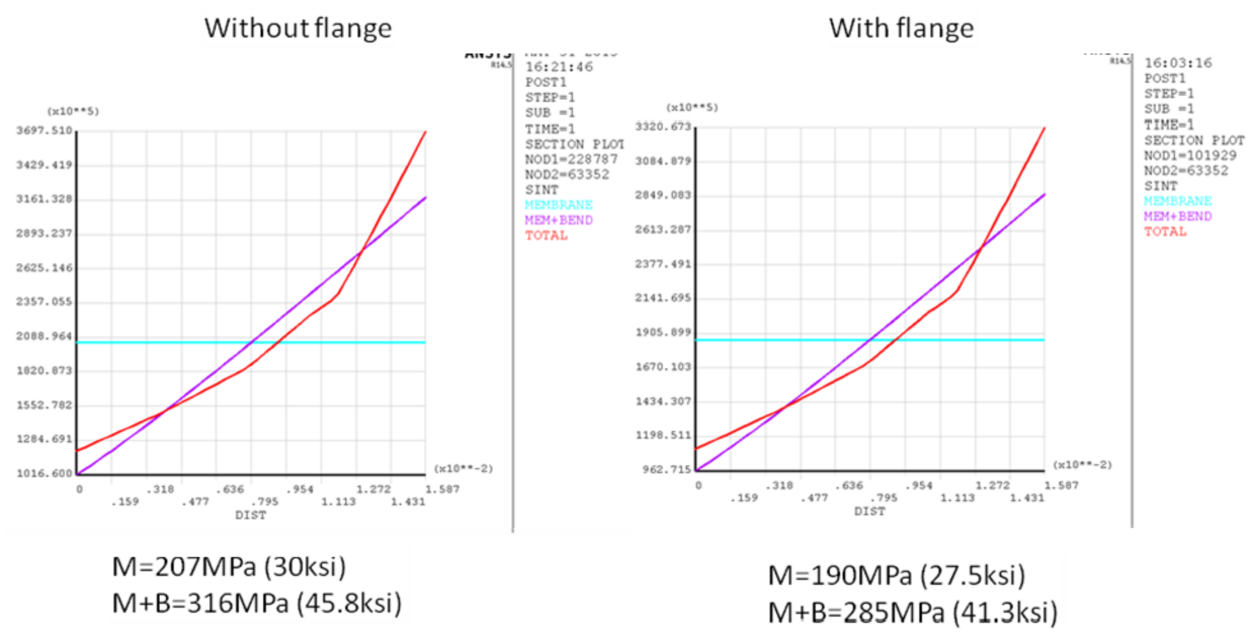

Figure 13: comparison of cases, without flange and with flange.

Allowables

Static Stress Criteria

	Sm	Sb	
Vessel 304 Away from weld	30 ksi 207 MPa	45 ksi 310 MPa	Mill Certs for the 304 Vessel Show a 45 ksi Yield
304 Vessel in Heat Effected Zone	20 ksi 138 MPa	30 ksi 206MPa	
316	183 MPa	275 MPa	
316 weld	160MPa	241MPa	
AISC/ASME/AWS 304 weld	20 ksi (w/PT)	14ksi (w/Visual)	

Due to the 2" weld around s flip, heat affected zone allowable should be used, which is 30ksi (206MPa) for Sb and 20ksi (128MPa) for Sm. Thus the stress is higher. But it is still very localized, Peter Titus suggested to do an elasto-plastic analysis to see how much area will above yield.

Elasto-plastic analysis

The same model is used, but material properties changed to non-linear, yield at 206MPa and tangent modulus is 1.8GPa. 5 cases are run which has 1x BC, 1.1x BC, 1.2x BC, 1.3x BC and 1.4X BC respectively. Because I applied displacement BCs to the partial model, reaction forces and moments are compared to see how much load increasing is.

	1x load (elastic)	1x disp (elasto-plastic)	1.1x disp (elasto-plastic)	1.2x disp (elasto-plastic)	1.3x disp (elasto-plastic)	1.4x disp (elasto-plastic)
Fx	-304105.3	-309547.8	-333013	-356497	-379961.3	-403434.4
Fy	-137.5615	-288.22	-308.5086	-329.7221	-352.1536	-377.2797
Fz	-2.88E-02	-17.53992	-19.19226	-20.7591	-22.3542	-24.10799
Mx	1245551	1228852	1346565	1462056	1574724	1684344
My	-42688.44	-40557.53	-43902.87	-47031.4	-49909.19	-52528.59
Mz	6.30E-03	-266.0182	-320.2027	-379.3395	-442.8597	-510.9269
Fxi/Fx1	1.00	1.02	1.10	1.17	1.25	1.33
Mxi/Mx1	1.00	0.99	1.08	1.17	1.26	1.35
Myi/My1	1.00	0.95	1.03	1.10	1.17	1.23

--Because Fy, Fz and Mz are almost zero (very small), they are not included when calculating the load ratio.
--When applying 1x displacement to the elasto-plastic model, the load is almost same (within 5%) as the elastic model.
--when applying 1.1x displacement, the load is ~1.07x.
--when applying 1.2x displacement, the load is ~1.15x.
-- when applying 1.3x displacement, the load is ~1.23x.
-- when applying 1.4x displacement, the load is ~1.3x.

Figure 14: Comparison of total reaction forces (in cylindrical system)

Figure 15-17 plot the elastic strain and plastic strain. Elasto-plastic results show that peak stresses are highly localized and yield areas are very small, even when the load goes above the designed load. With static requirement, this part should be safe because just a very small area will yield and most of the material is safe enough to take the load. For fatigue requirement, it remains a problem. And the welds should be added into the inspection list.

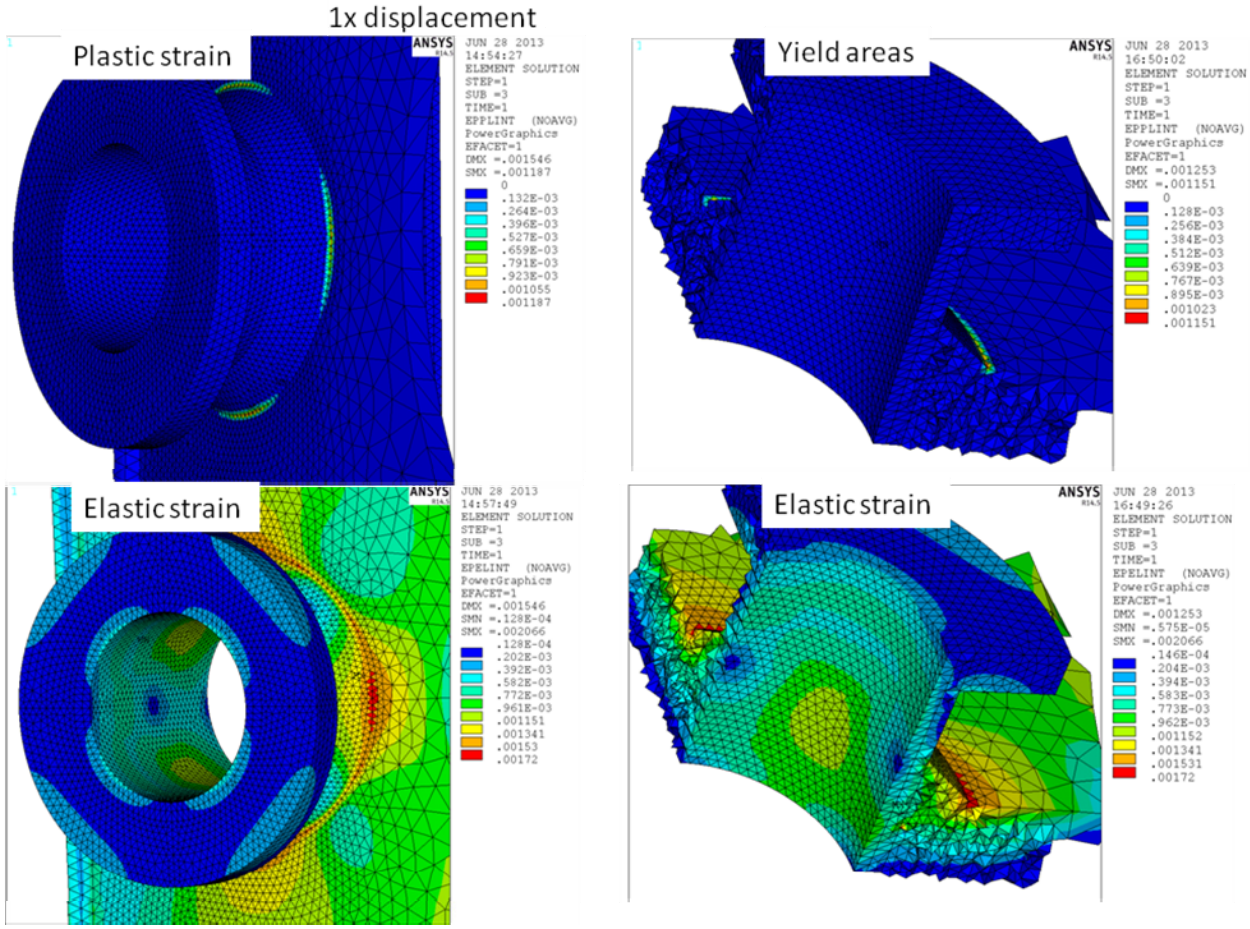

Figure 15: strain when apply 1x displacement.

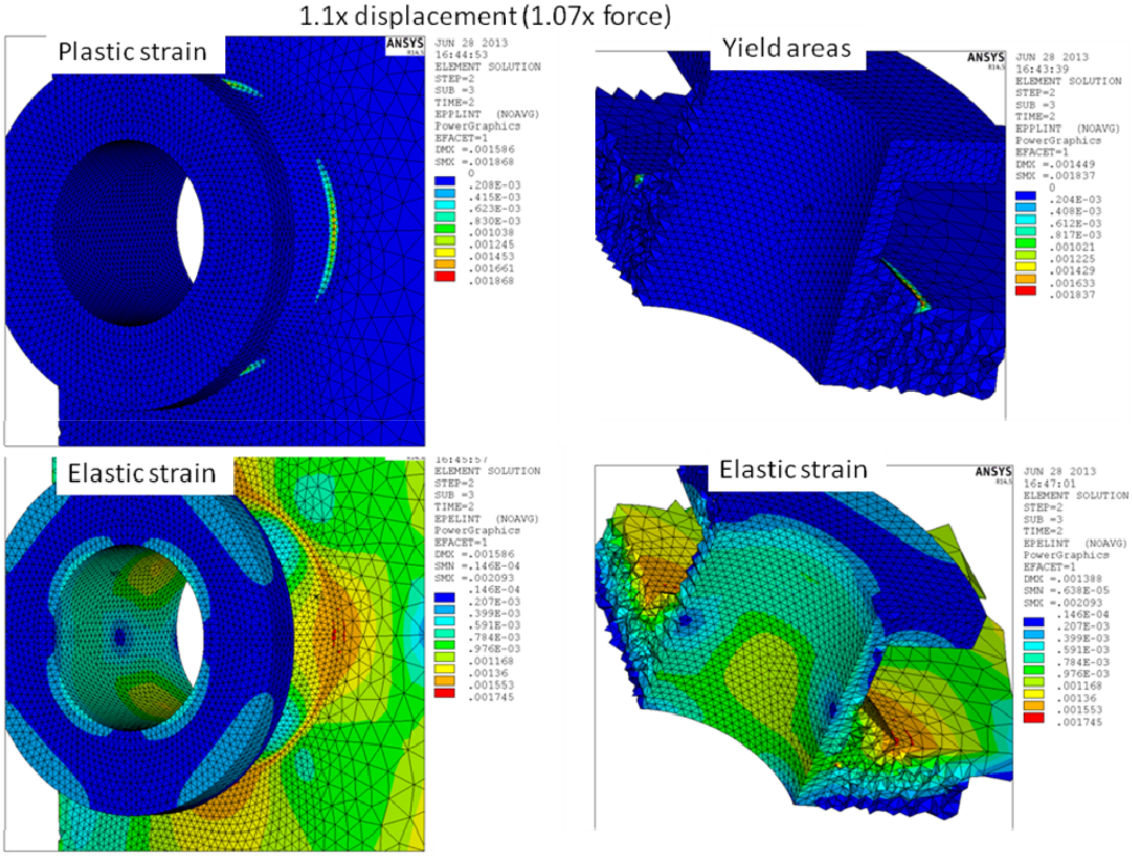

Figure 16: strain when apply 1.1x displacement (1.07x force).

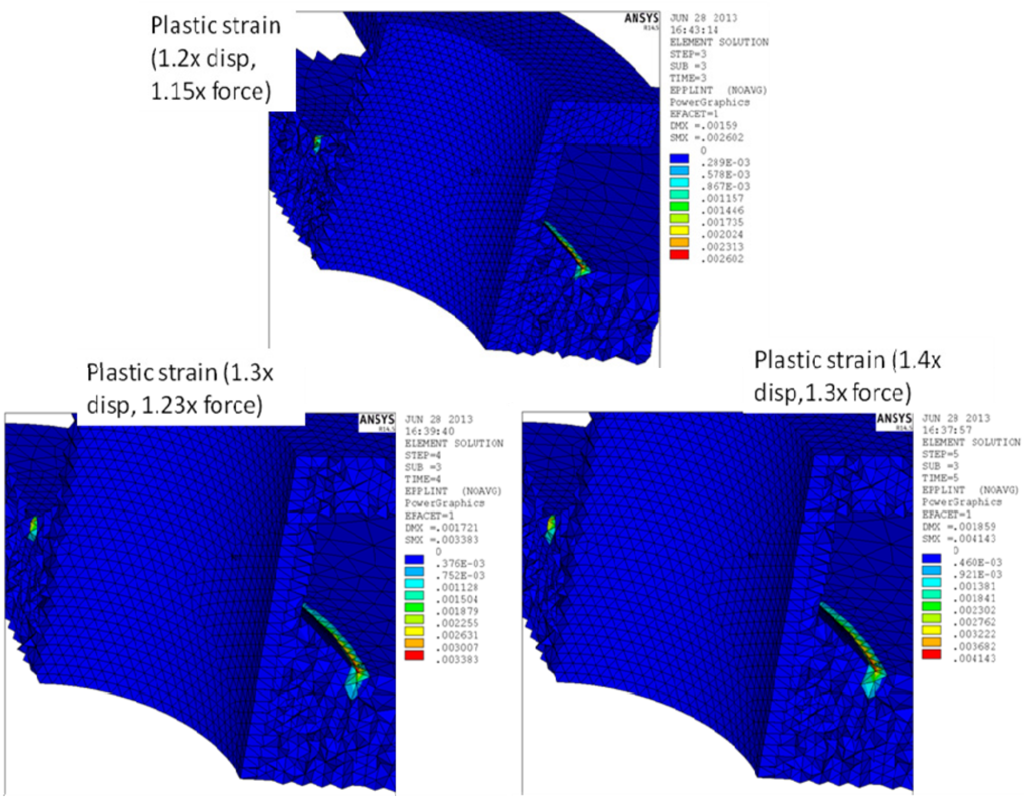

Figure 17: plastic strain upon higher loads.

Buckling analysis

P. Titus indicates that NSTX criteria [3] require demonstration that it doesn't collapse with twice the load. In my model, I applied displacement BCs to the partial model, when I increase the load to 3.1x, the load is about 2x (figure 18), the model still doesn't collapse (figure 19-22).

Elasto-plastic buckling analysis shows that the safety factor should be >2. Force load has been increased to ~2x but still no buckling appears.

	Fx	Fy	Fz	Mx	My	Mz	Fxi/Fx1	Mxi/Mx1	Myi/My1
1x load (elastic)	-304105.3	-137.562	-2.88E-02	1245551	-42688.44	6.30E-03			
0.5x disp (elasto-plastic)	-192891.5	-145.17	-9.14	621164.34	-21373.36	-69.69	0.634292	0.498706	0.500688
1x disp (elasto-plastic)	-309552.51	-287.46	-17.59	1228929.35	-40568.12	-265.94	1.017912	0.986655	0.95034
1.1x disp (elasto-plastic)	-333015.39	-308.01	-19.17	1346613.14	-43909.4	-320.36	1.095066	1.081139	1.028612
1.2x disp (elasto-plastic)	-356498.68	-329.33	-20.77	1462085.64	-47035.37	-379.27	1.172287	1.173846	1.101841
1.3x disp (elasto-plastic)	-379962.33	-351.94	-22.43	1574744.4	-49911.84	-442.95	1.249443	1.264295	1.169224
1.4x disp (elasto-plastic)	-403435.09	-377.05	-24.05	1684358.2	-52530.68	-510.84	1.32663	1.3523	1.230573
1.5x disp (elasto-plastic)	-426904.43	-399.15	-25.88	1790699.85	-54909.99	-583.27	1.403805	1.437677	1.28631
1.6x disp (elasto-plastic)	-450387.84	-423.04	-27.86	1893446.28	-57063.99	-660.18	1.481026	1.520168	1.336769
1.7x disp (elasto-plastic)	-473861.91	-447.93	-30.16	1992290.69	-58981.6	-741.12	1.558217	1.599526	1.38169
1.8x disp (elasto-plastic)	-497298.21	-474.69	-32.73	2087157.84	-60670.52	-825.95	1.635283	1.67569	1.421255
1.9x disp (elasto-plastic)	-520712.68	-502.1	-35.72	2178271.7	-62153.21	-914.91	1.712278	1.748842	1.455988
2x disp (elasto-plastic)	-544066.2	-531.82	-38.93	2265862.12	-63454.59	-1007.98	1.789072	1.819164	1.486474
2.1x disp (elasto-plastic)	-567305.72	-563.6	-42.52	2350010.89	-64599.6	-1104.89	1.865491	1.886724	1.513296
2.2x disp (elasto-plastic)	-590422.42	-594.59	-46.28	2430998.65	-65596.22	-1205.85	1.941507	1.951746	1.536643
2.3x disp (elasto-plastic)	-613304.08	-626.08	-50.21	2508959.21	-66450.59	-1310.1	2.016749	2.014337	1.556657
2.4x disp (elasto-plastic)	-635857.99	-660.26	-54.41	2584164.01	-67231.19	-1417.87	2.090914	2.074716	1.574944
2.5x disp (elasto-plastic)	-658056.93	-691.36	-58.7	2656835.36	-67915.06	-1528.62	2.163911	2.13306	1.590964
2.6x disp (elasto-plastic)	-679853.16	-718.93	-63.13	2727205.07	-68487.48	-1642.17	2.235585	2.189557	1.604373
2.7x disp (elasto-plastic)	-701141.48	-746.08	-67.88	2795337.45	-68948.92	-1757.79	2.305588	2.244258	1.615183
2.8x disp (elasto-plastic)	-721915.05	-770.58	-72.41	2861274.15	-69279.67	-1875.51	2.373898	2.297195	1.622931
2.9x disp (elasto-plastic)	-742039.82	-792.08	-77.33	2925068.65	-69545.93	-1994.95	2.440075	2.348413	1.629168
3x disp (elasto-plastic)	-761485.83	-816.73	-81.95	2986903.23	-69813.22	-2116.19	2.50402	2.398058	1.63543
3.1x disp (elasto-plastic)	-780298.5	-846.24	-86.86	3046873.84	-70049.61	-2239.04	2.565883	2.446206	1.640967

--Because Fy, Fz and Mz are almost zero (very small), they are not included when calculating the load ratio.
--When applying 1x displacement to the elasto-plastic model, the load is almost same (within 5%) as the elastic model.
--when applying 1.1x displacement, the load is ~1.07x.
--when applying 3.1x displacement, the load is ~2x (2.56x Fx, 2.45x Mx, 1.64x My).

Figure 18: Comparison of total reaction forces (in cylindrical system)

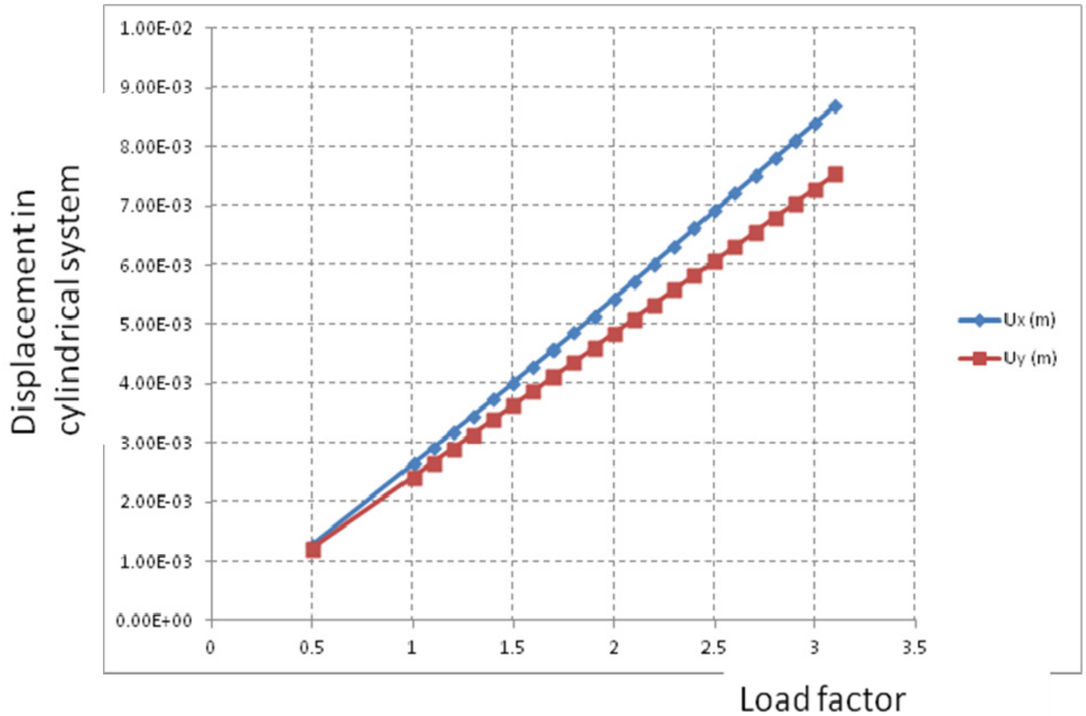

Figure 19: Displacement of the partial model.

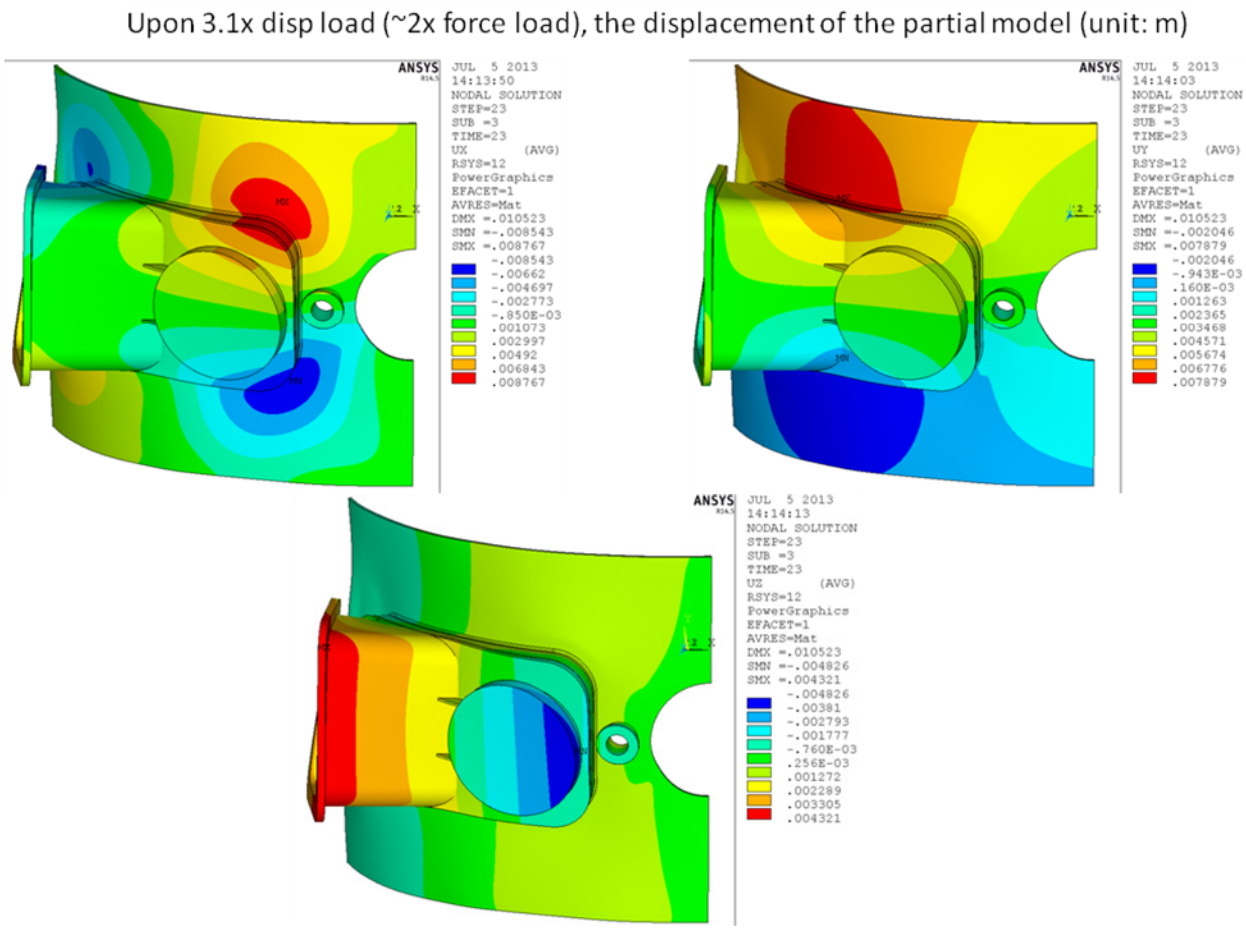

Figure 20: Upon 3.1x disp load (~2x force load), the displacement of the partial model (unit: m)

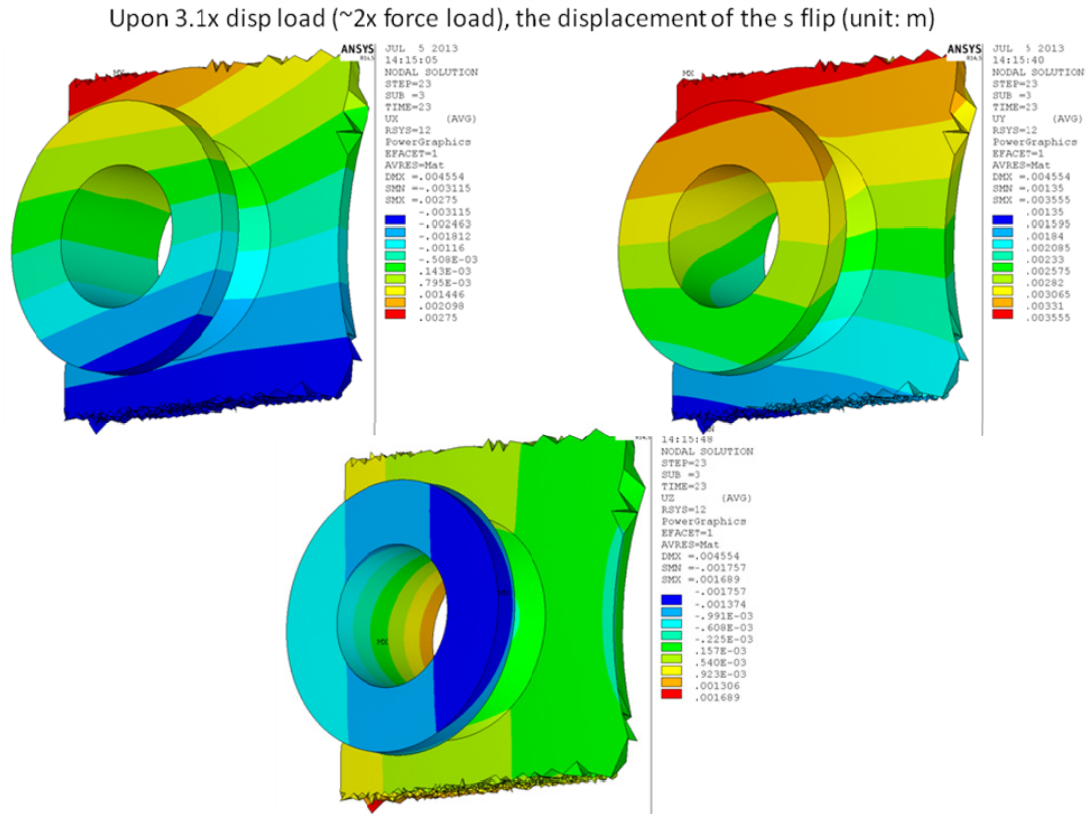

Figure 21: Upon 3.1x disp load (~2x force load), the displacement of the s flip (unit: m)

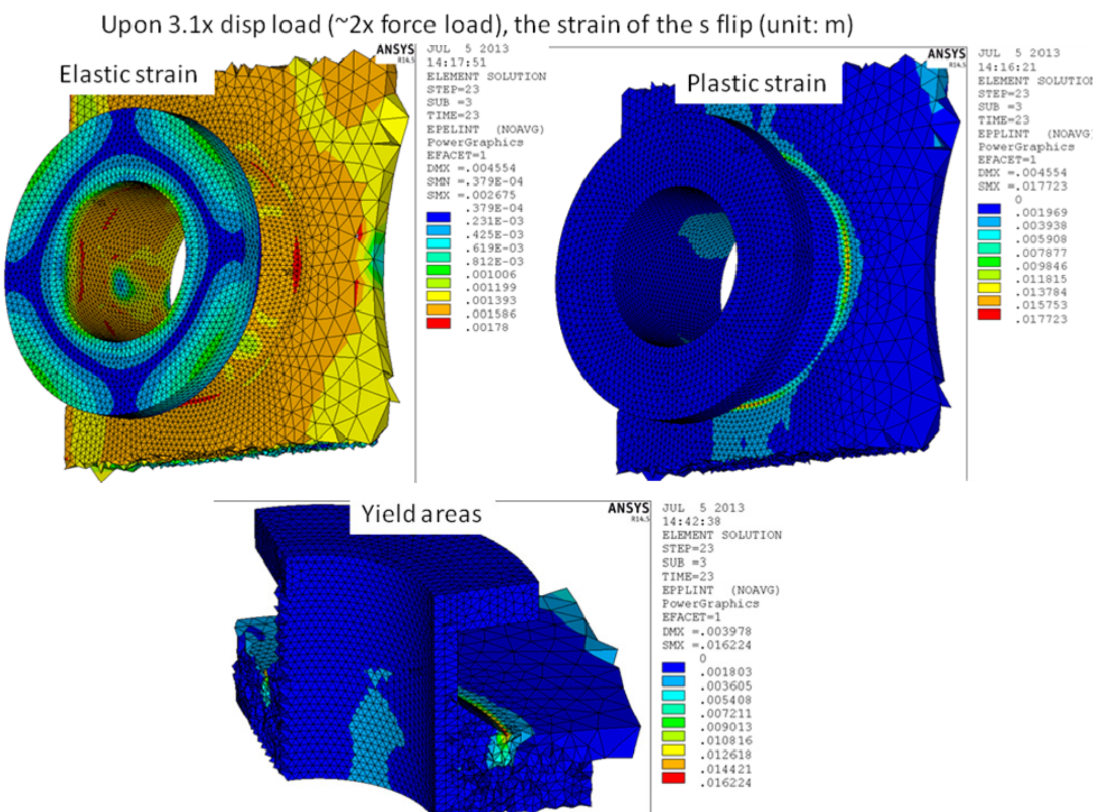

Figure 22: Upon 3.1x disp load (~2x force load), the strain of the s flip (unit: m)

Appendix 1: Influence of vessel deformation to diagnostics

Stress limits are not the only requirement for diagnostic penetrations. Deformation of the vessel may effect alignment of diagnostics, and certain types of diagnostics, especially those that are laser based may point at the wrong point in the plasma, first wall or optical targets, this calculation is added as an appendix to document the magnitude of the possible deformations and optical mis-alignments.

Since the ports and vessel deform upon the load from TF outer legs, the deformation may affect the diagnostic signal and thus we did this simulation. Two diagnostic signals are selected, which are bay J to centerstack and bay G to B. Beam elems are added to the model to see the tip deflection which is similar to diagnostic light deflection. Figure 23 and 24 shows the model and locations to add the beam elements (i.e. optical "rays"). Maximal tip deflection is from bay J to CS: about 7.8mm (0.31") (figure 25). The other one, from bay G to B, is very small, less than 1mm. Maximal deflection with equilibrium 79 loads is 7.8mm (0.31"). Considering other scenarios with different deflections and different orientations, it is better that the sensor size to be bigger than 16mm (0.62")

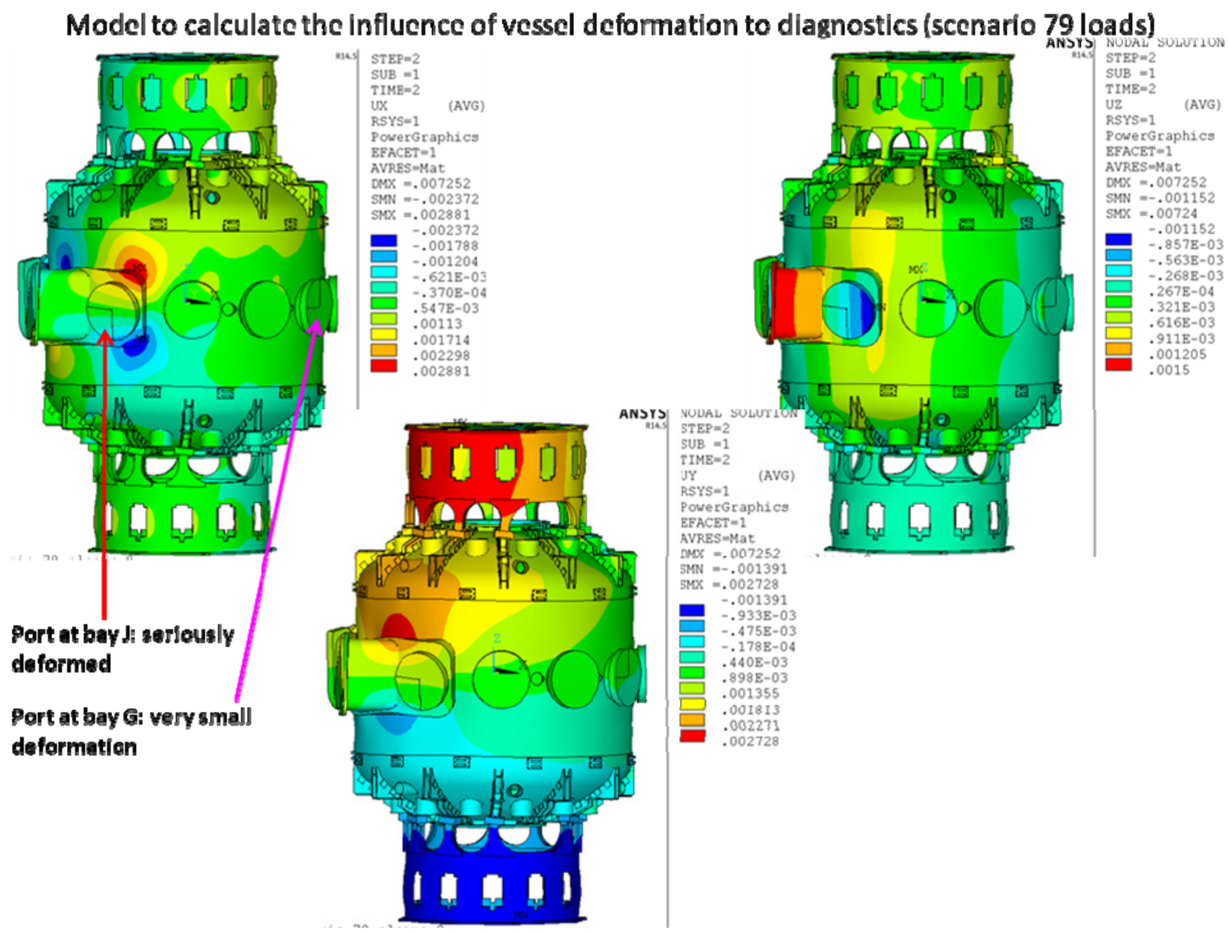

Figure 23: model to calculate the influence of vessel deformation to diagnostics.

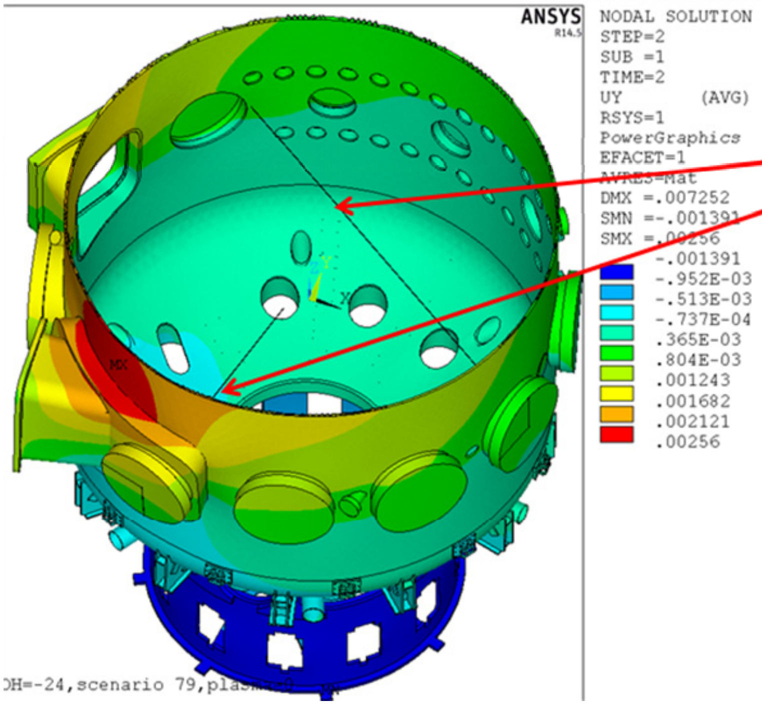

Figure 24: model to show the beam elems added.

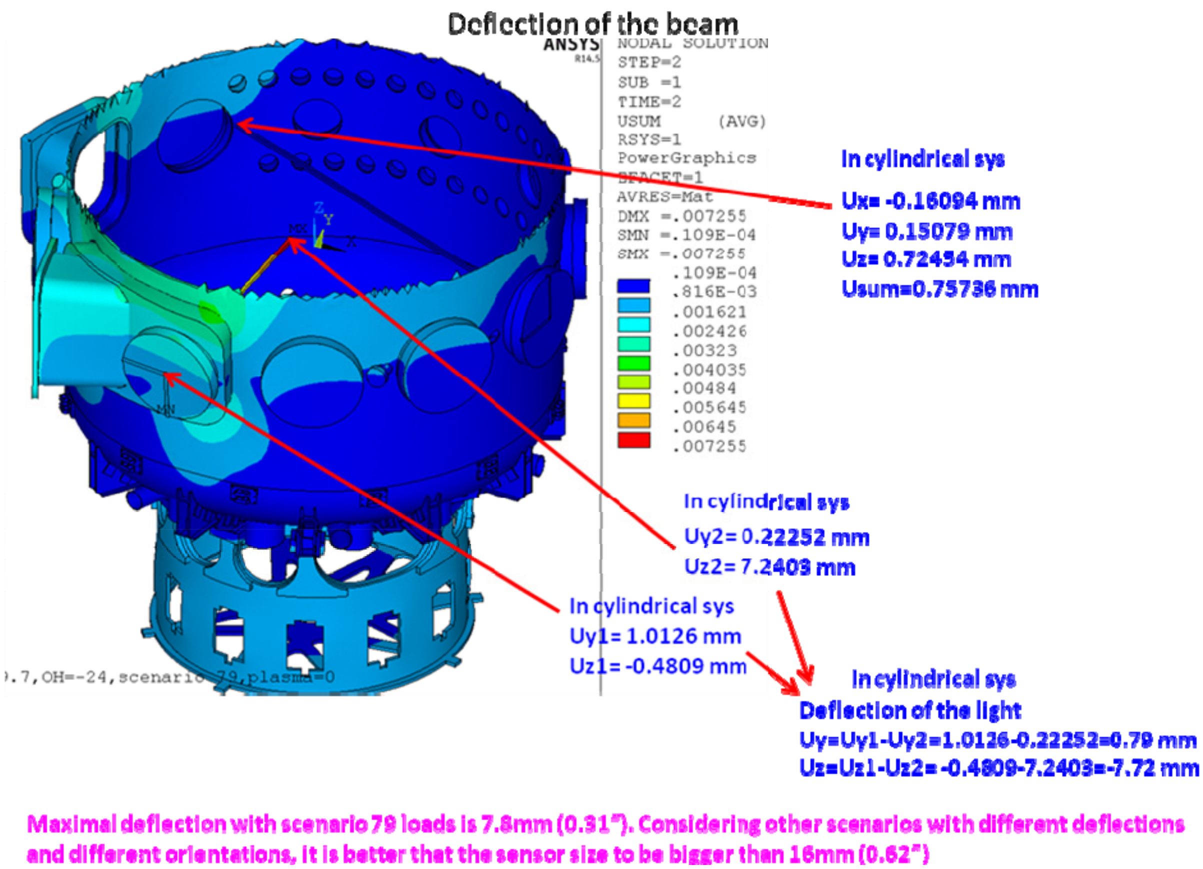

Figure 25: deflection of the beam.