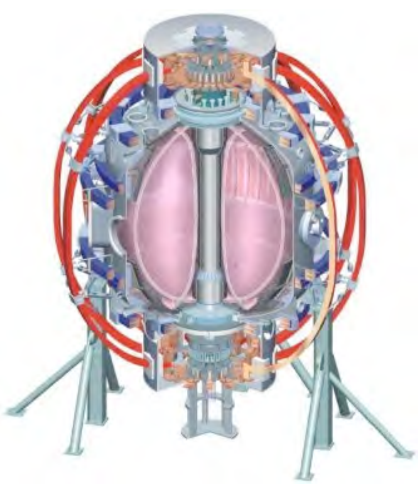
Magneto-Structural Analysis on Vacuum Vessel

Srinivas Avasarala

College W&M
Colorado Sch Mines
Columbia U
CompX
General Atomics
INEL
Johns Hopkins U
LANL
LLNL
Lodestar
MIT
Nova Photonics
New York U
Old Dominion U
ORNL
PPPL
PSI
Princeton U
Purdue U
SNL
Think Tank, Inc.
UC Davis
UC Irvine
UCLA
UCSD
U Colorado
U Illinois
U Maryland
U Rochester
U Washington
U Wisconsin

NSTX Centerstack Upgrade Peer Review
LSB, B318
August 13, 2009

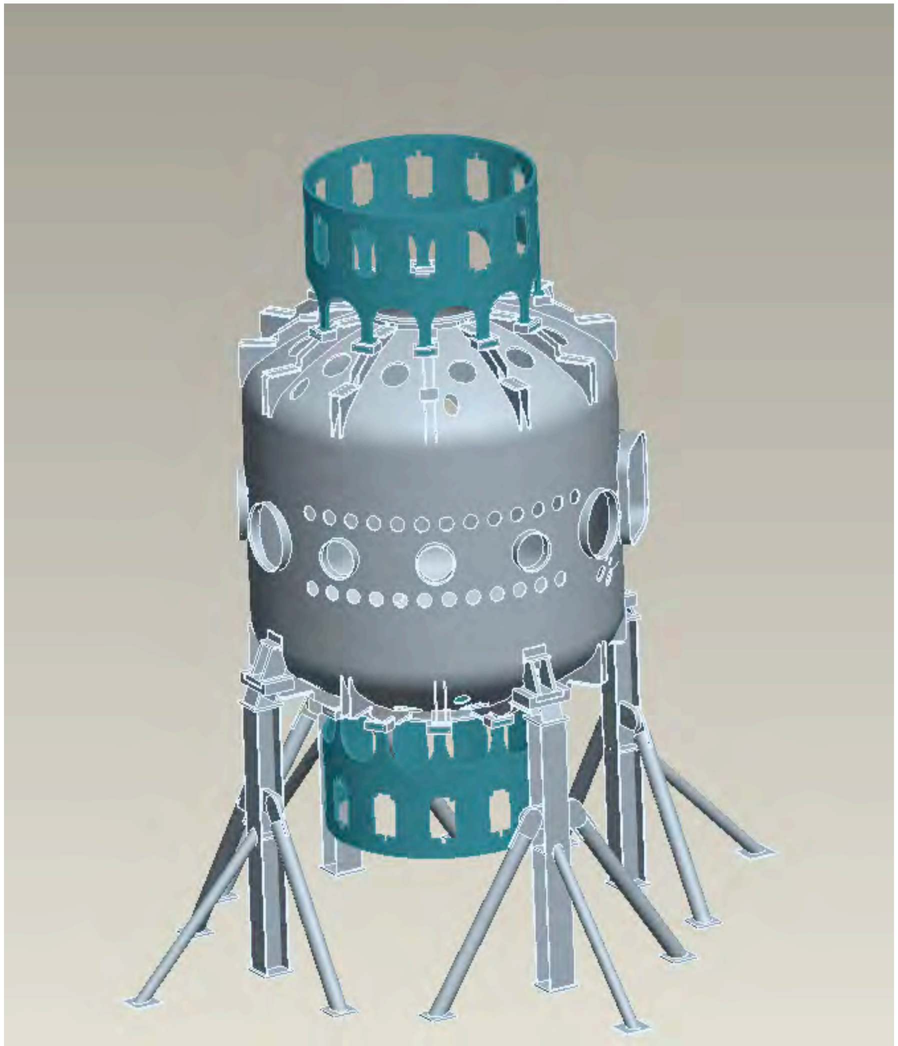


Culham Sci Ctr
U St. Andrews
York U
Chubu U
Fukui U
Hiroshima U
Hyogo U
Kyoto U
Kyushu U
Kyushu Tokai U
NIFS
Niigata U
U Tokyo
JAEA
Hebrew U
Ioffe Inst
RRC Kurchatov Inst
TRINITI
KBSI
KAIST
POSTECH
ASIPP
ENEA, Frascati
CEA, Cadarache
IPP, Jülich
IPP, Garching
ASCR, Czech Rep
U Quebec

SYNOPSIS

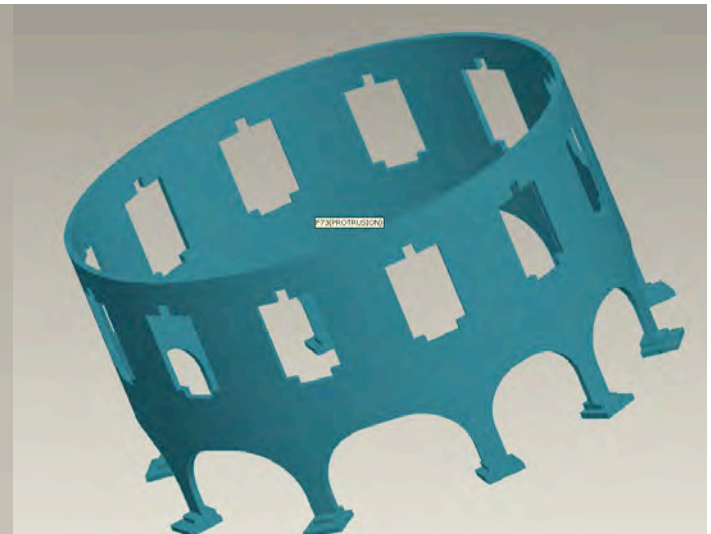
- Modeled the solid in Pro E. All bodies are merged to the vessel, except for the umbrella structure.
- Imported the model into workbench and meshed with 8 node brick elements.
- Carried the mesh into ANSYS-Classic and converted the element type to SOLID 97 (has Vector Potential degree of freedom). Encountered problems with DOF compatibility.
- Retained the node numbers and element connectivity and rebuilt the model in ANSYS-Classic.
- Applied the Vector Potentials from 2-D OPERA solution and obtained the eddy currents and Lorentz forces.
- Ran a static and dynamic stress pass and obtained the stresses, deformations and the dynamic load factors.
- Built the Passive Plates, constrained them to the vessel and repeated the above procedure.
- Developed macros which, with modest changes, yield stresses and deformation on other parts of the machine.

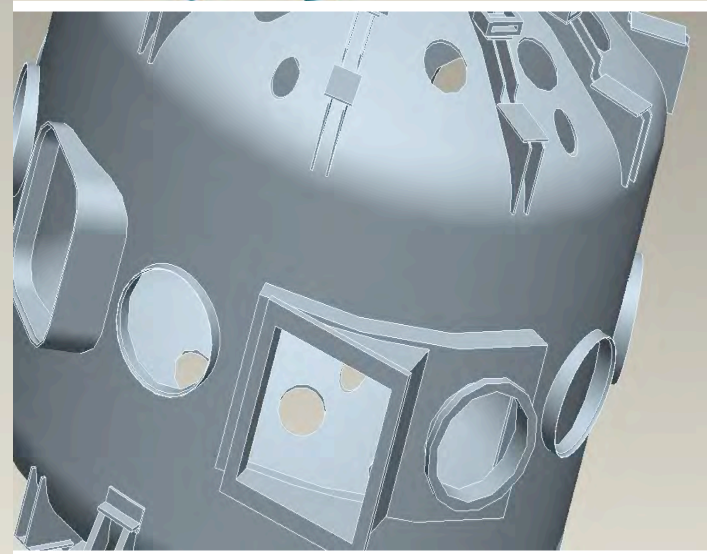
Modeling



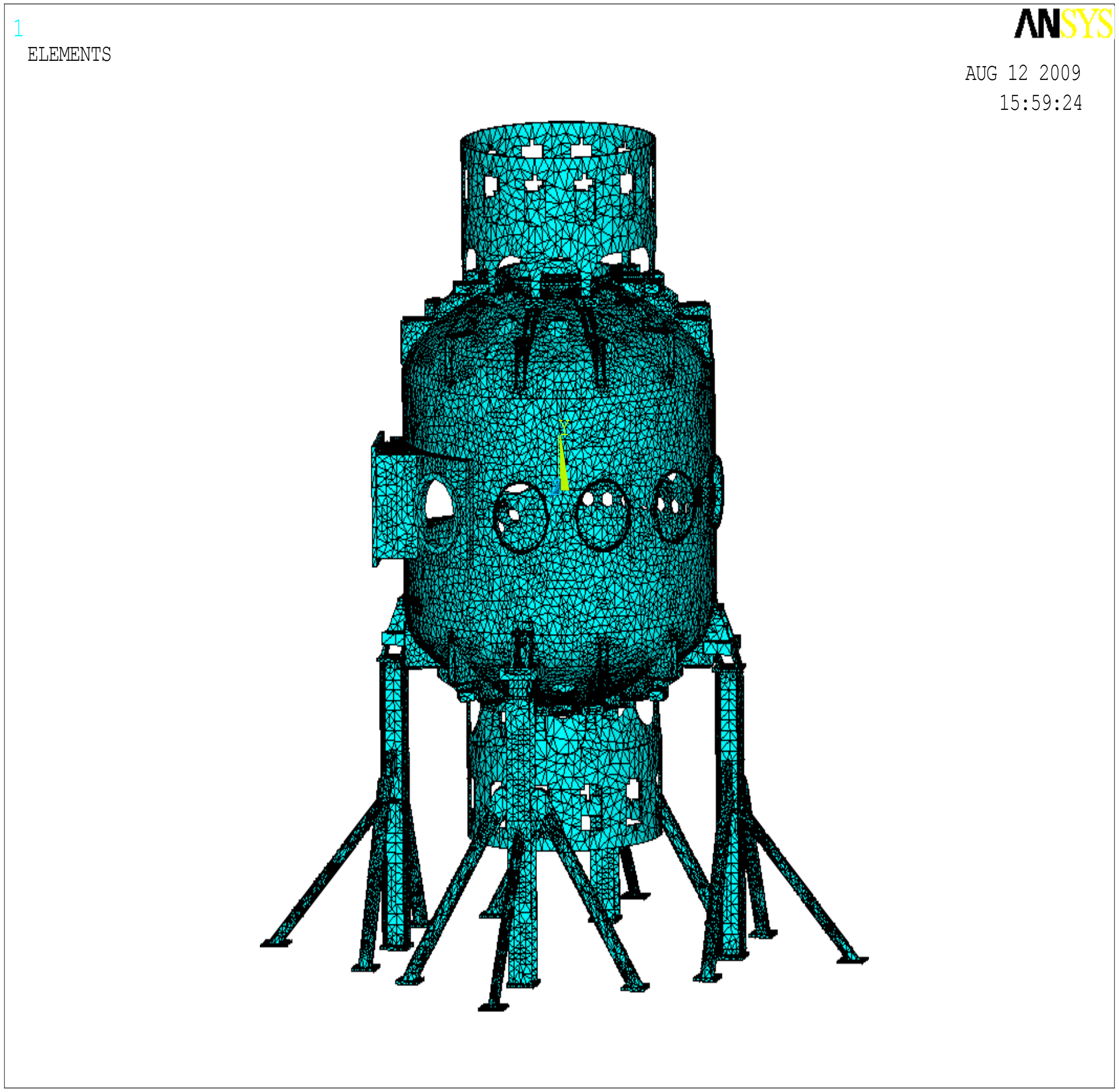
- All the components in the Pro-E model are merged for ease in FEA.
- The final model has two components: The Umbrella structure and the rest of the vessel with port extensions and the legs merged.

Solid Model





Finite Element Model

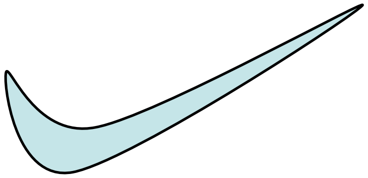


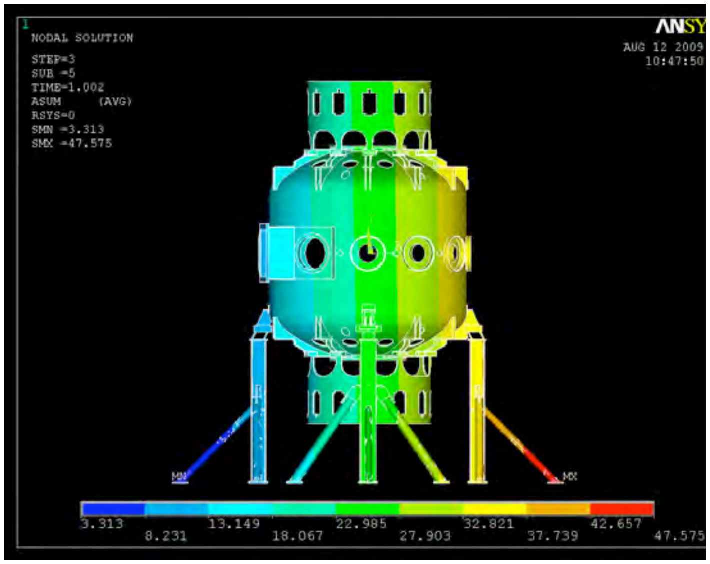
76436 Nodes

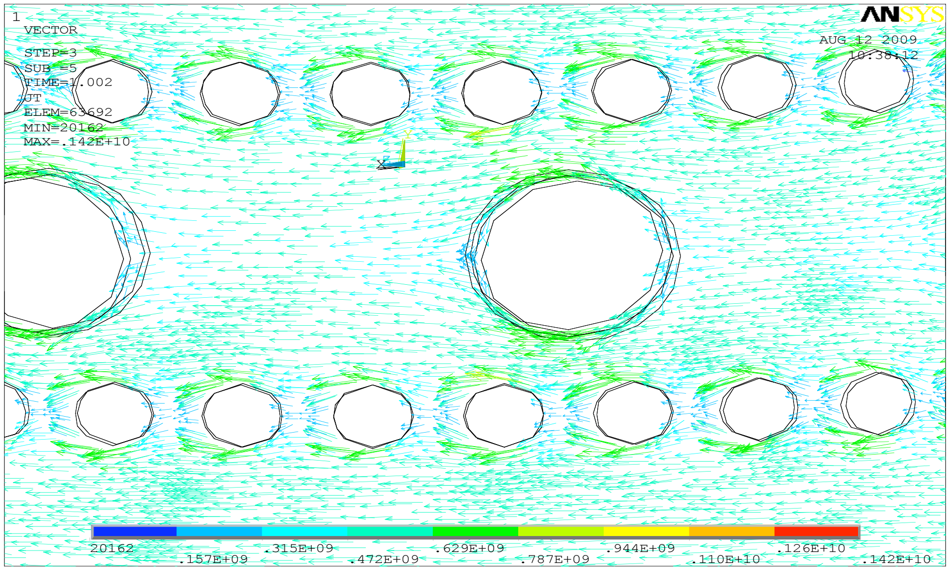
216112 8-Node Brick Elements

Testing the FE model

- Before going further, the model is tested to see if the procedure would work.
- A Vector Potential gradient is applied on the model.
- Solved for eddy currents and Lorentz forces.
- Solved for stresses.







OPERA RESULTS

- The Vector Potential data obtained from OPERA is a 6561 column.
- The results are written as a function of the radius of the vessel and the height.
- These results are rearranged into a 81x 81 matrix, for ANSYS to read, using MATHCAD.

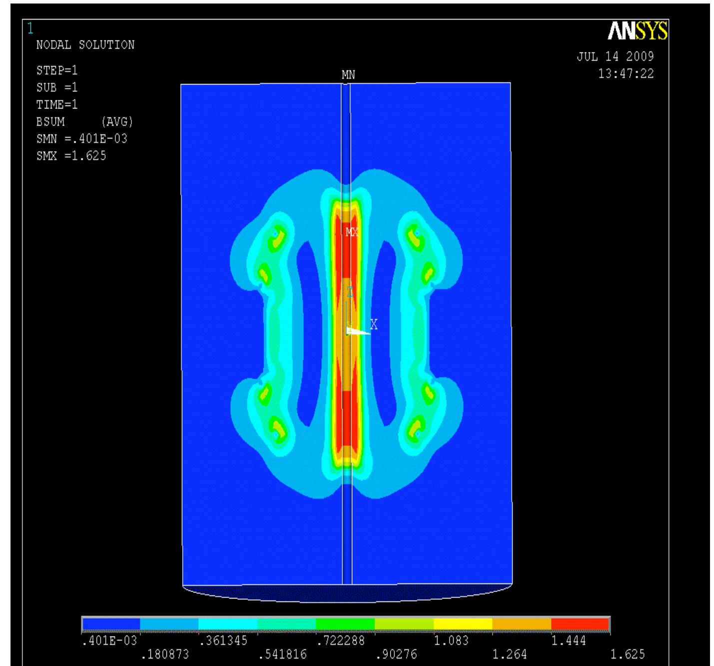
Reading Vector Potentials from OPERA

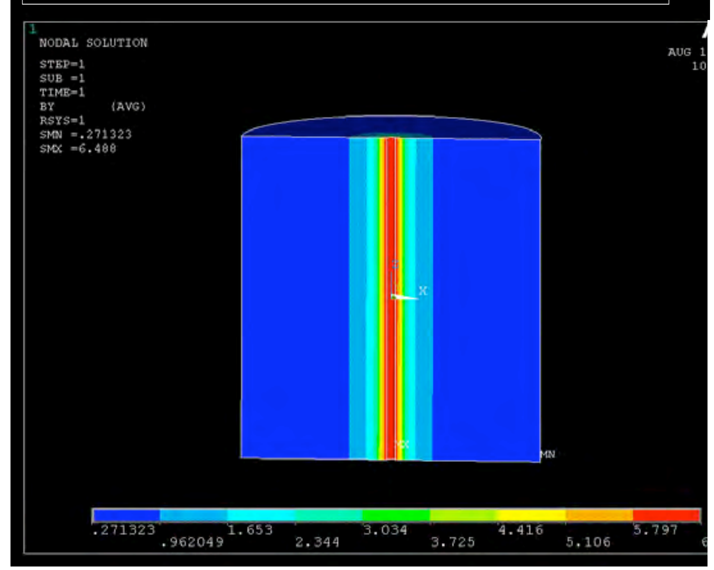
- The 2D Vector Potential solution is then imposed on a hollow cylinder to verify the procedure.
- A toroidal field was also applied on the cylinder

```
*dim,vect,table,81,81,1,x,z,,1
*tread,vect,'case4','txt'
nall
*get,nmax,node,,num,max
*do,i,1,nmax
z=nz(i)
x=nx(i)
d,i,ay,vect(x,z)
*enddo
BR=130000*12*3*2e-7
*get,nmax,node,,num,max
*do,i,1,nmax
csys,1
rad=nx(i)
d,i,az,-.5*BR*log(rad*rad)
*enddo
```

B = curl **A** , **A** is Magnetic Vector Potential (MVP)

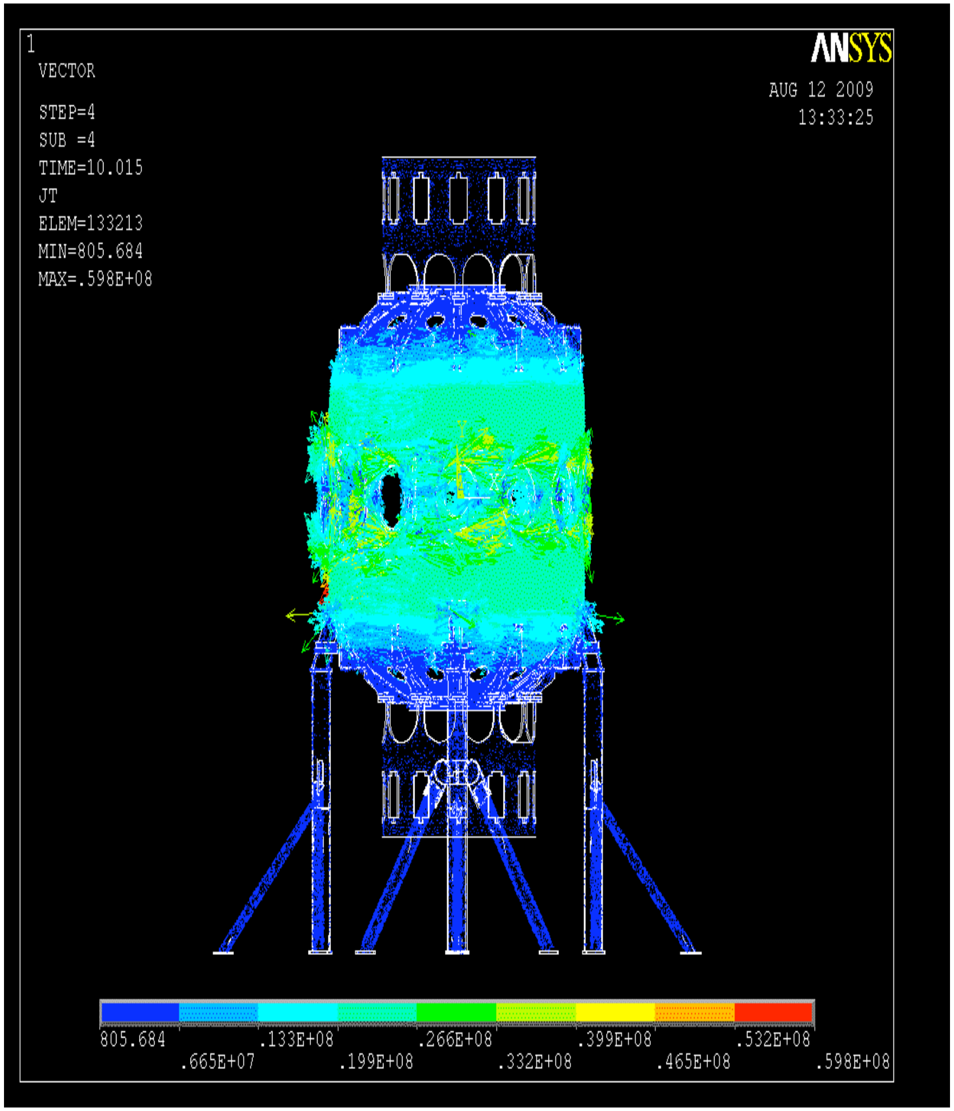
B = (dAz/dy - dAy/dz)**i** + (dAx/dz - dAz/dx)**j** + (dAy/dx - dAx/dy)**k**

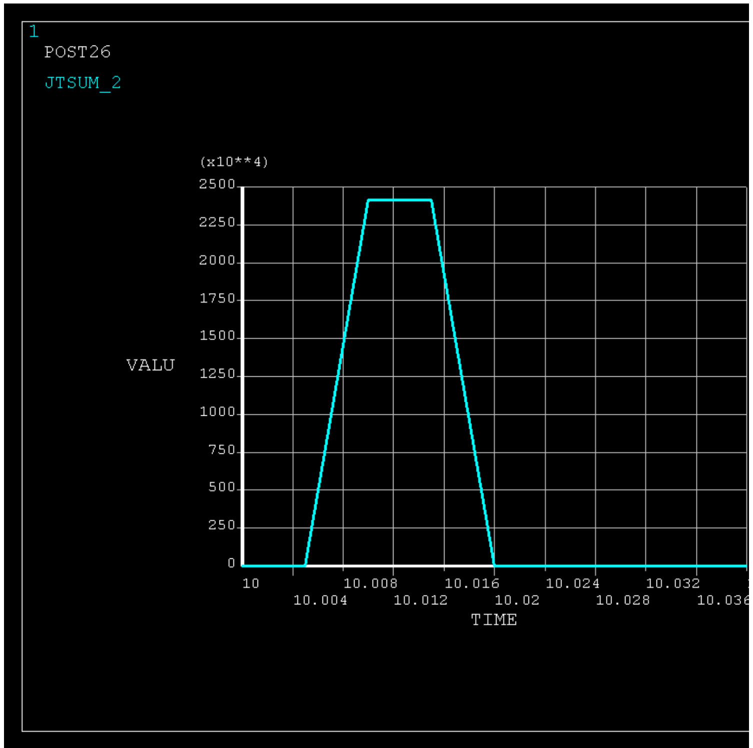




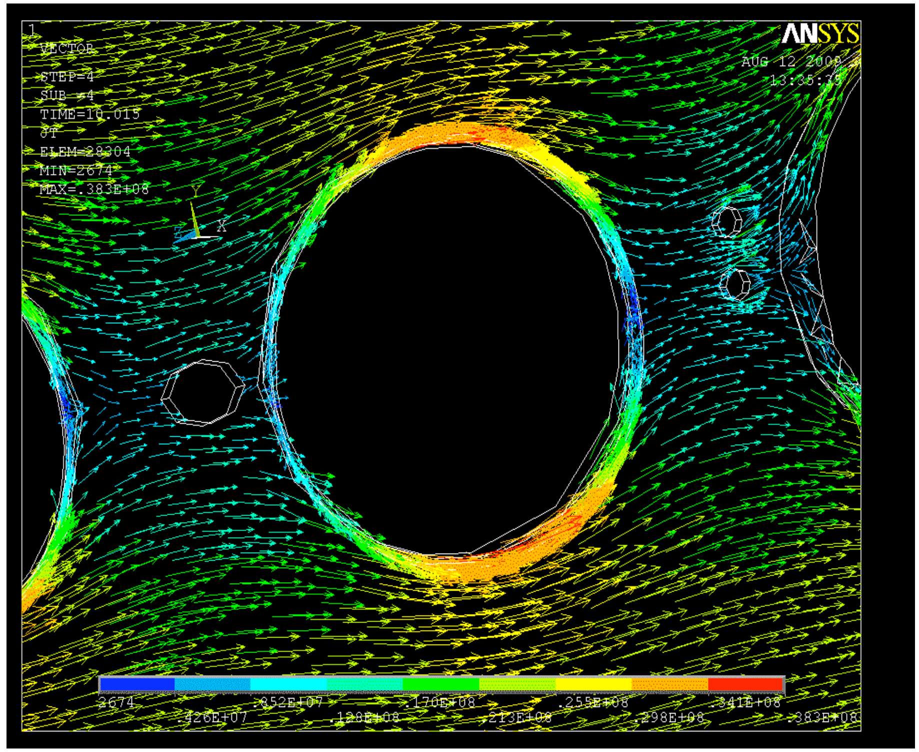
Eddy currents and Lorentz forces on the vessel

- About 9 time points that contain the disruption are obtained and read into ANSYS as parametric tables (using these tables, the Vector Potentials can be interpolated based on the nodal coordinates)
- 9 load steps are written and solved.

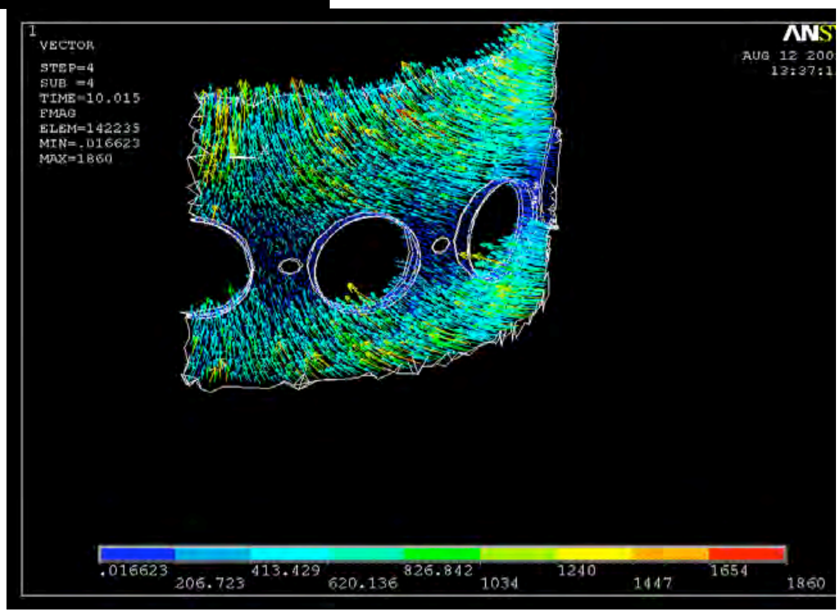




1
POST26
JTSUM_2
(x10**4)
2500
2250
2000
1750
1500
1250
1000
750
500
250
0
VALU
10
10.004
10.008
10.012
10.016
10.02
10.024
10.028
10.032
10.036
TIME

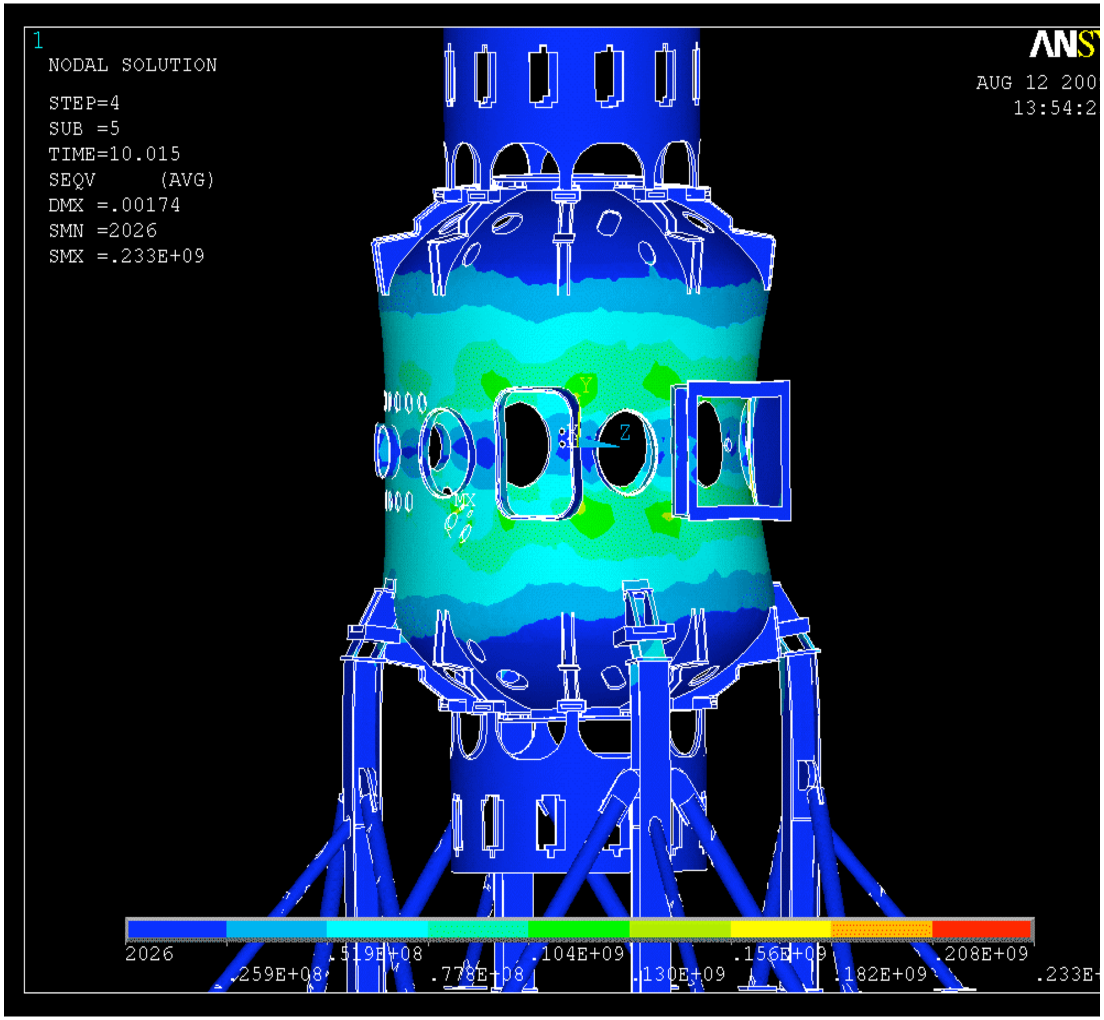


ANSYS
VECTOR
STEP=4
AUG 12 2009
TIME=10.015
MIN=2674
MAX=.383E+08
2674
.426E+07
.852E+07
.128E+08
.170E+08
.213E+08
.255E+08
.298E+08
.341E+08
.383E+08



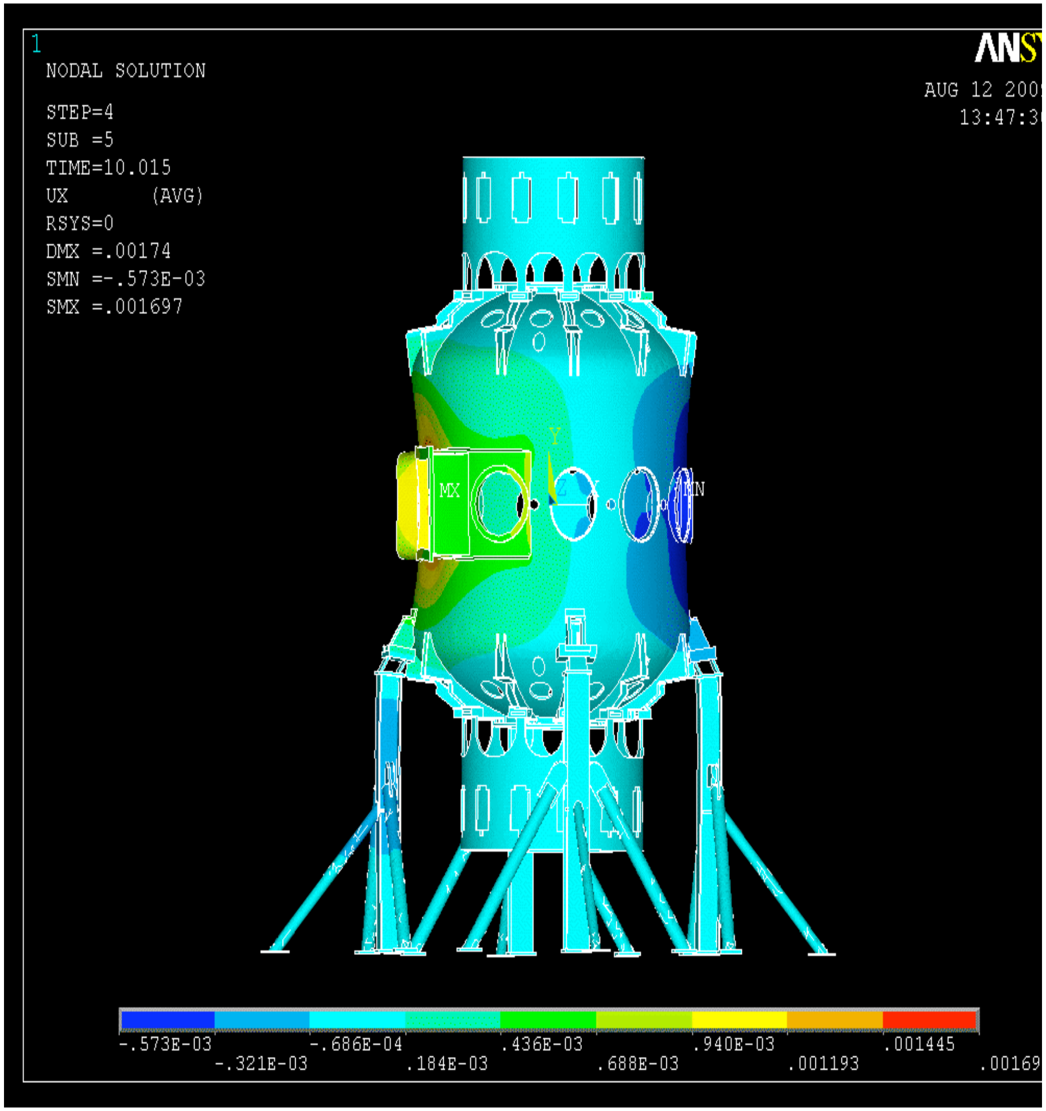
1
VECTOR
STEP=4
SUB =4
TIME=10.015
FMAG
ELEM=142235
MIN=.016623
MAX=1860
.016623
206.723
413.429
620.136
826.842
1034
1240
1447
1654
1860

STATIC STRUCTURAL ANALYSIS



Max
Stress
=233 Mpa

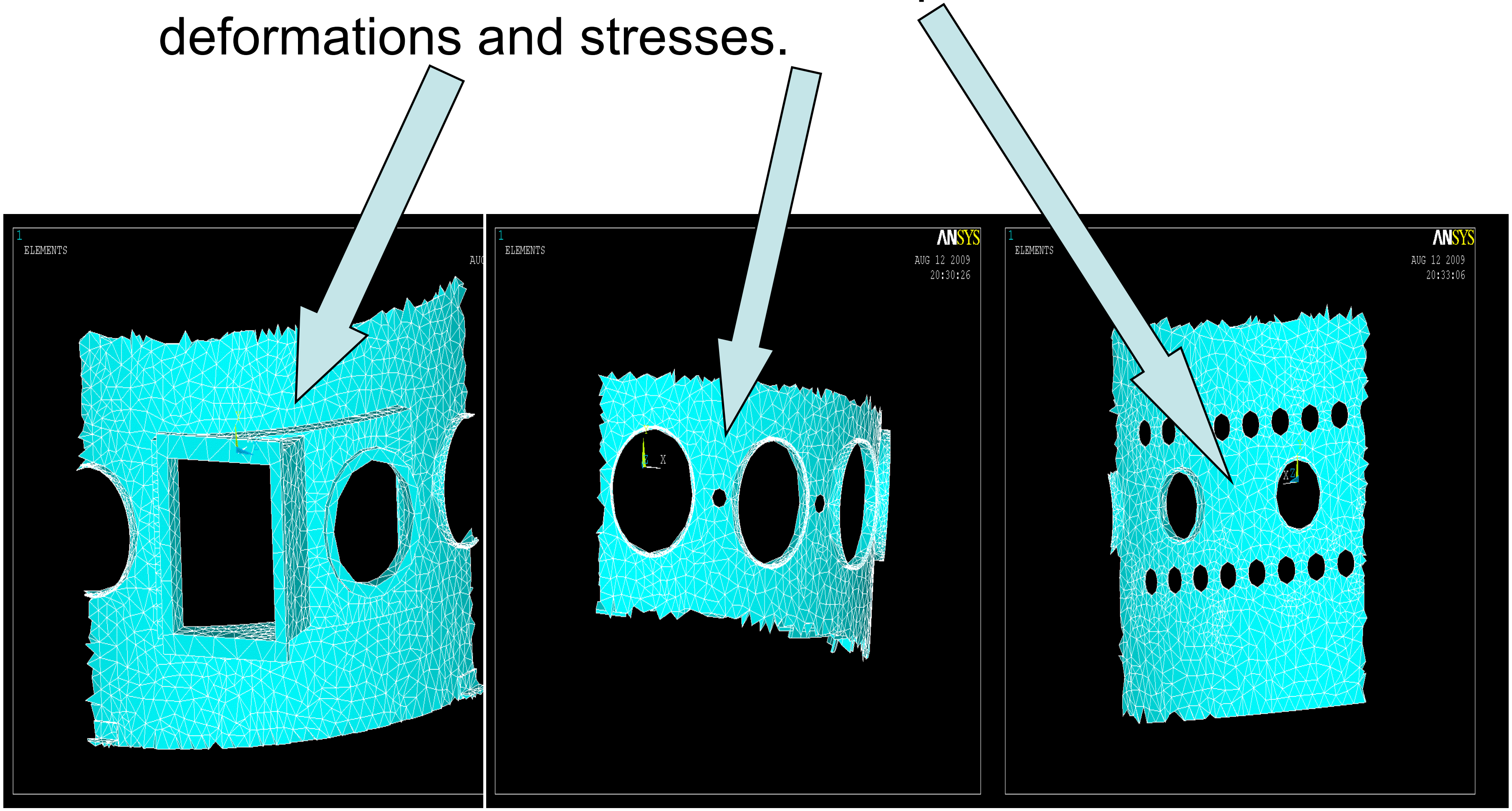
STATIC STRUCTURAL ANALYSIS



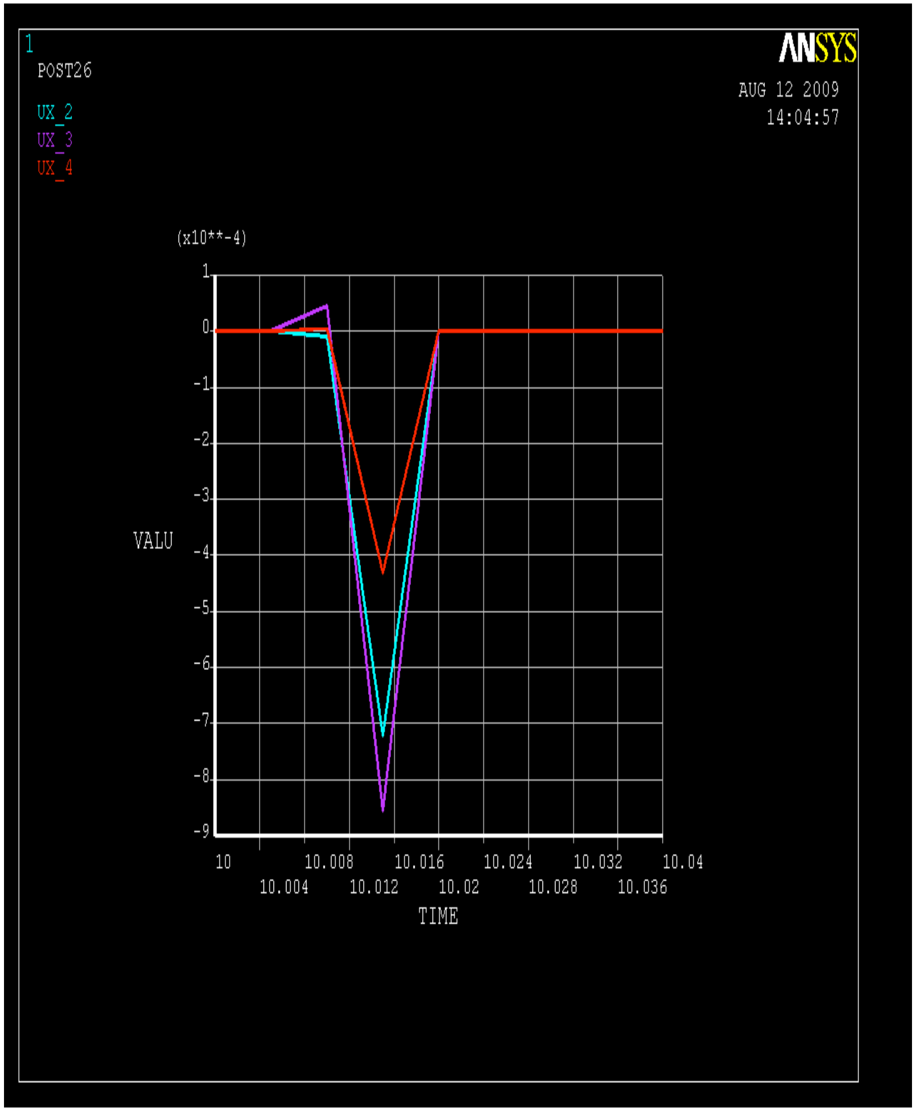
Maximum Deformation =1.697 mm

Nodal Deformations and Stresses for Static Analysis

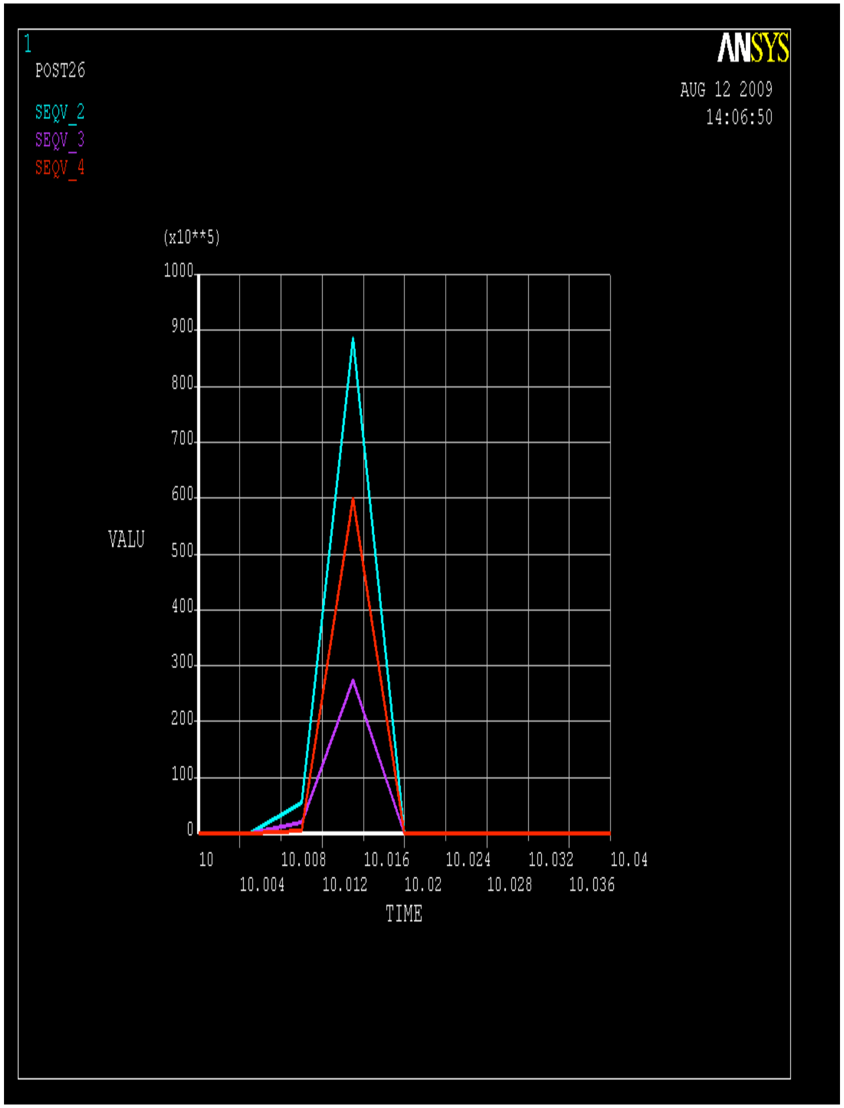
- Picked 3 nodes on the vessel to plot deformations and stresses.



Deformations and Stresses

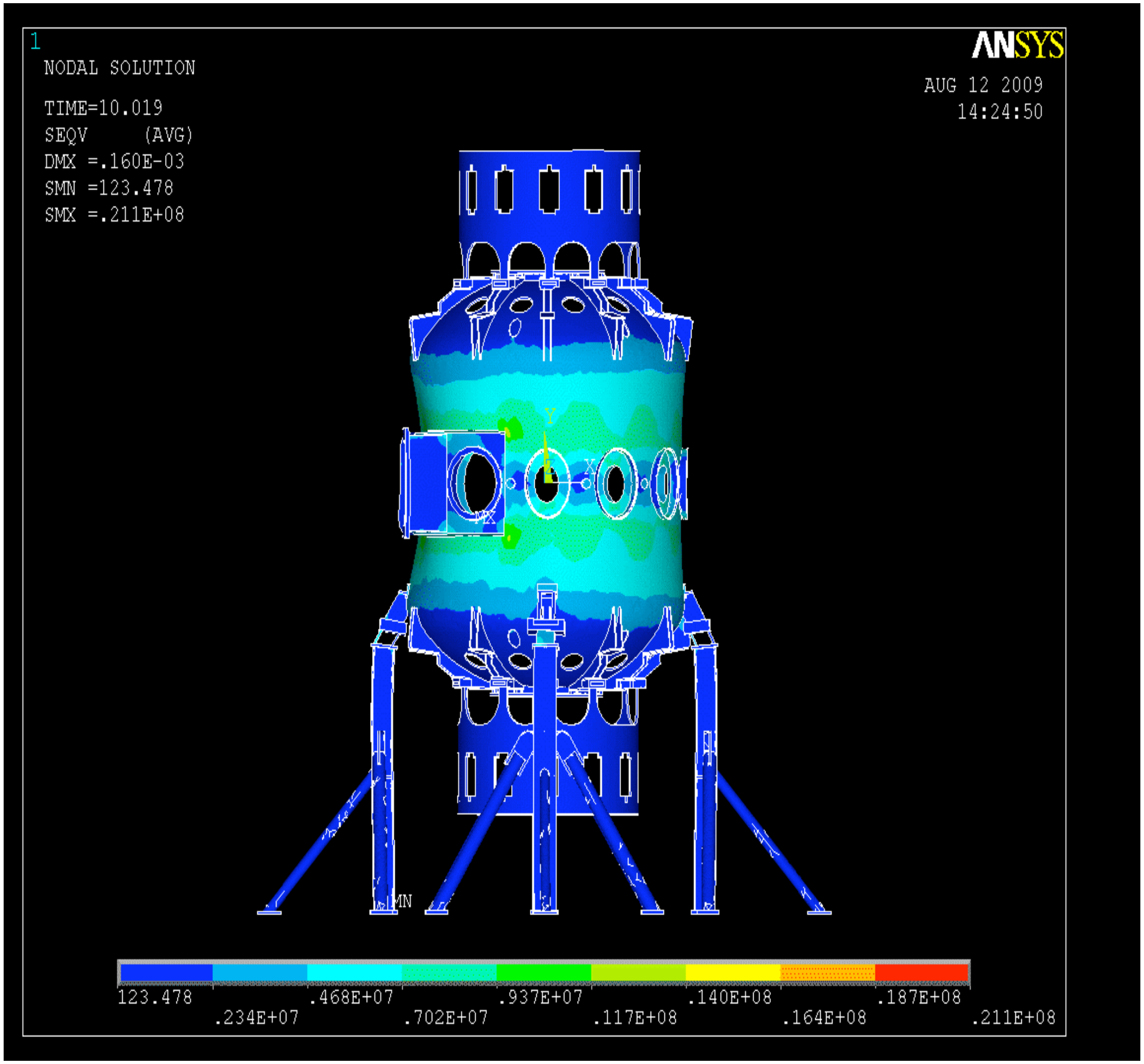


0.42,0.72 & 0.85mm



28,60 &89 Mpa

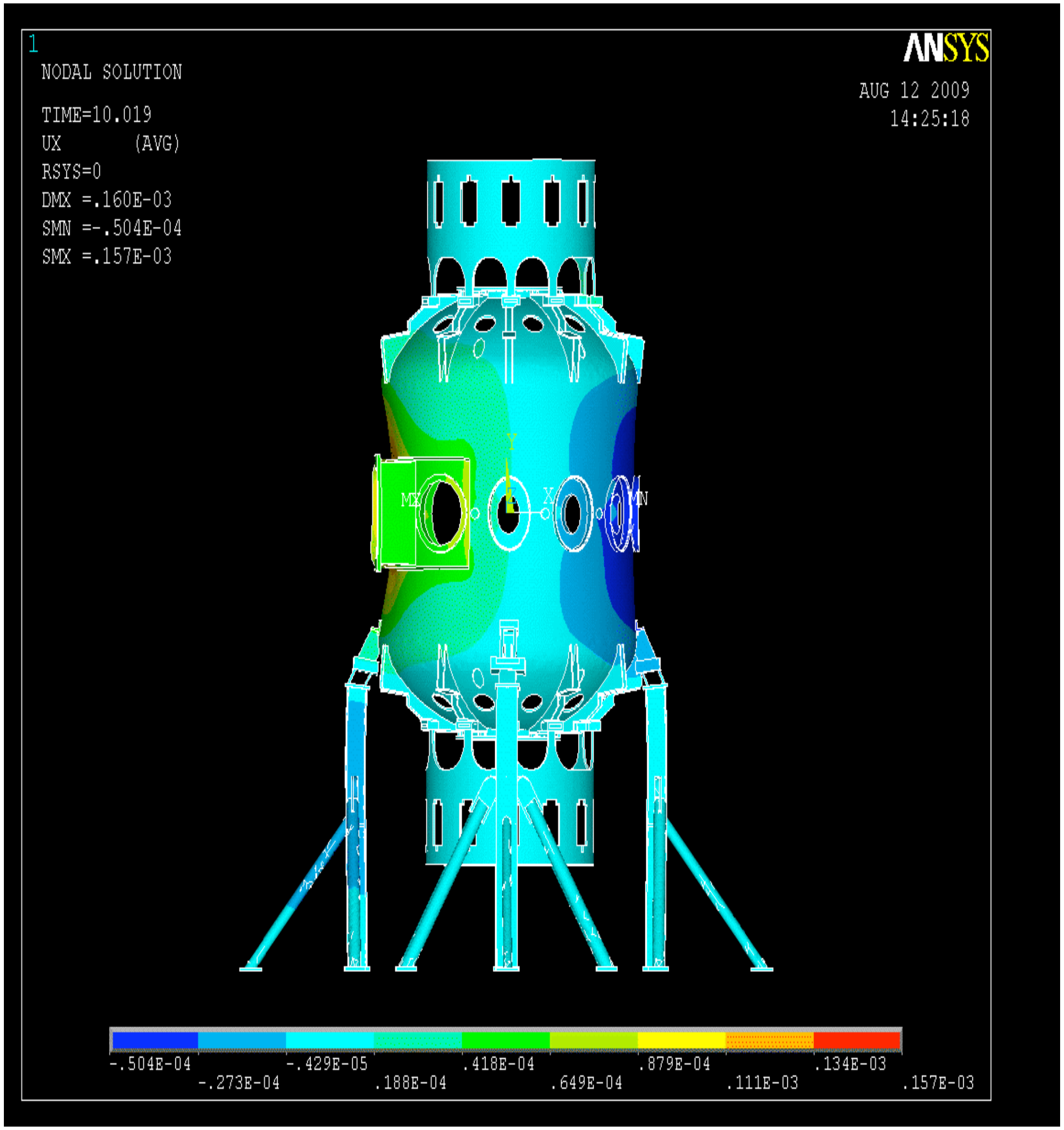
DYNAMIC STRUCTURAL ANALYSIS



Max Stress =21 Mpa

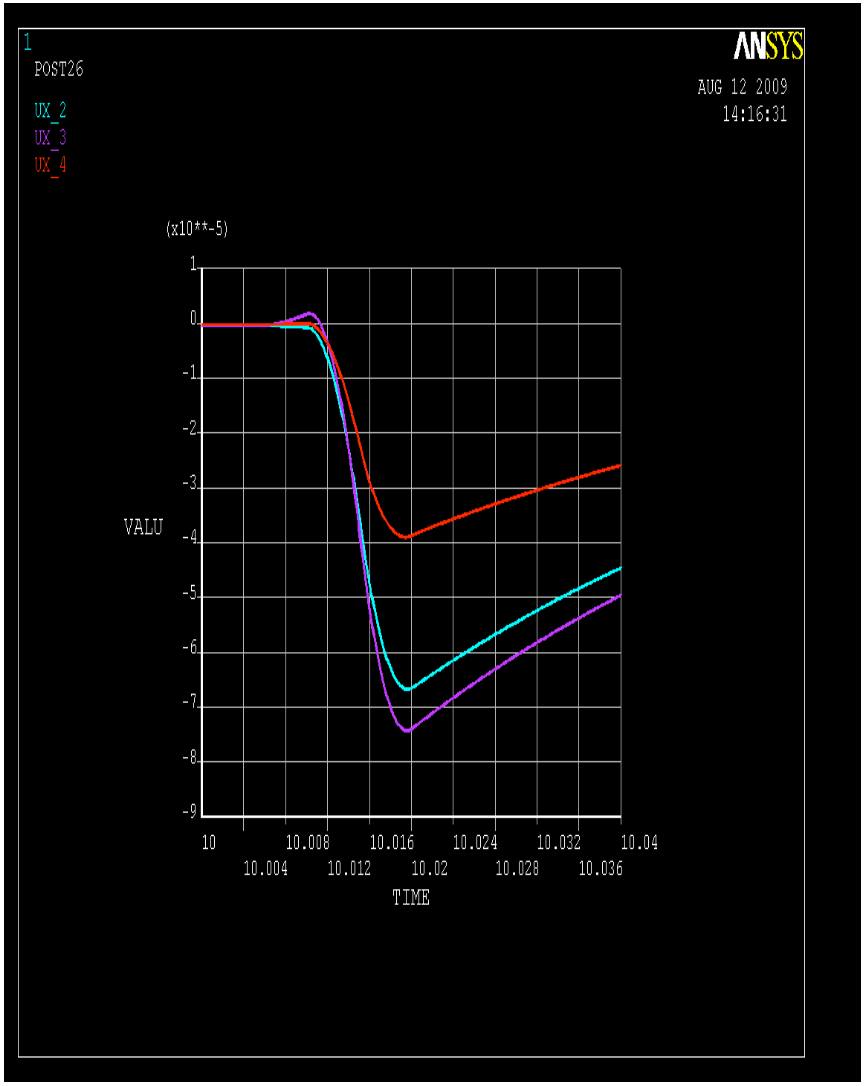
Damping = 5 %

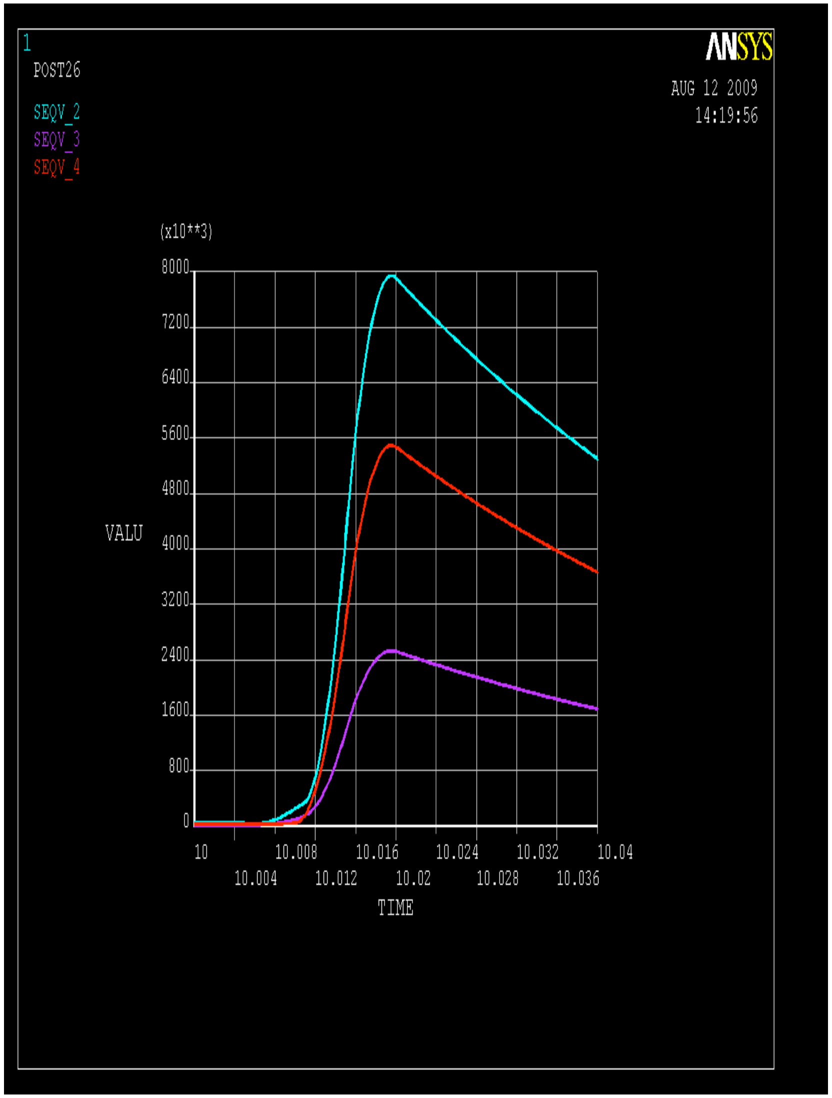
DYNAMIC STRUCTURAL ANALYSIS



Maximum Deformation =.157 mm

Deformations and Stresses



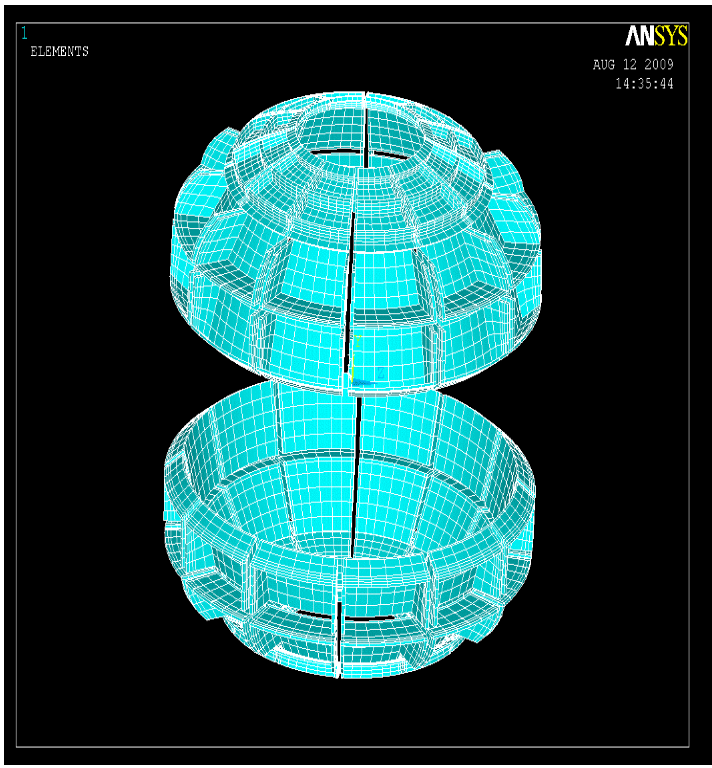


0.039,0.065 & 0.075mm

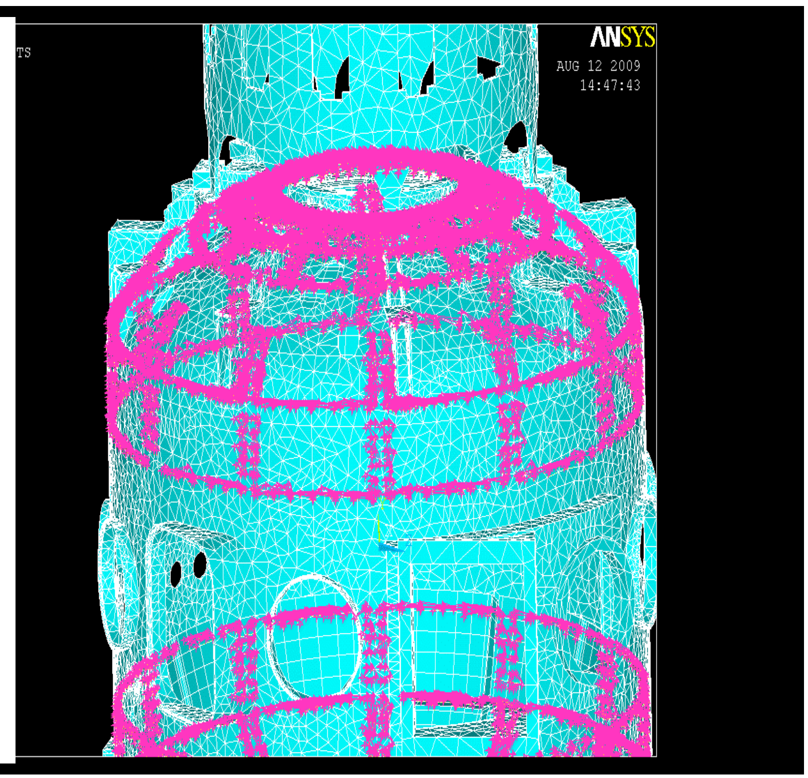
28,60 &89 Mpa

Adding Passive Plates

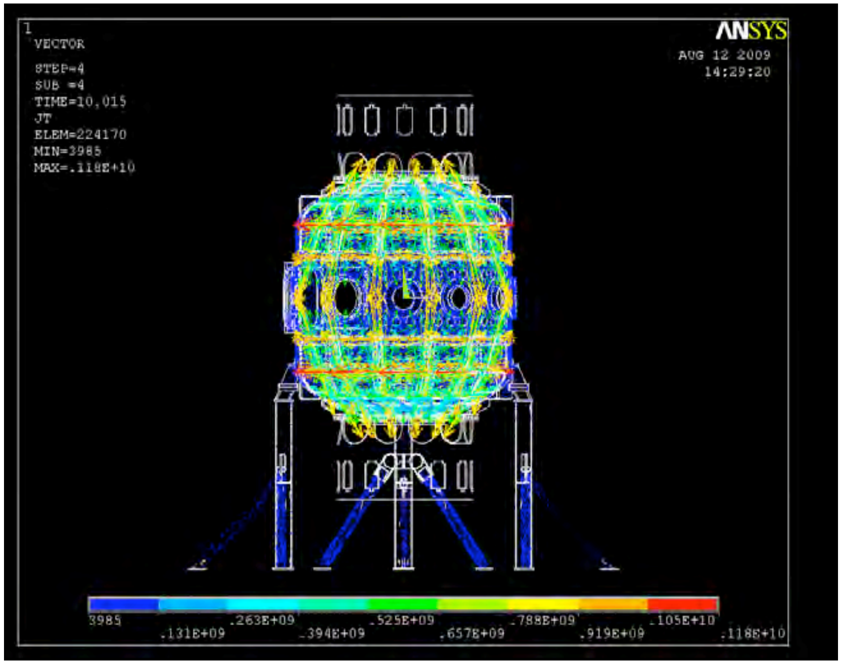
- CAD model of the Passive Plates is not available.
- An approximate model of the Passive Plates is built from the existing Shell element model of the machine and the ones used in 2-D OPERA simulations.
- These plates were glued to the vessel using the CEINTF command

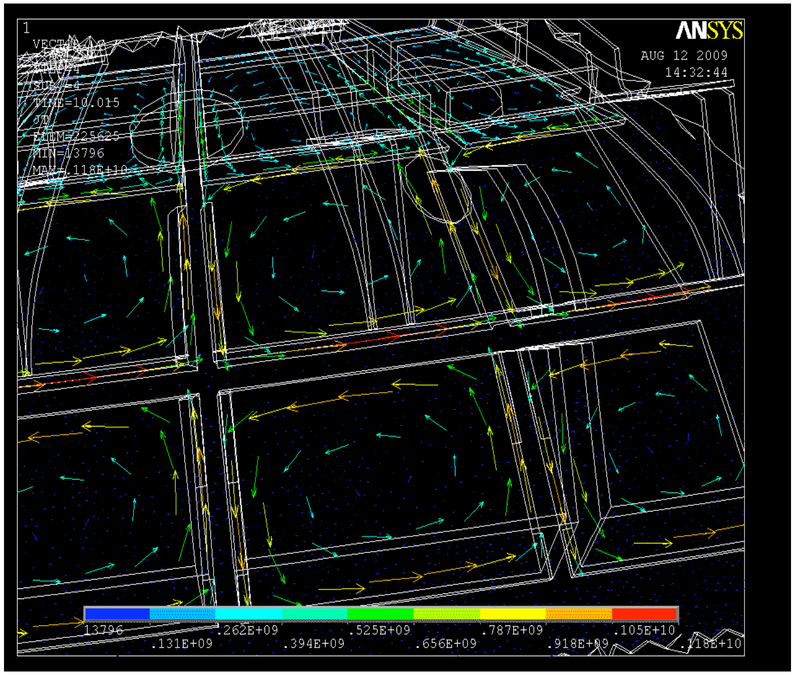


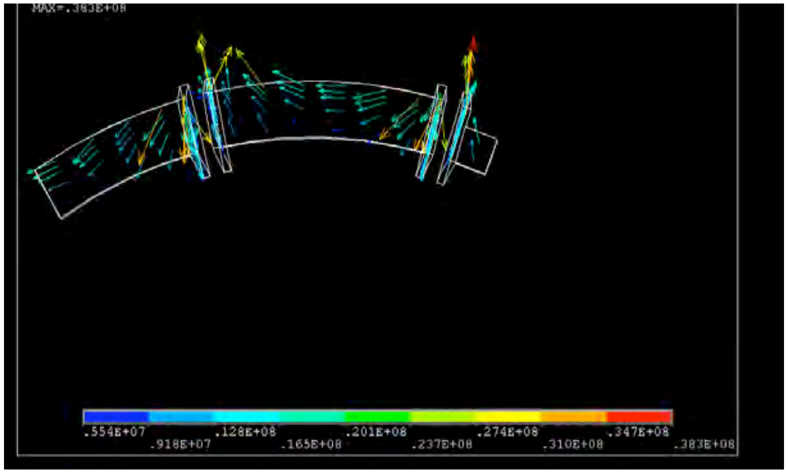

```
esel,s,type,,5
nsel,s,loc,z,.6504,1.0084
nsel,a,loc,z,-.6504,-1.0084
nsle,r,
nsel,r,loc,x,0,1.6892
esel,s,real,,12
ceintf,3,all,3

allsel,all

esel,s,type,,5
nsel,s,loc,z,1.0742,1.3676
nsel,a,loc,z,-1.0742,-1.3676
nsle,r,
nsel,r,loc,x,0,1.7491
esel,s,real,,11
ceintf,3,all,3

allsel,all

esel,s,type,,5
nsel,s,loc,z,1.7276,1.4998
nsel,a,loc,z,-1.7276,-1.4998
nsle,r,
nsel,r,loc,x,0,1.2377
esel,s,real,,10
ceintf,3,all,3

allsel,all
```



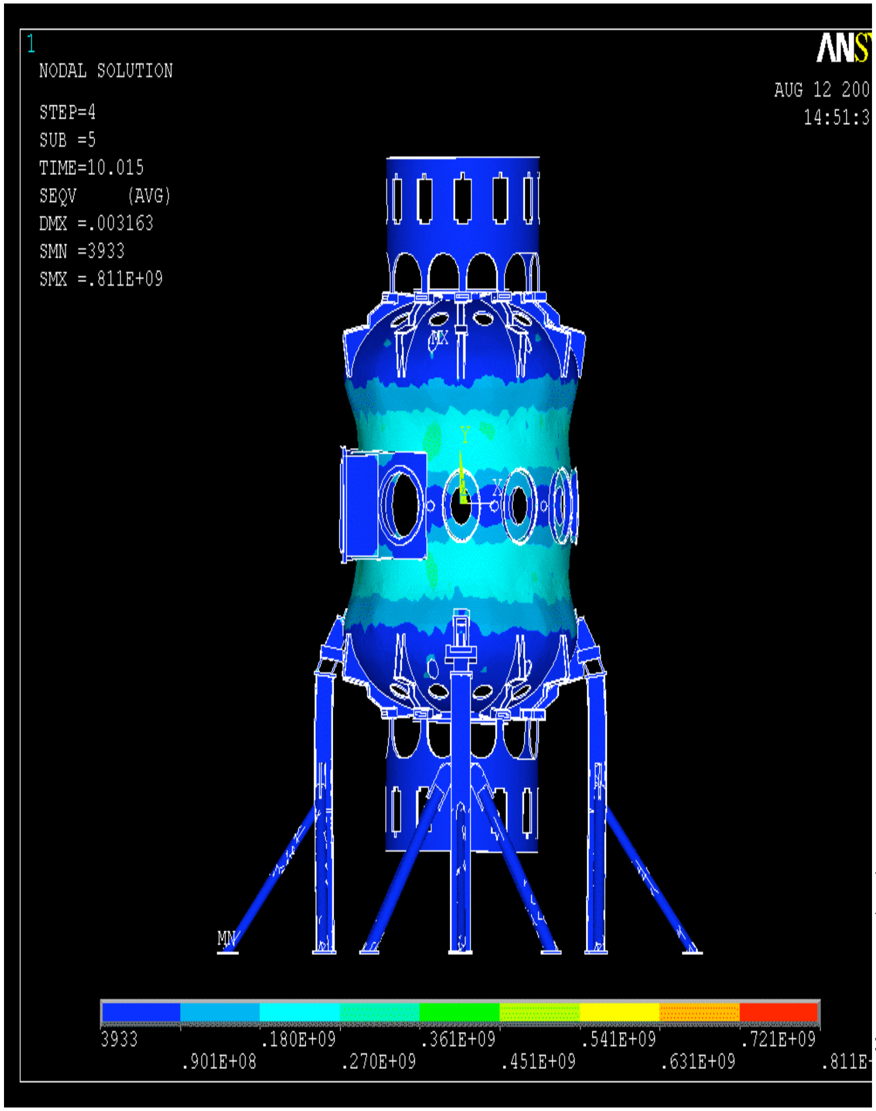
Currents with the Passive Plates



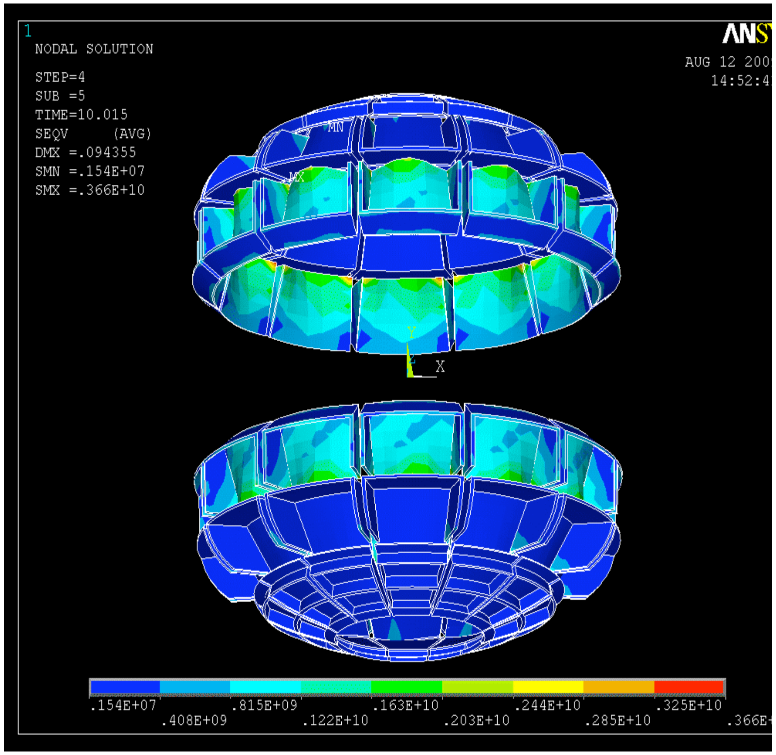




STATIC STRUCTURAL ANALYSIS
with Passive Plates

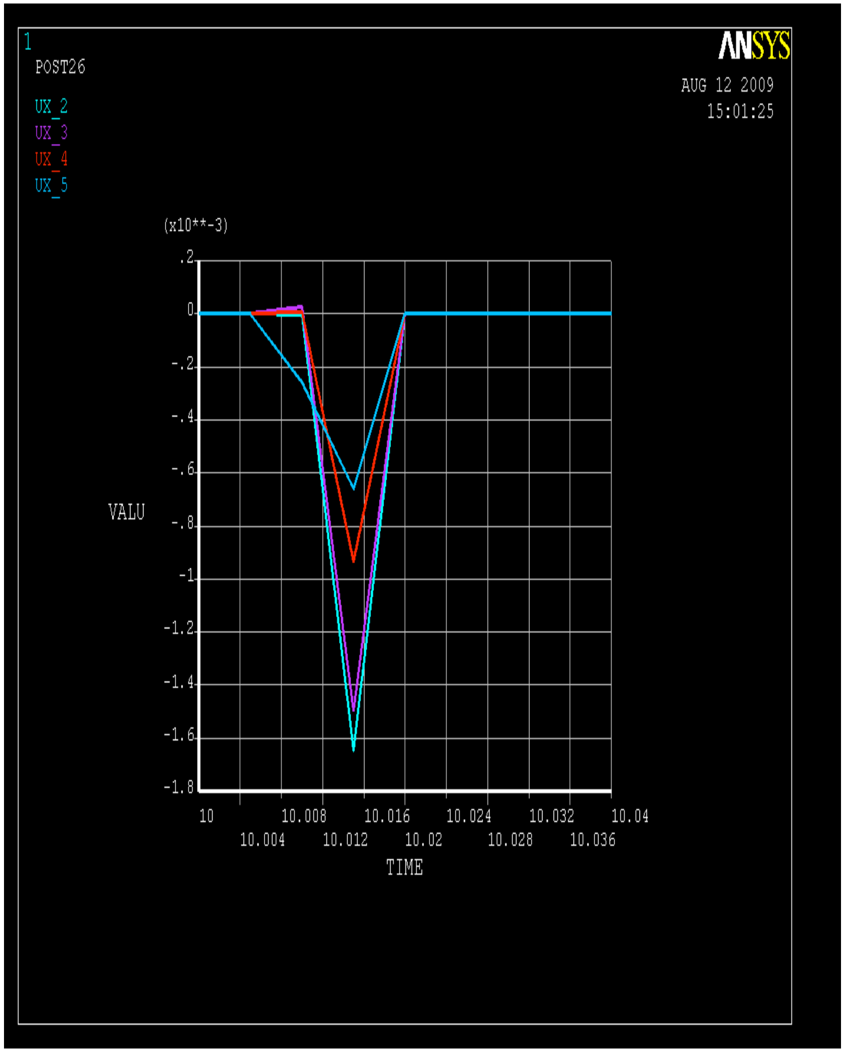


Max Stress =811 Mpa

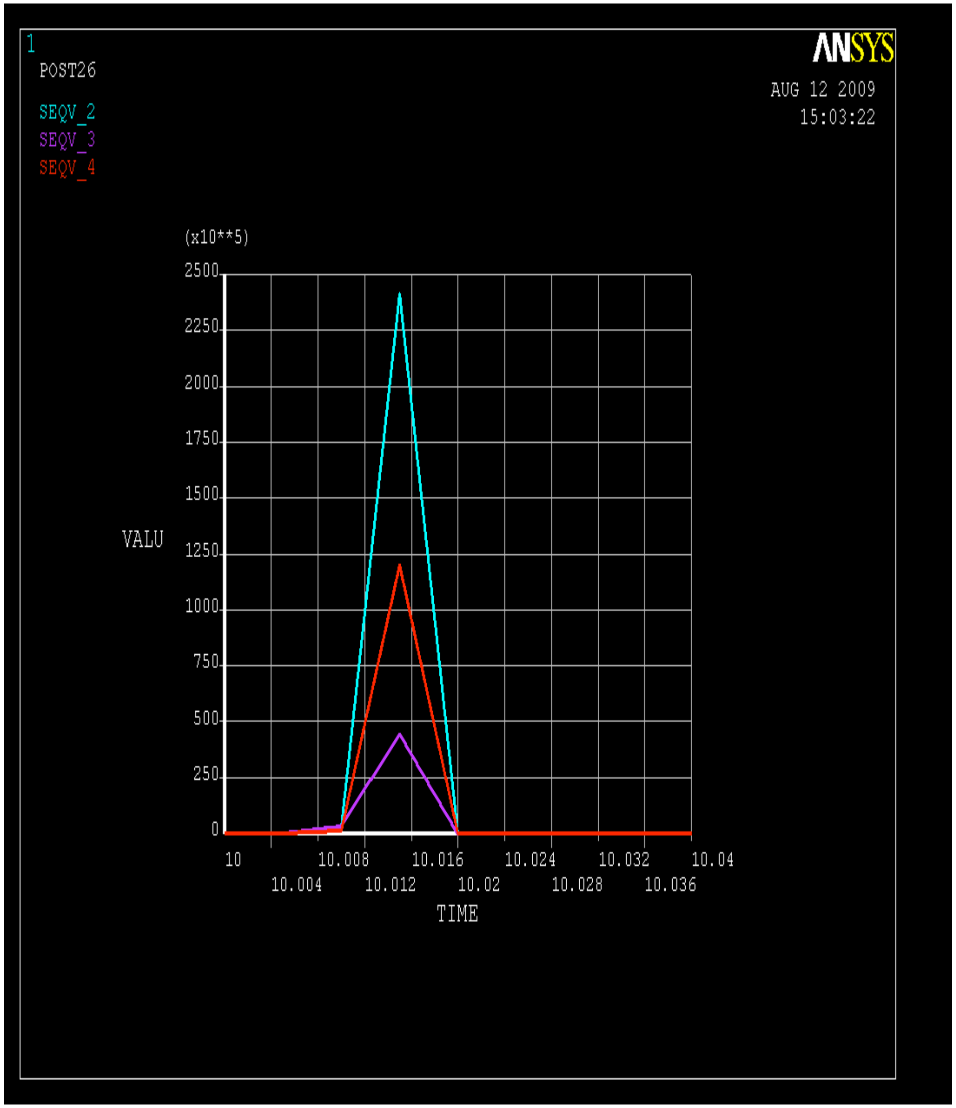


Max Stress =3660 Mpa

Deformations and Stresses

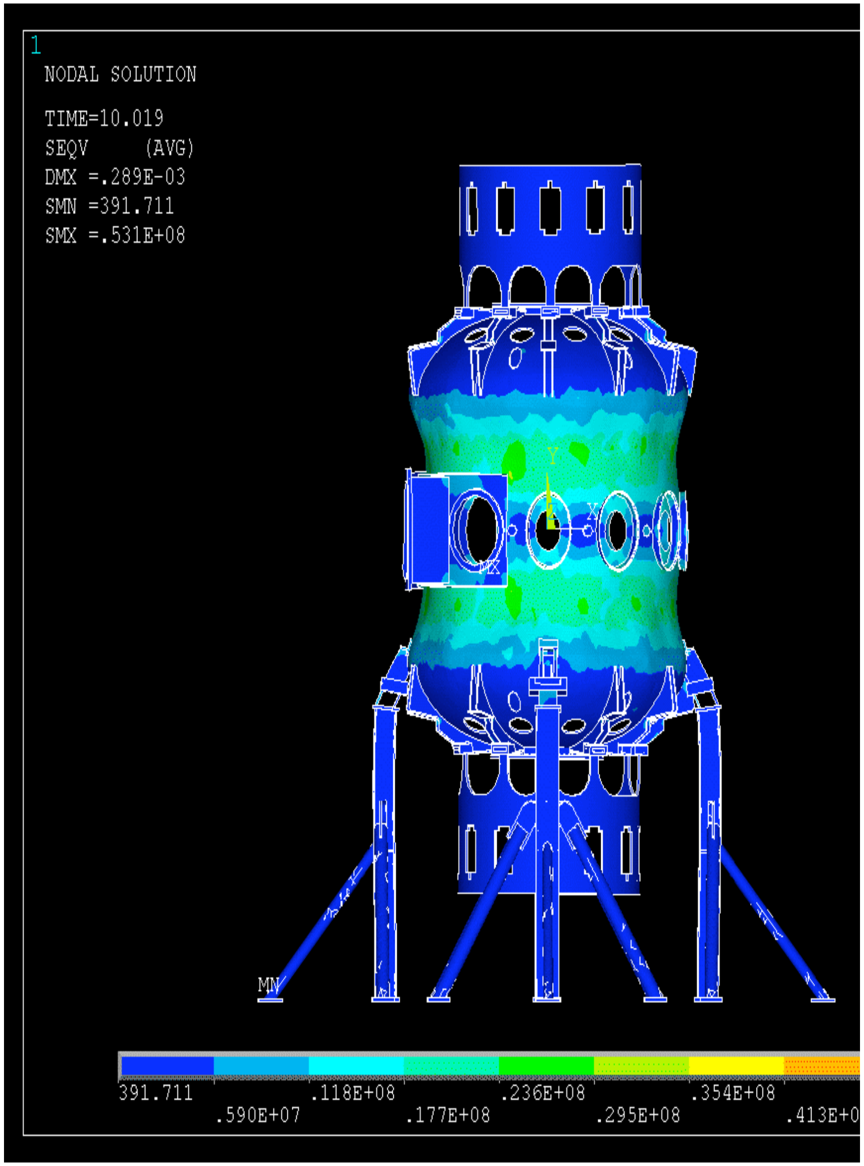


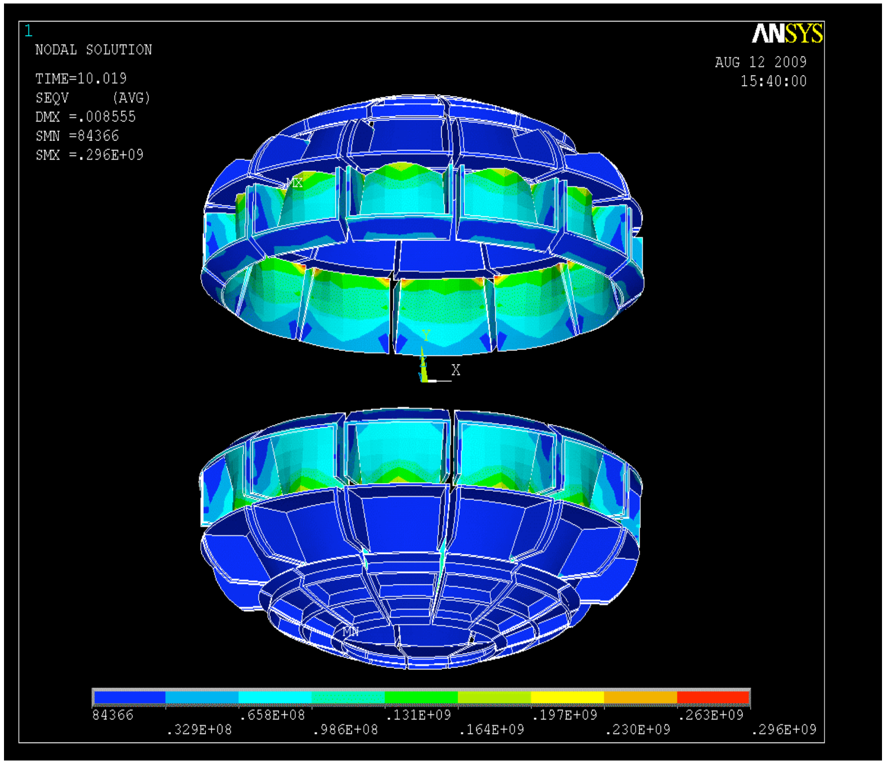
0.625,0.9, 1.5 &1.625mm



40,120 &240 Mpa

DYNAMIC TRUCTURAL ANALYSIS
with Passive Plates

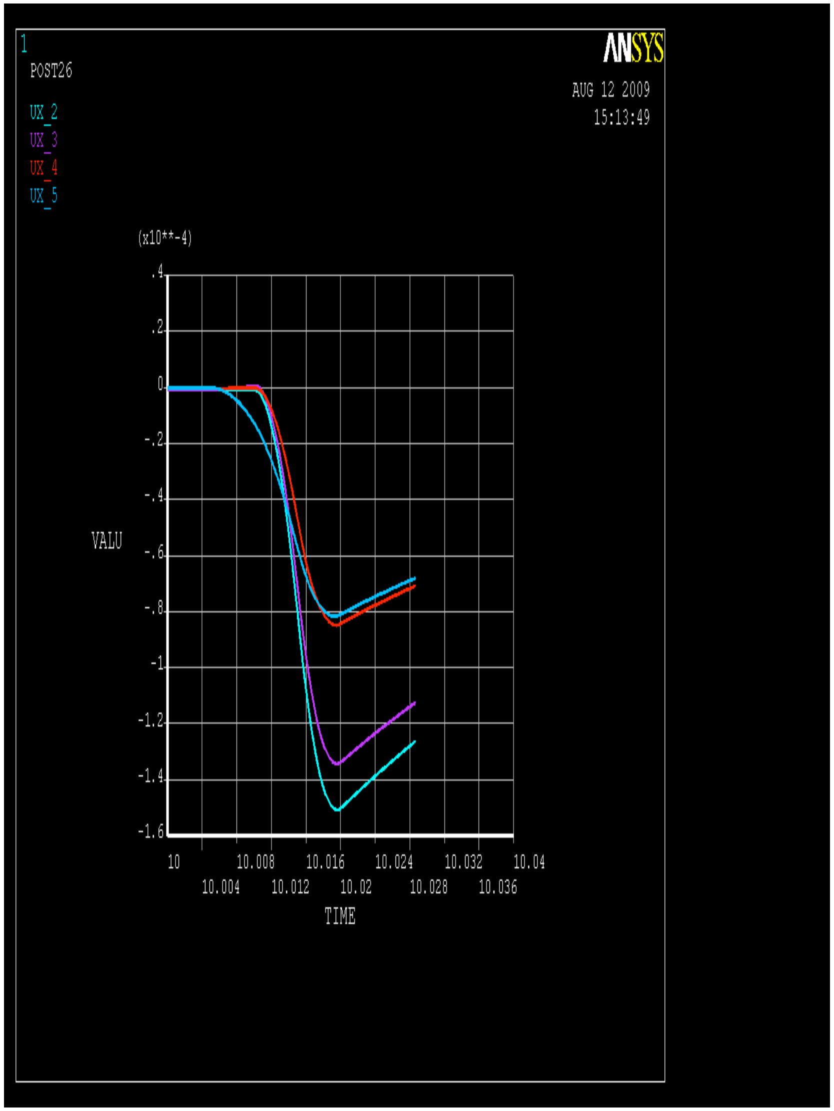




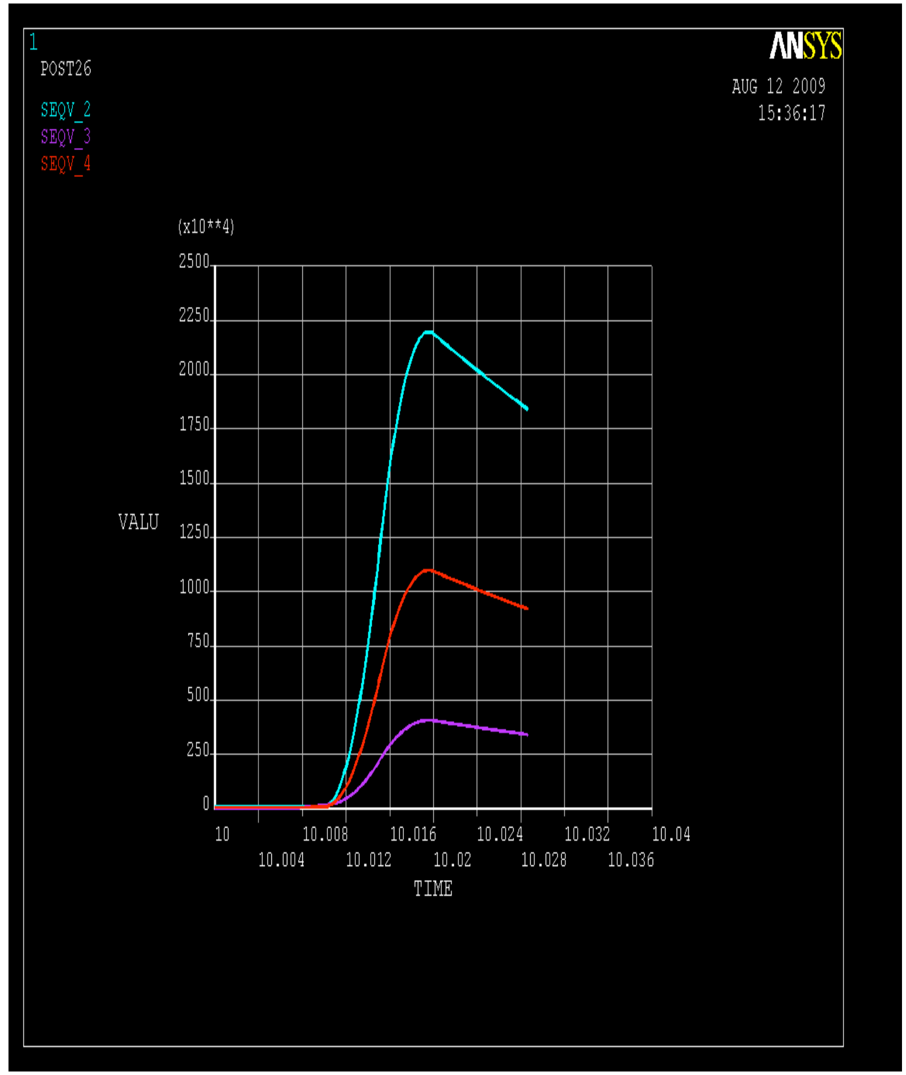
Max Stress
=296 Mpa

Max Stress
=53.1 Mpa

Deformations and Stresses



0.08,0.085, 0.13 &0.15mm



3.8,11 &22 Mpa

Conclusions

- The Dynamic Load Factors are found to less than 0.1
- The stresses are under acceptable limit.
- Macros developed here could be used for other models to simulate disruption stresses. (See Titus' Center Stack Analysis).
- The disruption scenario studied here is just the Out Board Diverter disruption. The other two scenarios : Primary Passive Plate and Secondary Passive Plate should be studied.
- CAD model of the Passive Plates are yet to be obtained and integrated into the model.