F.E. Solutions

CAE Consultants

Customer

About F.E solutions

- Established in 2002 as proprietary concern by Atul Tulpule ,M.S. (USA) having experience in CAE of more than 25 years. Managed by Atul Tulpule and his associates.
- We have supported more than 500 projects for our clients
- We provide CAE results along with design consultation.
- We have expertise in non-linear and dynamic analysis.

What is FEA?

- Finite Element Analysis (FEA) is a computer based method of simulating/analyzing the behavior of engineering structures and components under different working conditions. It is an advanced engineering tool that is used in design and to augment/replace “experimental testing”.

Why FEA and how it will help you?

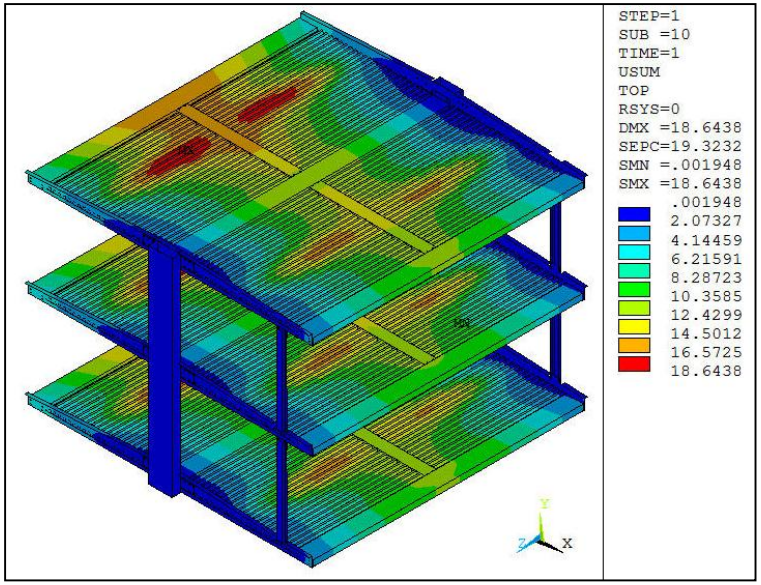
- Allows virtual simulation to anticipate response of structure (stress developed and deformations) to applied loads at design stage. This allows fixing problems at design stage.
- Increases the product reliability and robustness.
- Reduces product development time and cost.
- In almost any type of engineering endeavor, FEA saves time and money.
- Accepted validation procedure by different design codes/agencies.

Where FEA can be applied?

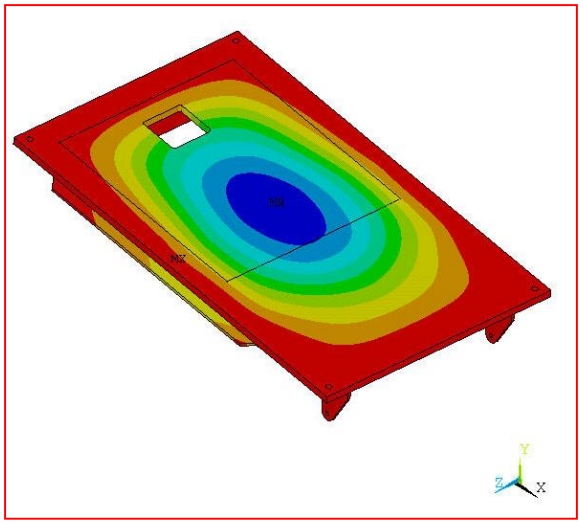
- Structure analysis: a cantilever, a bridge, an oil platform…
- Solid mechanics: a gear, a automotive power train …
- Dynamics: vibration of Sears Tower, earthquake and seismic waves , bullet impact, vehicle chassis
- Thermal analysis: heat radiation of finned surface, temperature distribution in PCB, …
- Biomaterials: dental implants, human organs and tissues…

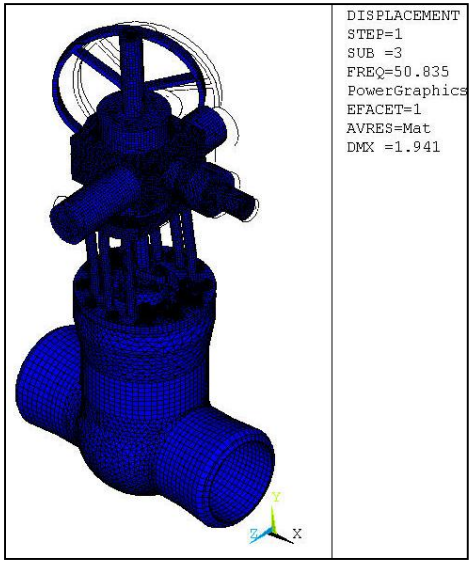
Finite Element Analysis (F.E.A.) Services.

✓ Static Structural Analysis

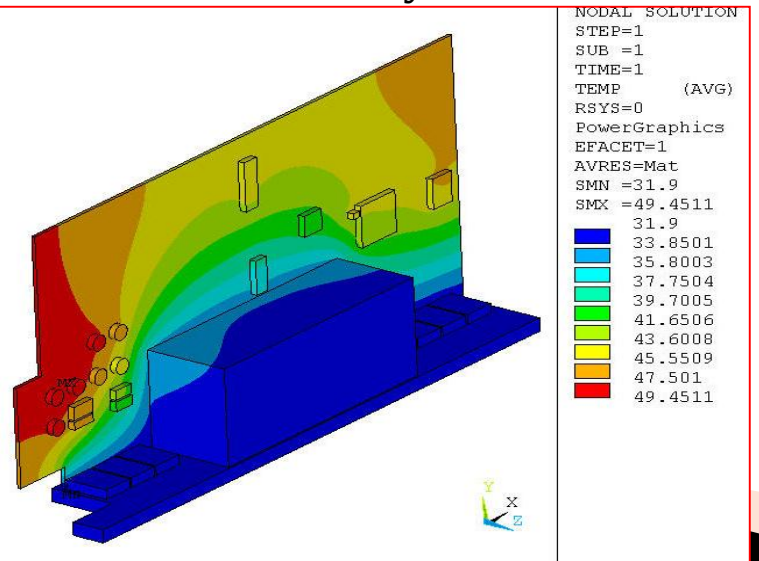


✓ Dynamic Analysis and modal analysis

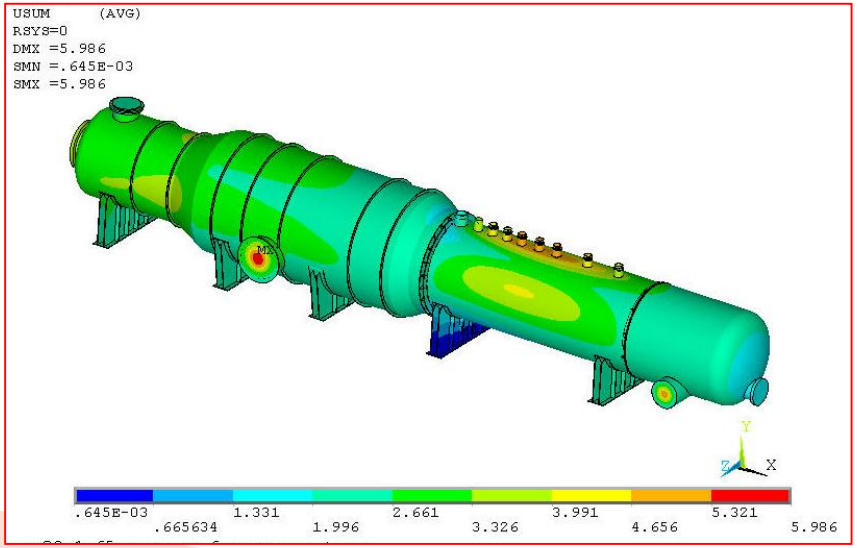




✓ Thermal Analysis

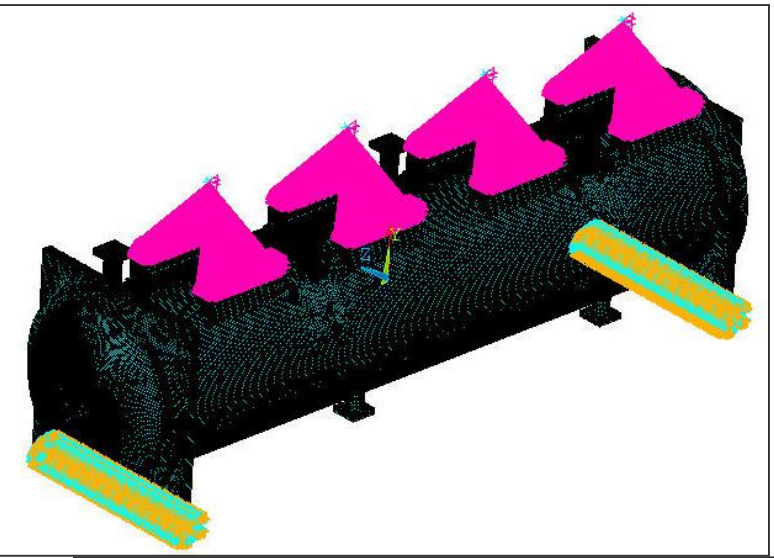


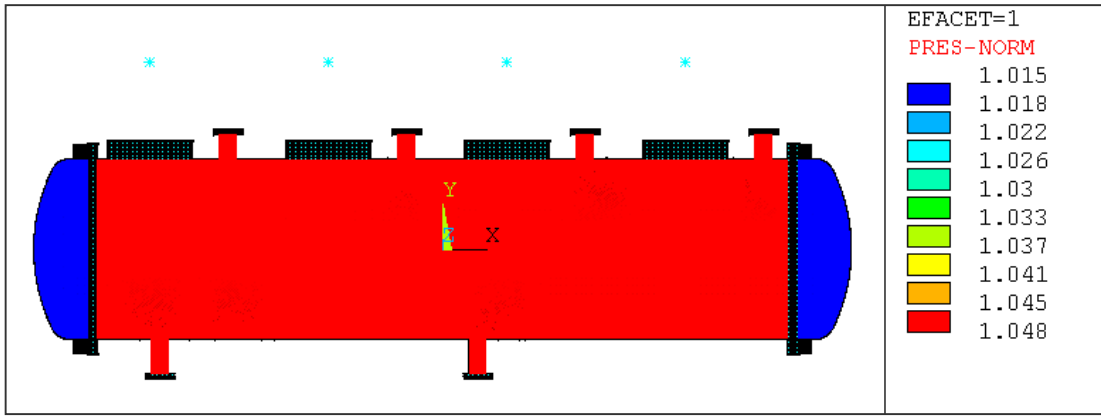
✓ Couple Field Analysis



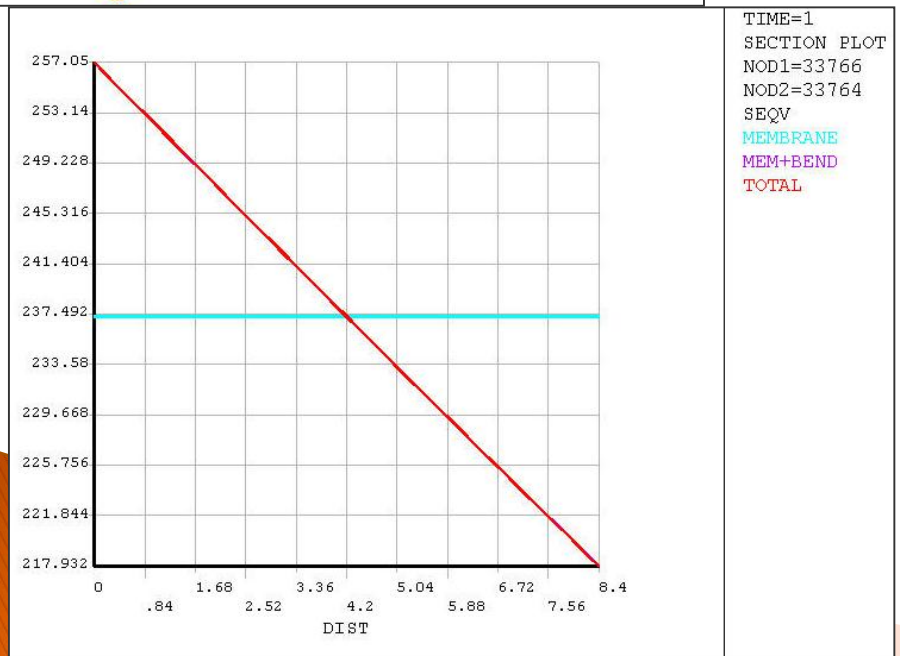
Some sample projects

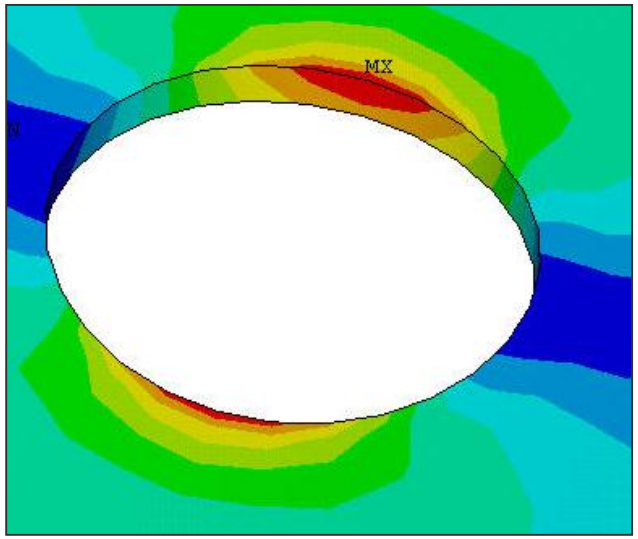
F. E. Analysis of pressure vessel





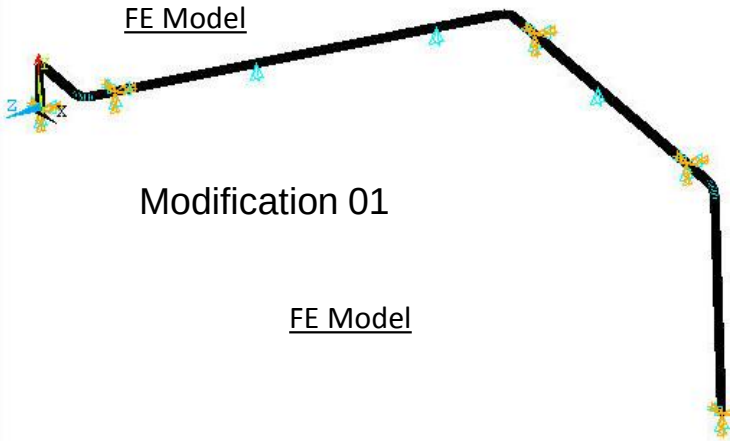
Objective : To find deflection and linearized stresses under different loading condition.





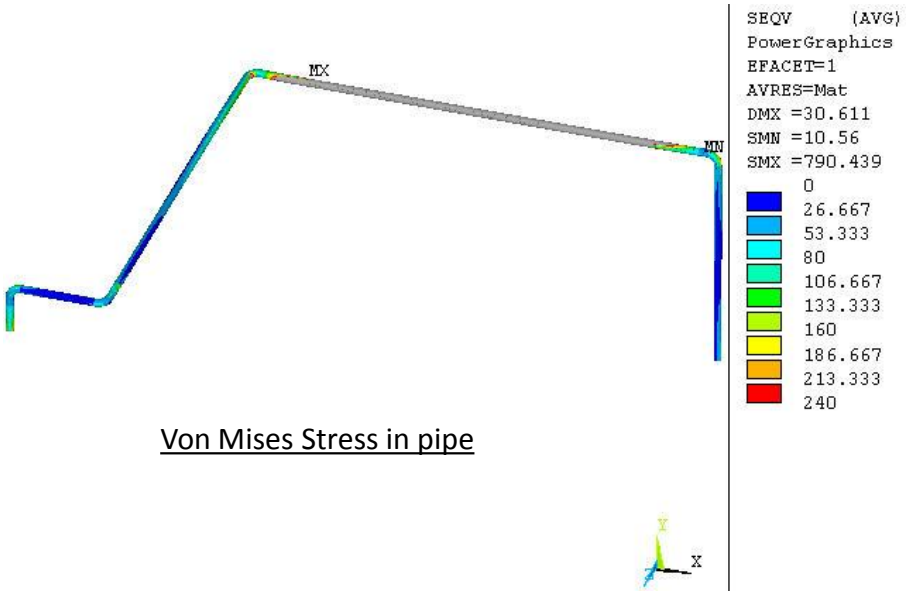
F. E. Analysis of steam and blow down piping

FE Model



Modification 01

FE Model

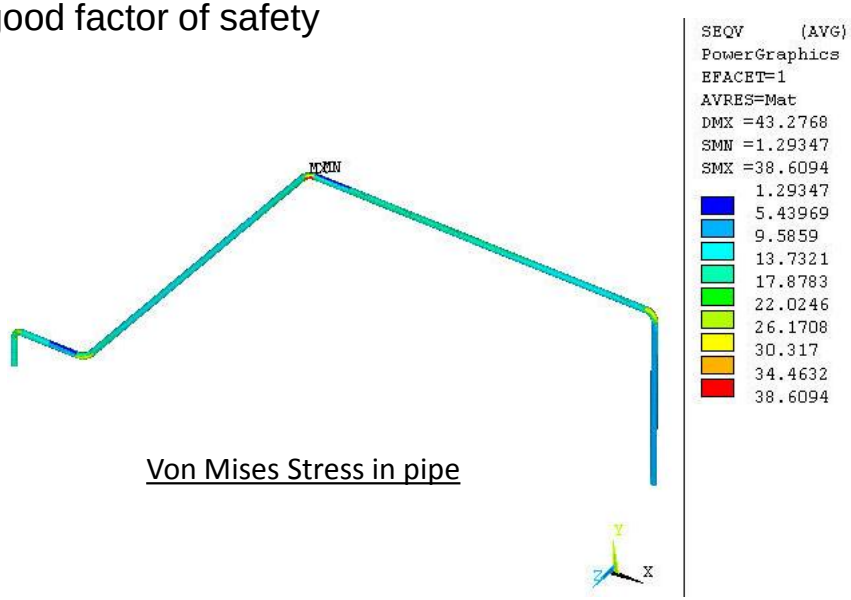


Von Mises Stress in pipe

FE analysis of a piping system for existing design was done. From that it was found that stresses in pipe sections were higher than acceptable limits. It was suggested to remove some fixed supports and increasing roller supports . FE analysis of modified design was done and was found that stresses observed are well below acceptable limit giving good factor of safety

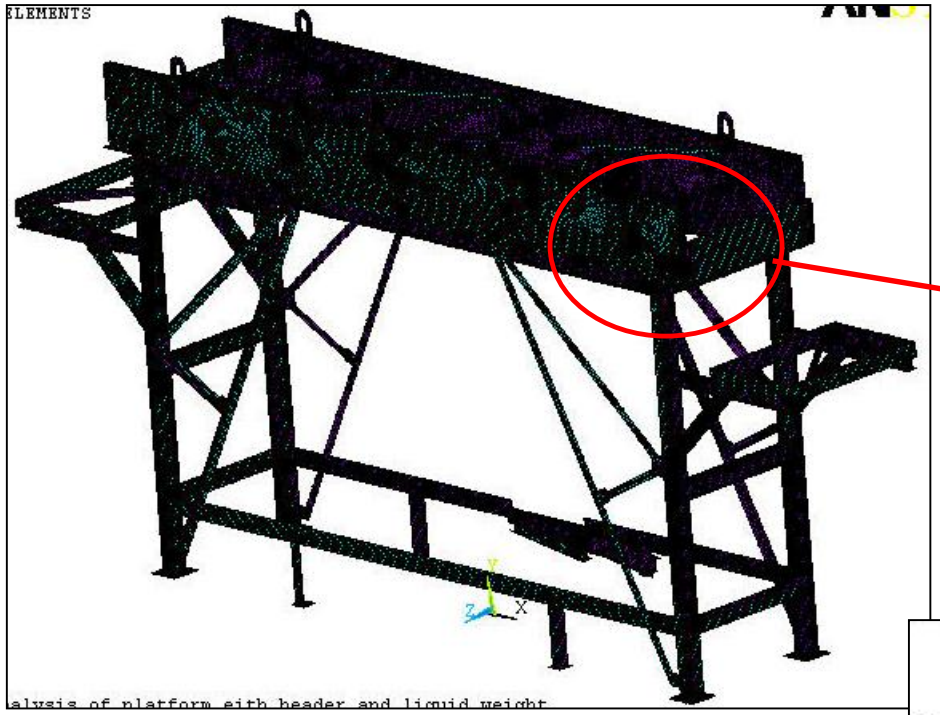
FE Model

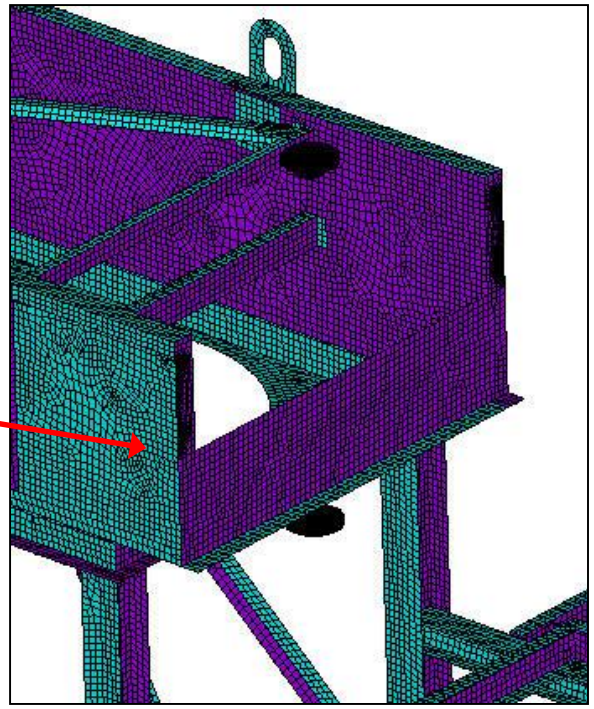
Modification 02

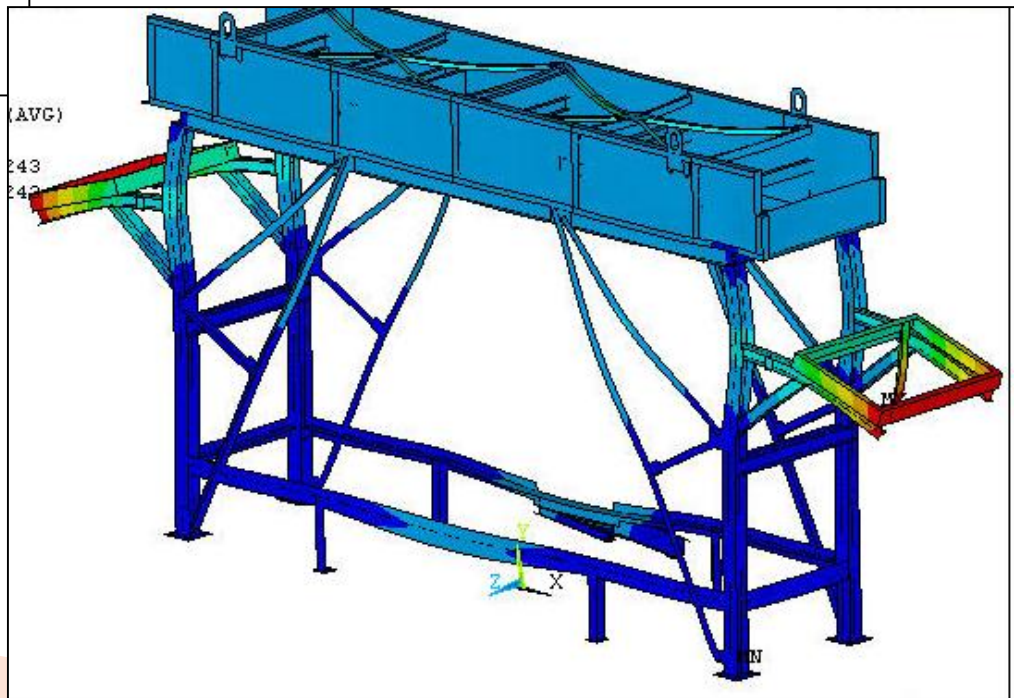


Von Mises Stress in pipe

F. E. Analysis of platform for water tank

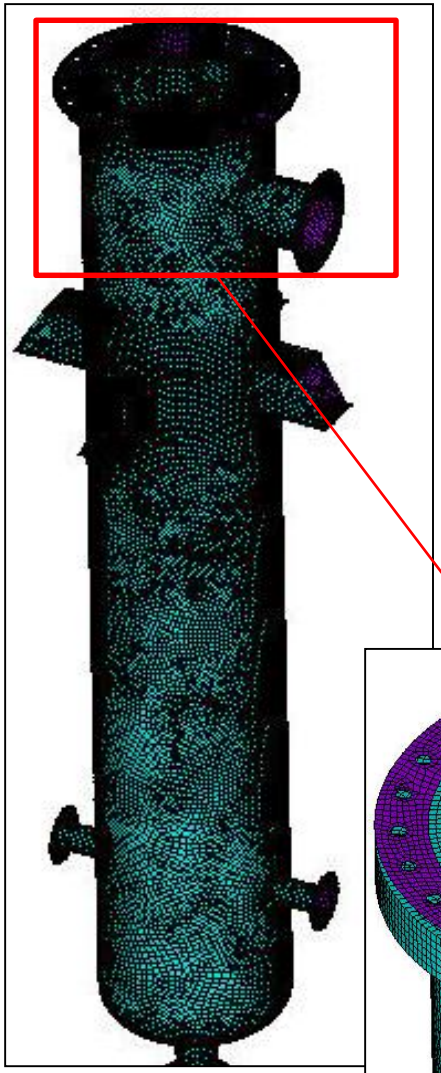


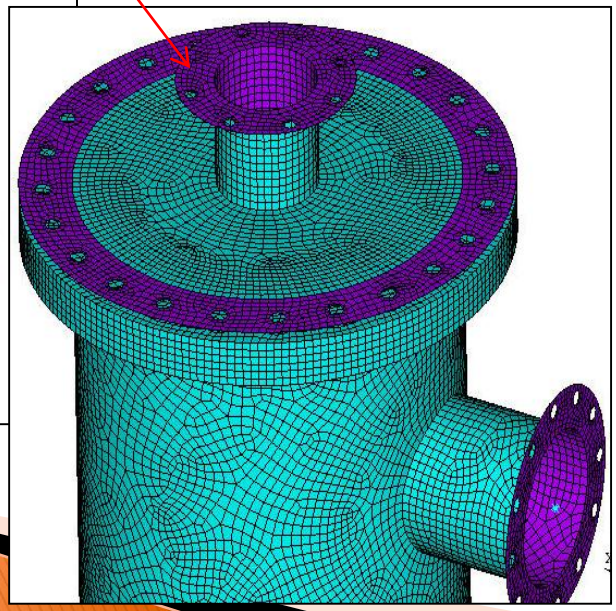


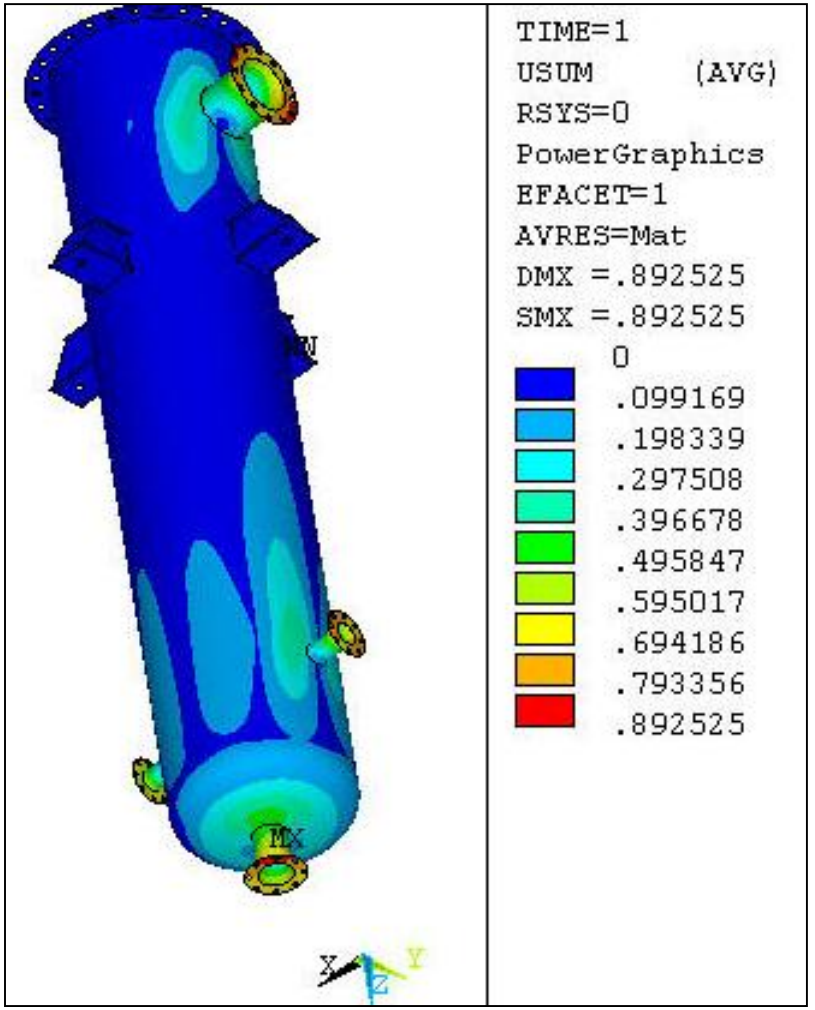


Objective : To find structural integrity of the platform for water load and wind loads and seismic shock.

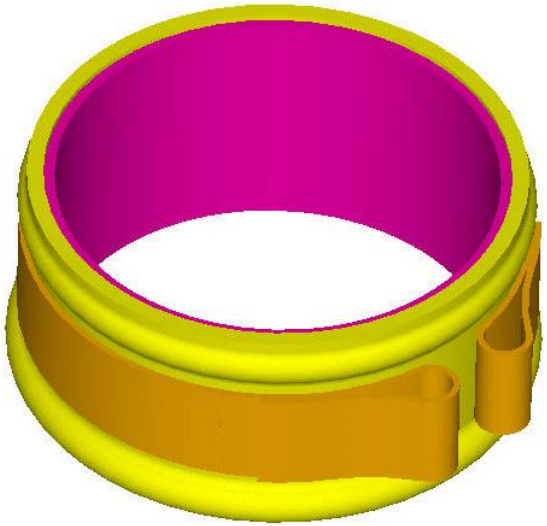
F. E. Analysis of pressure vessel under internal pressure and self weight



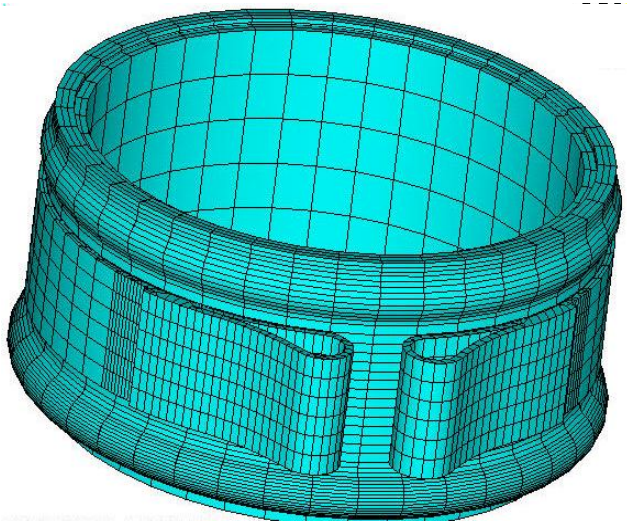




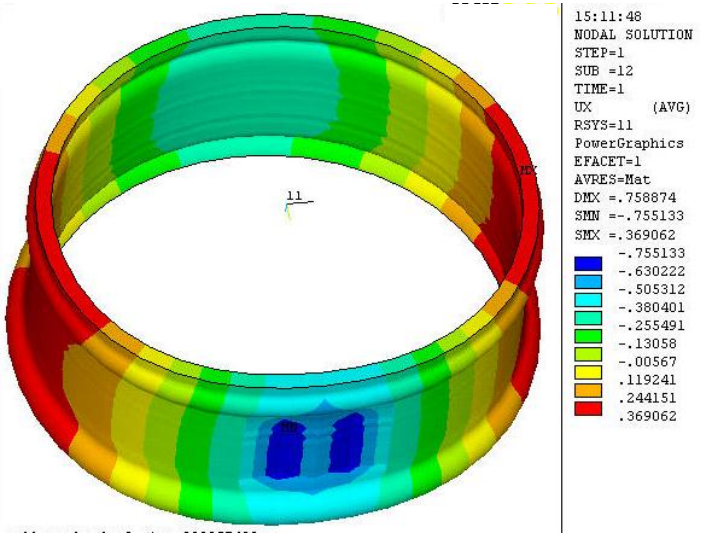
Effect of Clamping Torque on Plastic Pipe with Steel Sleeve (Units in Kg and mm)



CAD Model



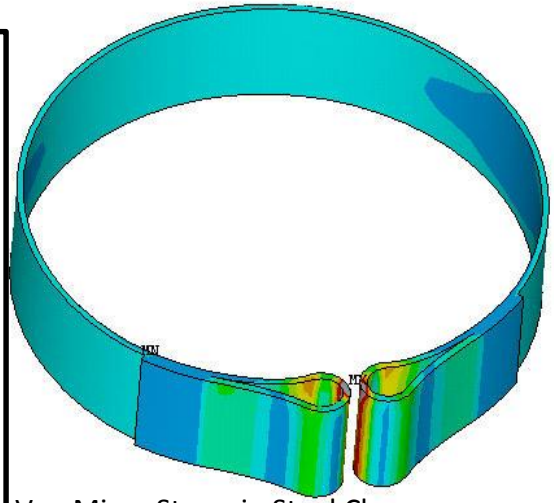
FE Model



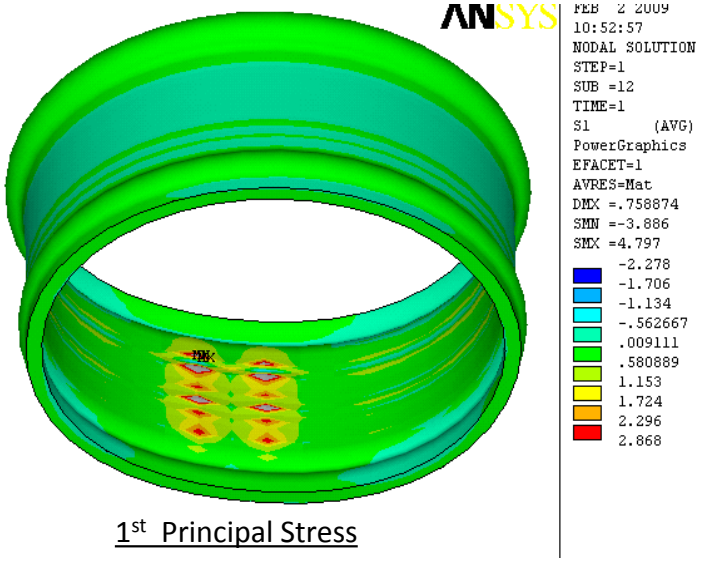
Radial Deformation

Objective: To Evaluate structural integrity of the pipe under tightening torque of 75 lb-inch applied to steel clamp.

Conclusion: Maximum stress in plastic pipe at the area of contact between clamp ends and pipe is very high. The design should be modified for higher FOS.



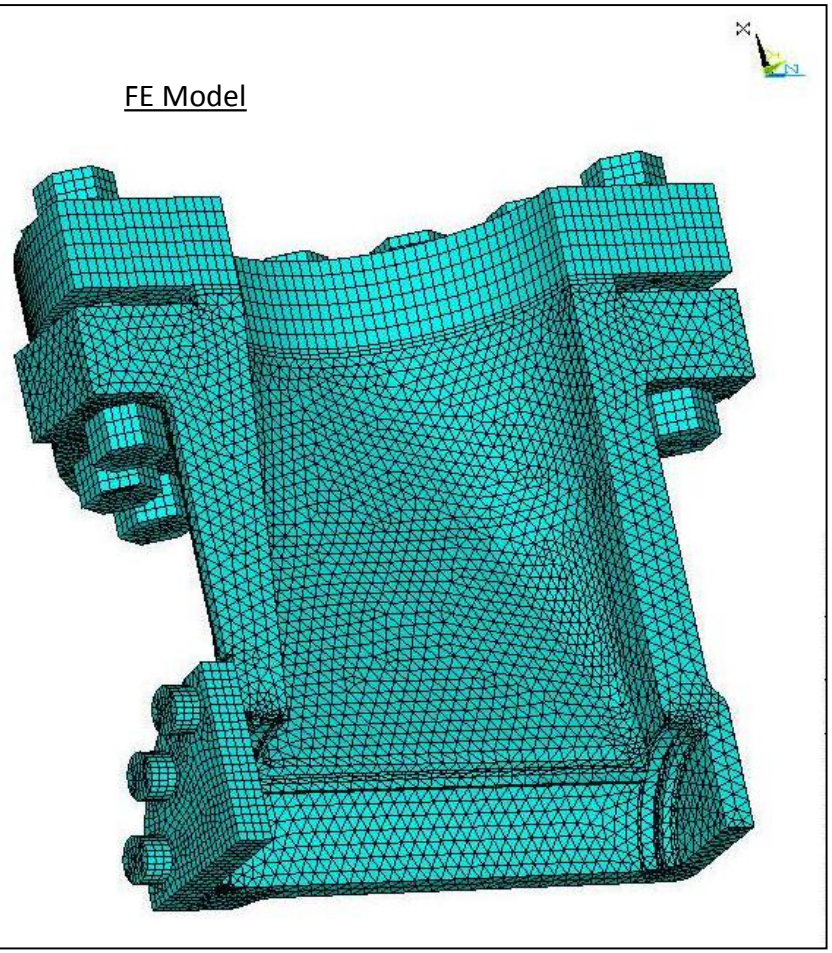
Von Mises Stress in Steel Clamp

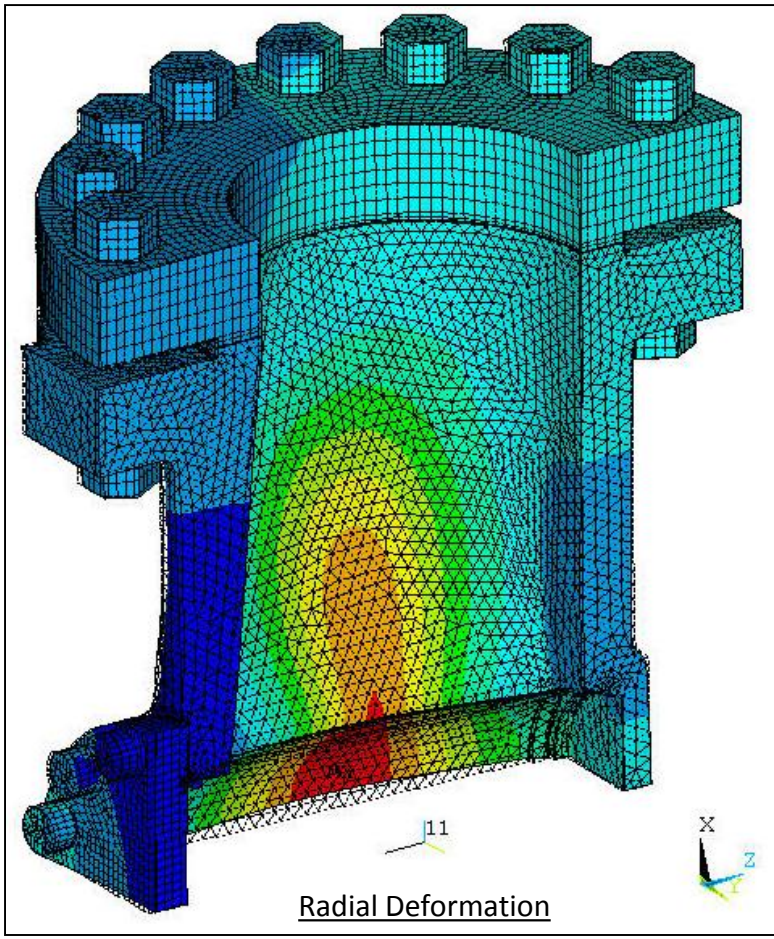


1st Principal Stress

F. E. Analysis of plug body of valve under internal pressure

FE Model



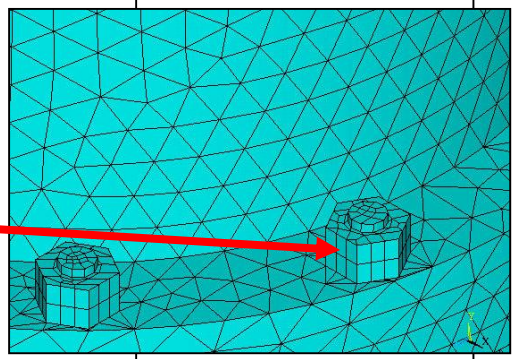


Radial Deformation

Objective: To evaluate radial stress, hoop stress and equivalent stress induced in plug body due to internal pressure and bolt pre-tension.

Modal Analysis of valve body

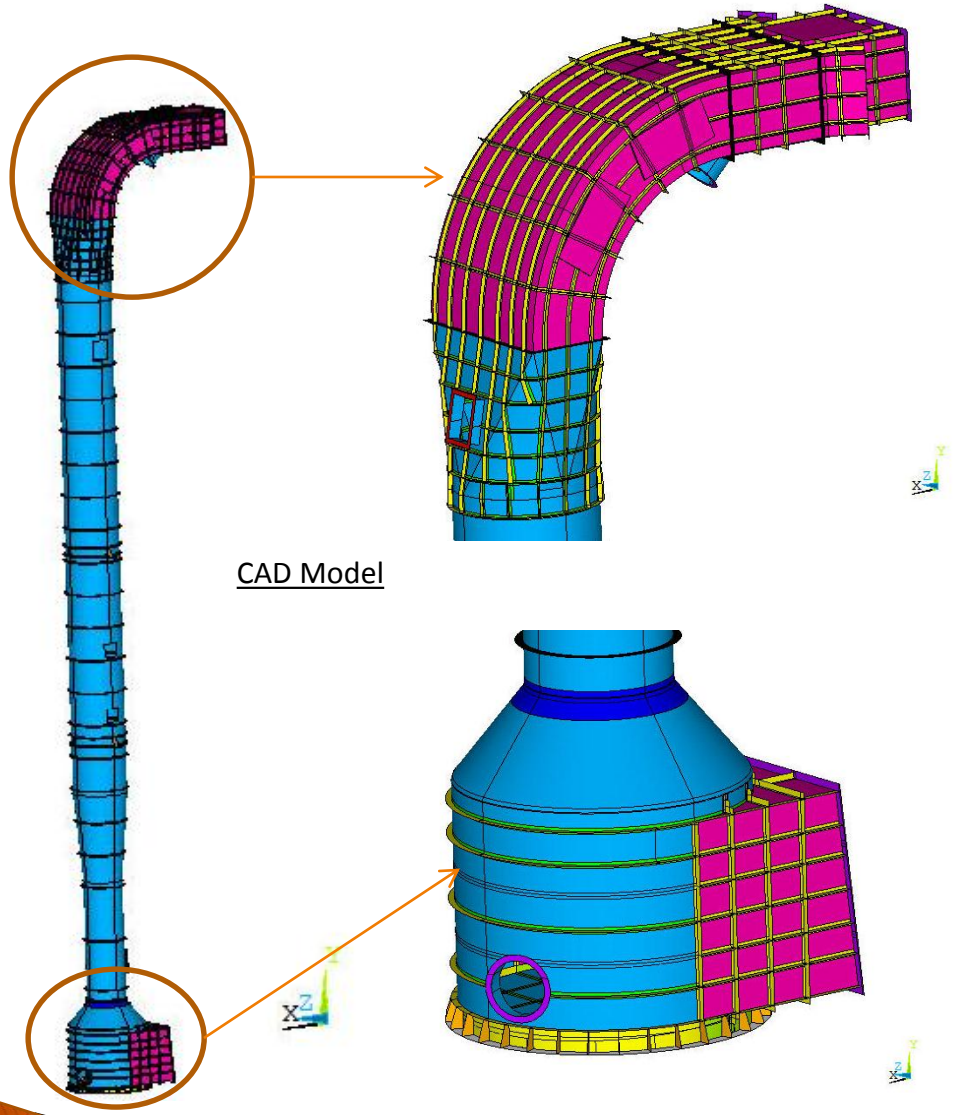
FE Model



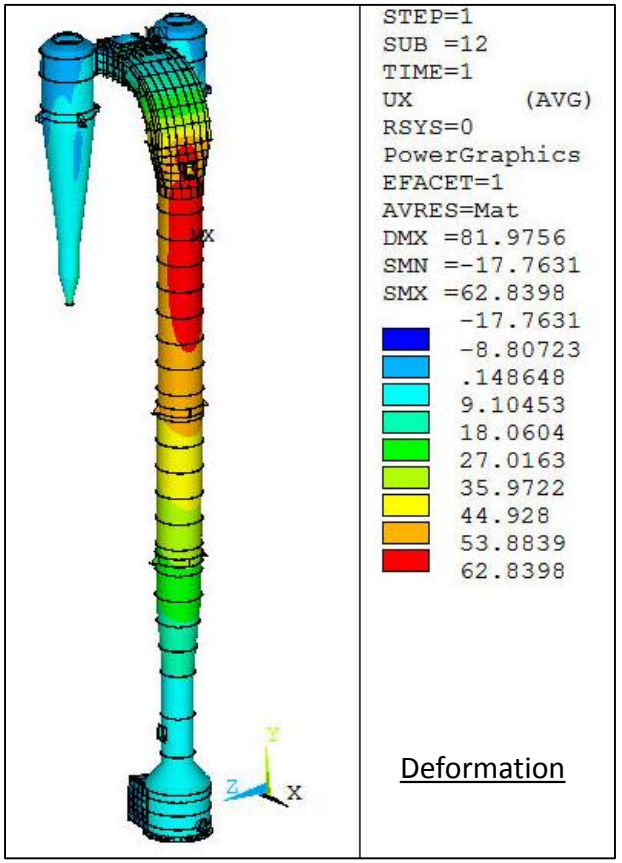
Modal shape

Objective: To find natural frequencies and mode shapes for valve.

F. E. Analysis of vertical duct and cyclone



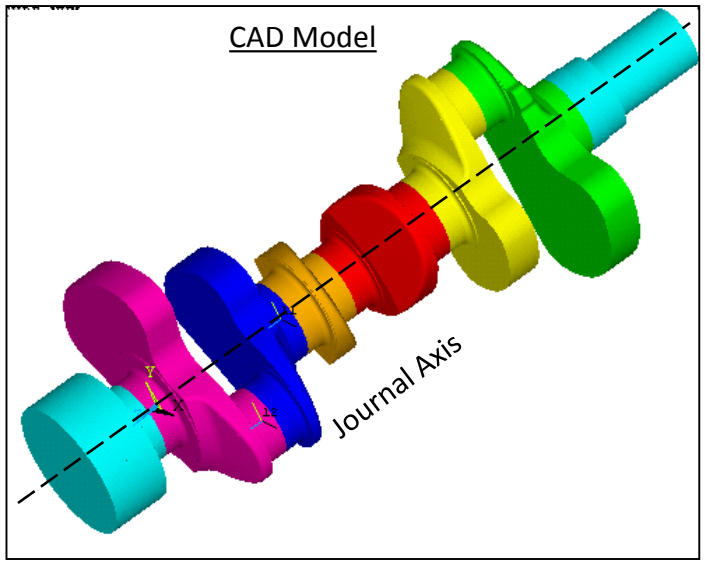
CAD Model

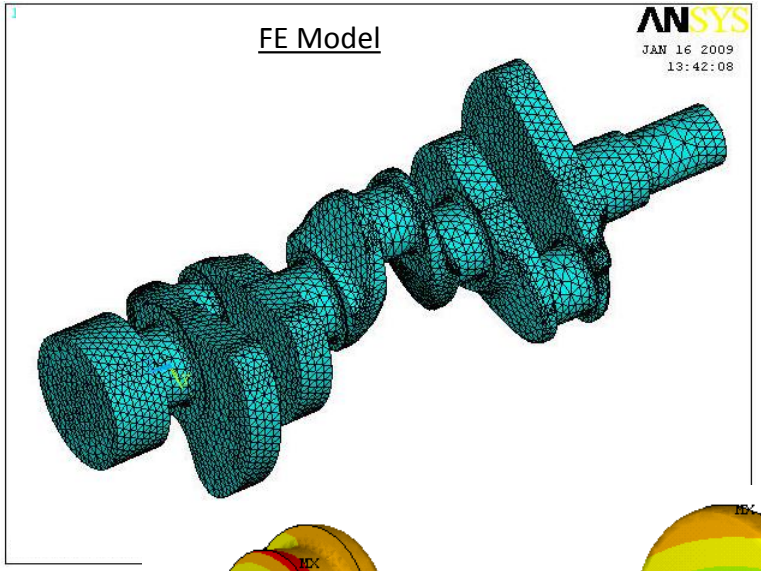


Deformation

To find out stresses and deflections in a the duct load od internal pressure. Also the effect of temperature was also considered during analysis.

F. E. Analysis of Crankshaft to find the Torsional Stiffness

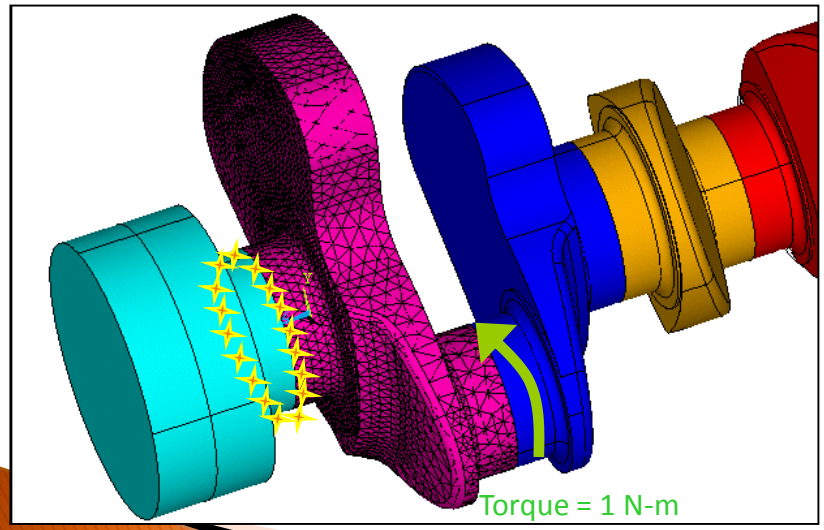


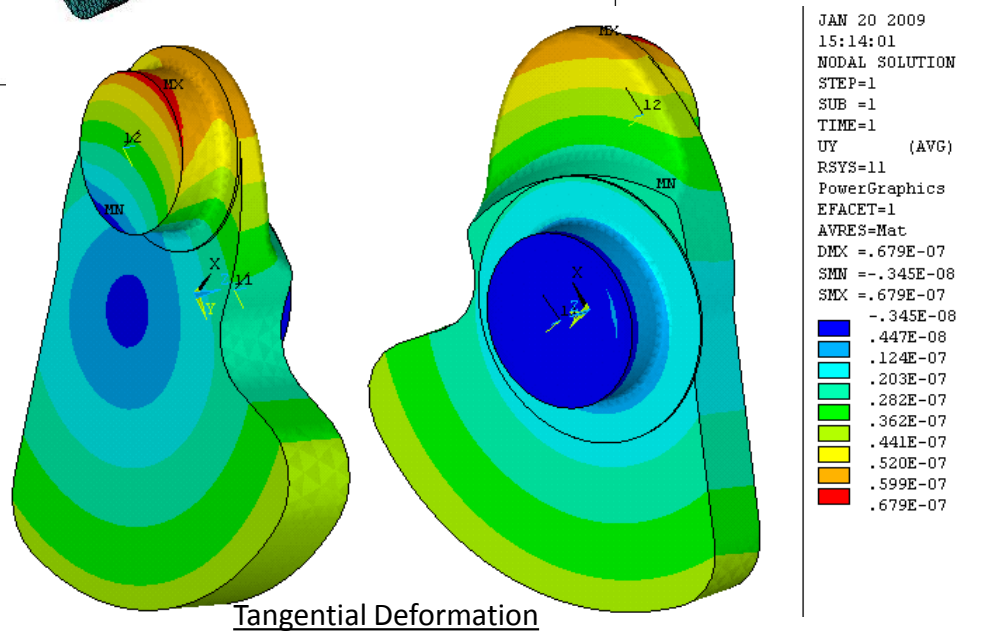


Results:

Tangential deform. (mm)	Torsional Stiffness (N-m/rad)
0.777E-08	3730815

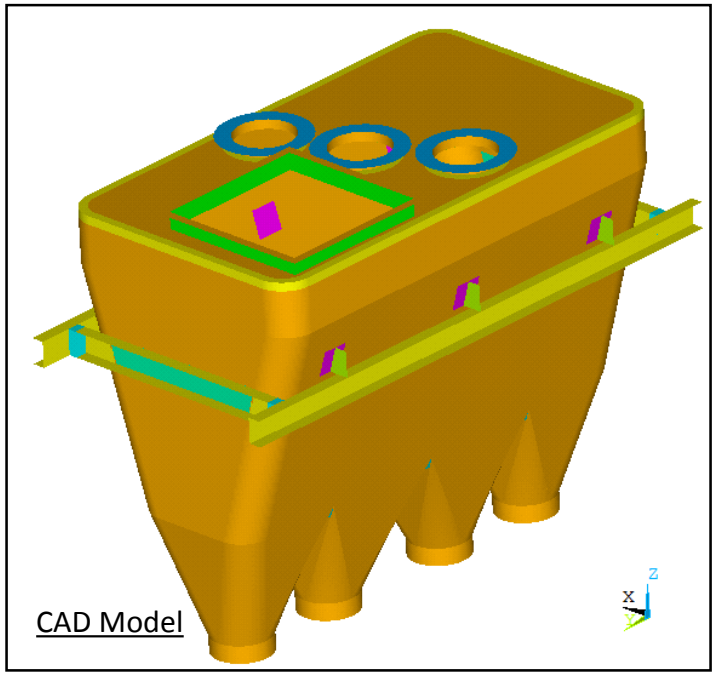
Units N and mm

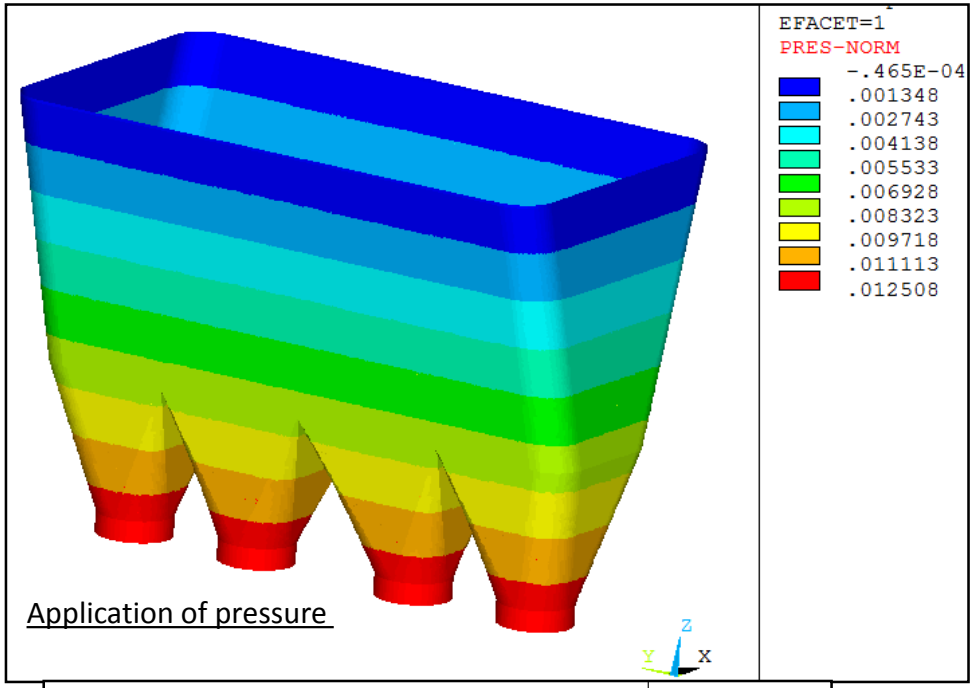


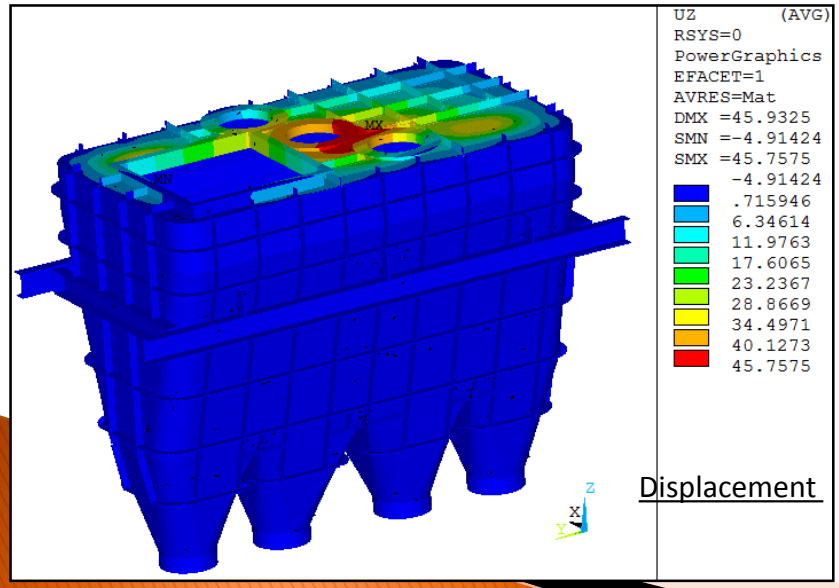


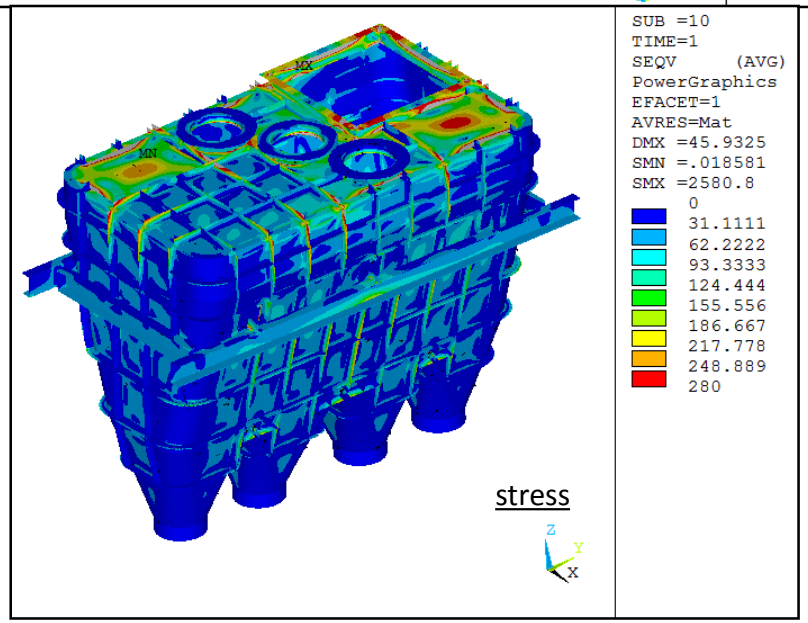
Tangential Deformation

F. E. Analysis of a hopper



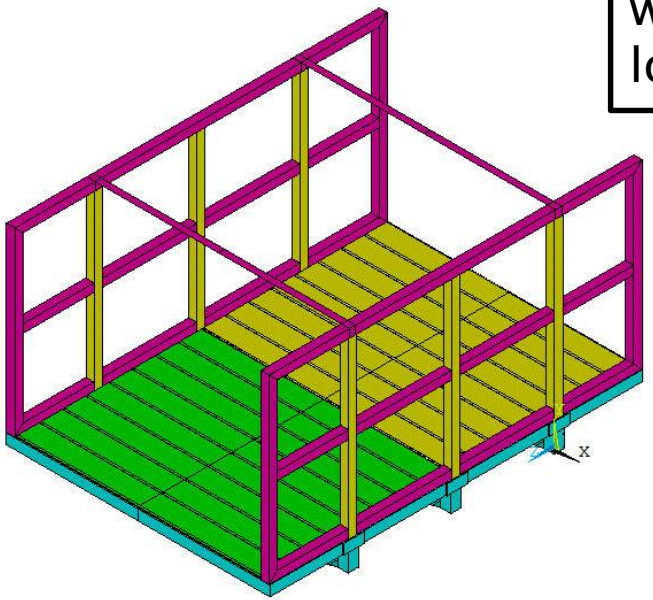




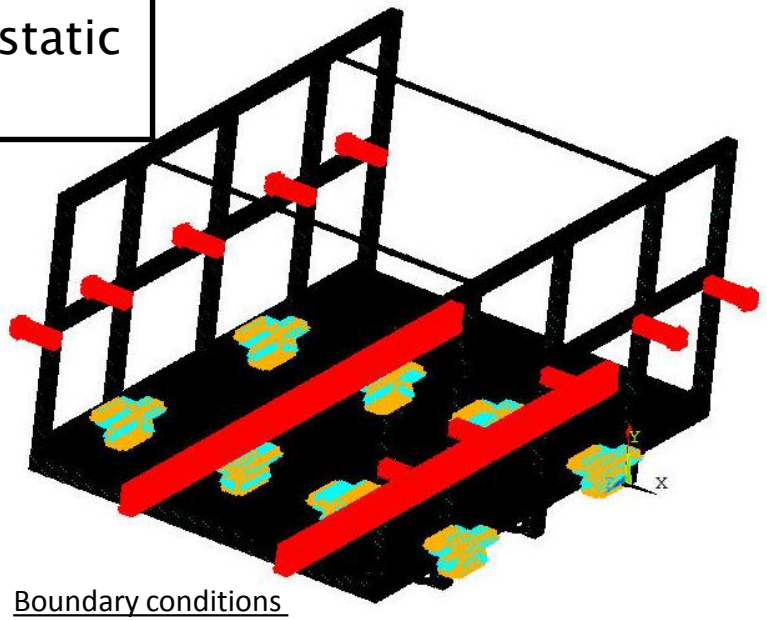


F. E. Analysis of a pallets

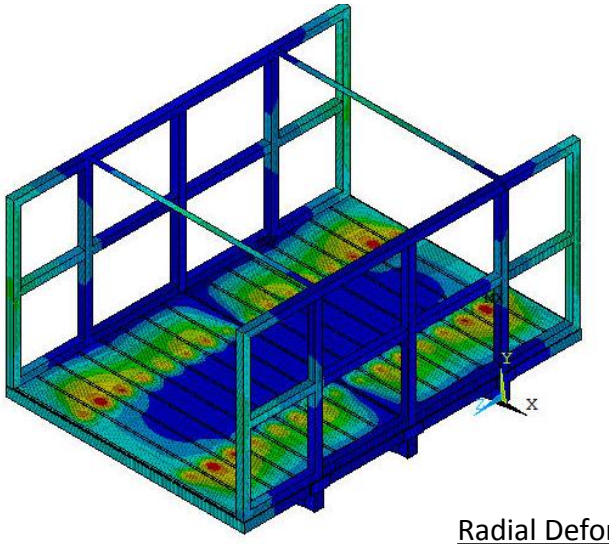
CAD Model

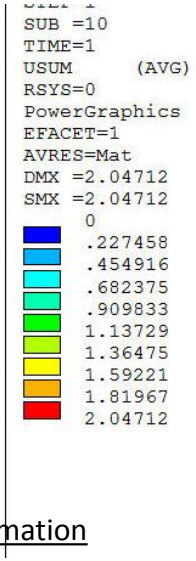


Structural analysis of was done under static loading

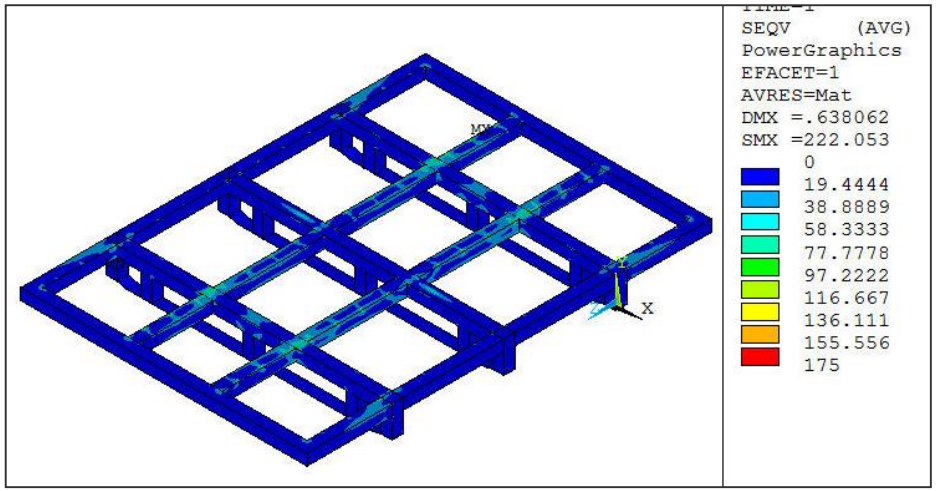


Boundary conditions





Radial Deformation



von Misses Stress frame

F.E.A. Services.

✓ **Credentials**

- Run by team of ME Mechanical Engineers specialized in CAE having an experience of 5+ years each.

- Engaged in Academic Activities providing FEA Learning Solutions to Students of BE Mechanical Course and for industry professionals.

We have more than 500 clients

- ✓ some of them are

Contact us ….

- Address : 1195/4 A, Fergusson College Road, Shivaji Nagar, Pune 411005

- Contact no. : 09823287987, 08237573406

- Email ID: fesolution@gmail.com , atulstulpule@yahoo.com, info@fesolution.in