8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

FINITE ELEMENT APPLIED TO THE ANALYSIS AND SIMULATION OF A CAR TANK DOUBLE CONE

***Cesar Barrón[1], Gearado Atanacio[2], Jorge Sanchez**

[1] Ms
Av Tecnologico S/N
76000 Queretaro, Mexico
gatanacio@hotmail.com

[2] Ing.
Av Tecnologico S/N
76000 Queretaro, Mexico

[3] Ms
Av Tecnologico S/N
76000 Queretaro, Mexico

Key Words: *Finite element method, ASME codice, Industrial Application.*

ABSTRACT

The double cone tank, is a pressure tank, specifically design to trasnport chemical products. It requires an effective and efficient mechanical design.

The double cone tank has been an alternative for carriers, thanks to its geometry and capacity to interchange liquids; this is because of the slope awarded by the tank, which one is noted in the discharge.

Once it is established the design for the double cone tank, the complementary task allowds to develop a mathematic model that diagnoses the structural behaviour in base to the specifications.

The fundamental tool is the application of the finite element models based in 3D models. The initial conditions were defined by using as references geometrical and physics datas of the components of the real tank.

The static structural analysis was the scenario in which one is validated the general assemble of the tank with regard to ASME, visualizing the deformations

agree with the elements asigned by FINITE ELEMENT METHOD to the parts inside of the security margin.

It is considered that the geometry generated in Solid Works in the momento to be exported to SOFTWARE OF FINITE ELEMENT METHOD, conserves the properties of welding determined.

In the contact points that exist in the geometry, is added mass points to evit separations between the parts.

It is determined a global coordinates system to locate the tank symmetry with regard to the Z axis. SOFTWARE OF FINITE ELEMENT METHOD recognized the other part of the tank virtually.

The material asign by ASME carry out with the characteristics of the pressure container design. These materials facilitates the welding and the corrosión that present the tank.

In the static structural analysis it is determined by the sectional geometry the initial conditions of hidrostatic pressure, external pressure and fixed supports at a temperatura of 150°F; simulating the total deformation, the máximum equivalent stress and máximum shear stress, base don the theory of Von Mises.

In the total deformation analysis, it is observed the inform scroll in X, Y and Z, as the strain and compression elements. In the máximum equivalent stress analysis is obtained the graphic representation of the critical stress in the tank. Also, it can be observed the shear stress generated in the tank.

REFERENCES

[1] O.C. Zienkiewicz and R.C. Taylor, *The finite element method*, 4th Edition, Vol. I, McGraw Hill, 1989., Vol II., 1991.

[2] Sanders, William P.; "Turbine Steam Path", Volume IIIa, Mechanical Design & Manufacture; PennWell; Primera Edición; 2004

[3] Dieter, George E.; "Engineering Design"; Mc Graw Hill; Segunda Edición; 1991