Finite Element Analysis and Design Modification of Heat Exchanger and Pressure Vessel for Chemical Industry

E-mail: 9607535@ mail.dyu.edu.tw

ABSTRACT

This report focuses mainly on the analysis of the common pressure container or heat exchanger used in the chemical plant. The formula provided by the design handbook and the design code was employed for finding the required thickness of the container wall for the heat exchanger. The geometrical model prepared by the CAD software SolidWorks was then imported into the CAE software ANSYS for further stress analysis. The internal pressure loading and temperature difference of the heat exchanger under normal operation were imposed on the finite element model. The associated boundary conditions at the supporting stands were also specified accordingly. The simulated results were not only used to validate the design formula but also to find the critical stress concentration sites. Under the condition of keeping the thickness of exchanger wall unchanged, the simulated result can give the distribution of the structural stresses, and thereby the stress concentration sites. Several effects of the local design modification were discussed. Suggestions on the design guideline were concluded and can provide engineers with useful information on the design of pressure vessels and heat exchanger tanks.

Keywords : expander, Young's modulus, tensile strength, yield strength finite element method

Table of Contents

REFERENCES

[1] 1981
[2] 1986
[3] 1989
[4] 1998
[5] 1999
[6] 1999
[7] 2001
[8] 2002
[9] 2002
[10] ANSYS 2004
[11] 2004
[12] ANSYS Workbench 2004
[13] ANSYS ( ) 2005
[14] ANSYS 2006
[15] ANSYS 2006