

ABSTRACT

THERMAL STRESS ANALYSIS OF HEAT EXCHANGER DESIGN SHELL AND TUBE TYPE IN BINARY POWER PLANT CAPACITY 100 KW BY USING FINITE ELEMENT ANALYSIS SOFTWARE (FEA)

**BY
ILHAM DAIRANY**

Heat exchanger is a device used in a binary system of power plant .One of the most commonly used in this plant is the heat exchanger shell and tube type. To improve the accuracy and safety in the design of heat exchanger it is necessary to analyze the temperature distribution and thermal stress that occurs in every component of the heat exchanger using finite element analysis software.

The research method consists of four stages: preprocessing, analysis, postpreprocessing and modeling optimization. Preprocessing consists of two stages, namely modeling heat exchanger component by using Solidwork and meshing with solid 186. Furthermore, the analysis of the temperature distribution by using fluent while for stress by using a thermal stress analysis.

Based stress simulation and optimization modeling be obtained the maximum heat exchanger components, including: shell 155 MPa, 56 MPa tube, rear head 61.4 MPa, front head 83.4 MPa, 6.67 MPa inlet nozzle, nozzle outlet 113 MPa and baffle 73.6 MPa. Then the comparison result of allowable and maximum stress of each component that is above the safety factor used is 1.5. So the heat exchanger components are included in the safety category.

Keywords: heat exchanger, the temperature distribution, thermal stress, finite element analysis software