Stress Analysis of a Pressurizer in a Pressurized Water Reactor Using Finite Element Method

Authors : Tanvir Hasan, Minhaz Uddin, Anwar Sadat Anik

Abstract : A pressurizer is a safety-related reactor component that maintains the reactor operating pressure to guarantee safety. Its structure is usually made of high thermal and pressure resistive material. The mechanical structure of these components should be maintained in all working settings, including transient to severe accidents conditions. The goal of this study is to examine the structural integrity and stress of the pressurizer in order to ensure its design integrity towards transient situations. For this, the finite element method (FEM) was used to analyze the mechanical stress on pressurizer components in this research. ANSYS MECHANICAL tool was used to analyze a 3D model of the pressurizer. The material for the body and safety relief nozzle is selected as low alloy steel i.e., SA-508 Gr.3 Cl.2. The model was put into ANSYS WORKBENCH and run under the boundary conditions of (internal Pressure, -17.2 MPa, inside radius, -1348mm, the thickness of the shell, -127mm, and the ratio of the outside radius to an inside radius, - 1.059). The theoretical calculation was done using the formulas and then the results were compared with the simulated results. When stimulated at design conditions, the findings revealed that the pressurizer stress analysis completely fulfilled the ASME standards.

Keywords : pressurizer, stress analysis, finite element method, nuclear reactor
Conference Title : ICAMSD 2022 : International Conference on Analyzing Mechanical Systems and Designs
Conference Location : Singapore, Singapore
Conference Dates : January 07-08, 2022