

Steam drum is one of the important devices which installed in the steam generator unit. The primary objective of steam drum is to permit separation of the saturated steam from the steam water mixture which leaves the boiling heat transfer surface. The steam drum which installed at unit 7 PLTU UBP Suralaya, one of the biggest steam power plant unit in Indonesia, is designed on the design pressure 2972,68 psi whereas operated on the actual pressure and temperature 2346,85 psi and 351,2 °C respectively. This drum is inspected in every 3 year in order to detect any flaw such as crack in its component and this is done by liquid penetrate test. In order to optimize the inspecting action, therefore, is needed some data to predict the stress distribution in the drum caused by the internal pressure load which is the dominant load resisted by the vessel wall.

There are 3 methods could be used to get the stress distribution, mathematical method, numerical method which commonly uses finite element method and experimental method. In this minithesis, however, the methods being used in analyzing are mathematical referring to ASME code and numerical method using finite element method software. The main parts of the drum to be analyzed are shell assembly, head assembly, riser stub, head downcomer stub, economizer stub and hanger assembly. The primary loads which apply are internal pressure load, total weight of the vessel on the operating temperature.

According to the analysis result, the maximum total stress based on von mises formulation may exist locally at the inside surface of the shell around the riser stubs, that is 25.714,152 psi and 31.414,7 psi based on numerical and mathematical method.

Keyword : steam drum, stress, mathematical method, numerical method.