

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Volume 5, Issue 6, March 2025

## Weld Analysis using Ansys Software

Kunal H. Borase, Susheel Pote, Sagar L. Kute, Nilesh G. Dafade Mechatronics Department Guru Gobind Singh Polytechnic, Nashik

Abstract: A primary responsibility of a designer is to select the type and size of the weld. A number of varieties of welds are available. When it is properly chosen with the correct electrode, it develops full strength of the parent material. The chosen type of weld should develop minimal residual stresses and distortions. Finite element analysis (FEA) has become a practical method of predicting stresses and deflection for loaded structures. FEA accurately identifies the load path, which can be difficult using classical analysis with complex structures. FEA shell element models are effective for predicting loads in weldments fabricated from plate, sheet, structural shapes and tube. The formulation used for a finite element shell model is that of full penetration welds at every joint. Although the loads carried through joints are calculated by FEA, they are not readily presentable. This article presents a method to derive the loads at weld joints from the stress results of FEA shell element models. Additionally, using the calculated weld loads, weld throat stresses or size requirements are calculated using classical methods. [1] Most common basic FEA packages ANSYS was used for this analysis. With its parametric command files, design variations areeasily evaluated. With any FEA package, accurate load estimation depends on the quality of the model built by the analyst. [2] As presented, this method is standard classical weld stress analysis, except that the forces on the weld joint are determined using FEA. The forces through the weld are divided by the weld throat area and compared to the stress maximum. [3].

Keywords: Weld Analysis, Stress Analysis, FEA, Ansys



