Mechanical and Industrial Engineering Department-College of Engineering-Sultan Qaboos University

Modeling and Simulation of Transformation and Twinning Induced Plasticity Phenomena in Austenite Based Steels using Finite Element Method

Rashid Khan - PhD

Abstract

Advanced high strength steels have expanded their application horizon from space to oil/gas industry. Their prominent characteristics of high strength and excellent formability make them an optimum choice for applications where large deformation of a material is desired along with its integrity. Among advanced steels, first generation ferrite based transformation induced plasticity and second generation austenite based twinning induced plasticity steels have excellent combination of strength and ductility. Due to many cost and manufacturing related constraints with second generation, the researchers are now focusing on steels which would have the combination of superior properties of both.

These are placed in the third generation. To relate the effects of phases on the overall properties of these steels, it is necessary to understand the deformation and transformation behavior of phases once subjected to an external thermo-mechanical load. This thesis is aimed to develop a micromechanical model to combine transformation and twinning induced plasticity phenomena in an austenite based steel. First, an elastic-plastic deformation of austenite is modeled using crystal plasticity theory by considering slip only. The model is further extended to include twin deformation in the slip based crystal

plasticity model. This model is used to predict elastic-plastic behavior of an austenite crystal which exhibits twinning induced plasticity phenomenon. Finally, the phase transformation mechanism is incorporated in the model to predict the deformation cum transformation characteristics of an austenite crystal subjected to thermo-mechanical load. Numerical integration algorithms are formulated for the developed models and implemented into a commercial finite element software ABAQUS through a user-defined material subroutine.

Models are then validated through the published experimental and simulation results. Finite element simulations are done for single and polycrystal austenite subjected to simple and complex loading conditions. The observations and analysis of results are expected to provide a meaningful understanding which could help in expanding the application envelop of 3rd generation advanced high strength steels and its specific application to expandable steel used in onshore and offshore oil and gas wells.