Proceedings of the Canadian Society for Mechanical Engineering International Congress 2022
CSME Congress 2022
June 5-8, 2022, Edmonton, AB, Canada

Macroscale Simulation of Tensile Test of Expandable Tubular Steel Using Crystal Plasticity FEM

Omar S Al-Abri[1], Tasneem Pervez[2], Sayyad Zahid Qamar[3], Farooq K al-Jahwari[4]
[1234] Mechanical and Industrial Engineering Department, Sultan Qaboos University, Muscat, Sultanate of Oman
[2]tasneem@squ.edu.om, [3]sayyad@squ.edu.om

Abstract

Expandable tubular steel is a specific class of advanced high strength steel (AHSS), and has been successfully used to drill slim wells as opposed to conventional large diameter wells. Expandable tubulars are made of low carbon steel with distinct amount of Mn, Si and other additives. The steel tubular along with associated parts is used as a complete technology tool during down-hole applications in oil and gas wells. The technology has developed from its primitive form since the first half of last decade to the current much improved form in recent years. However, little effort has been done to fully understand the material and mechanics involved at all scales to develop this technology. Majority of research and development work done for expandable tubular in the petroleum industry relies on trial-and-error approach to find a quick-fix solution without investigating the fundamentals. In the last decade, the elastic-plastic deformation of expandable tubular has been modeled using crystal plasticity finite element method (CPFEM) by considering slip only or slip-cum-twin deformation modes at crystal level. The model is then scaled up to macro-level through simple homogenization scheme. This paper presents the simulation of tensile test of expandable tubular steel specimen through inclusion of all phases present in the material using representative volume element (RVE) for single phase and combination of multiple phases. Each RVE is composed of a few hundred to a few thousand crystals. Averaging the behavior of all the RVEs in the finite element model allowed bridging the gap between single crystal and actual specimen. The commercial software ABAQUS is used to obtain numerical solution through user- defined subroutines. Simulation results of numerical model are first compared with published experimental results and then applied to expandable tubular steel. Stress-strain behavior of actual material from CPFEM simulation agrees very well with experimental data obtained from uniaxial tensile test. It is interesting to observe that the multiscale CPFEM model is also able to closely capture the necking behavior.