**8th International Conference on Physical and Numerical Simulation of Materials Processing (ICPNS)**

**14–17 October 2016**

**Seattle, Washington | Hosted by Purdue University**

**SESSION 4: WELDING AND COATING, SALON E**

Co-Chairs: Pingsha Dong, University of Michigan; Honggang Dong, Dalian University of Technology; Oksana I. Shpigunova, Tomsk State University

**SUNDAY, OCTOBER 16, 2016**

# Numerical modeling and simulation of welding residual stresses using finite element method

Jing Zheng; Ayhan Ince, Purdue University

## ABSTRACT

Welding is one of the most commonly used joining methods in many industries for structures and components fabrication. Welding is considered a highly complex metallurgical process that results in material imperfections and flaws and residual stresses. Residual stresses and distortion induced by welding process has a significant impact on fatigue life of structures and thus a topic of great concern in product design and safety. The residual stresses need to be taken into for accurate fatigue life prediction of welded structures. Finite element (FE) method has become a popular tool for calculation of welding residual stresses and distortion. This paper present modeling and simulation works to perform thermal elasto-plastic analysis using the FE code ABAQUS to predict residual stresses at fillet welds. The magnitude and distribution of distortion, and residual stresses are analyzed considering different weld sizes, the temperature-dependent material property, and the movement of heat source. The simulation results show that the distribution of residual stresses correlates well with experimental data.

**KEYWORDS:** finite element method, weld process, residual stress, weld distortion, welded structures, Abaqus