



Contribution ID: 5

Type: **not specified**

ABAQUS for fracture mechanics analysis

This module covers the fracture mechanics analysis of RPV materials under pressurized thermal shock using the finite element method in ABAQUS. This requires that the participant has basic knowledge in ABAQUS software as well as intermediate knowledge on fracture mechanics analysis. A simple example is developed and analyzed.

Presenter: MORA MENDEZ, Diego Fernando (PSI - Paul Scherrer Institut)