Simulation

Abstract

The ultimate end-goal of this collaborative project is to find a high throughput method of performing ductility testing on materials. In order to successfully obtain this end goal, the problem must be approached from a couple of different areas including simulations of the experiments to prove relevance, and experimentation of said experiments after its relevance has been proved from the simulation.

On this side of the project is where the simulation information can be found. Once the simulation team receives information from the experimental team such as possible set ups, feasible parameters and intended uses, the simulation team can then use CAE software to test their hypothesis. In this particular group, the simulations will be conducted via Finite Element Analysis through the ABAQUS software. Through this software, the team is able to set up the apparatus, run the projected tests on the material and analyze various parameters such as the stresses, elastic and plastic strains, and much more to test and predict the final results. The results from these various simulation can be found on this page.

Introduction

The research data and visuals found in this section will be used to test hypothesis and confirm relevance of future ductility testing. The simulations will begin at it’s most simplest form and continue to increase in complexity as the team moves forward.

Research workflow

- step 1: 2D Spherical indentation simulation: Die contact included
  - 2D Indentation Simulation

- step 2: 3D Indentation Simulation: Square plate
  - 3D Indentation Simulation

- step 3: 3D Indentation Simulation: Thin plate

- step 4: Multiple Small Punch Test (SPT) simulation: including damage

- step 5: Tensile test simulation to calibrate material properties accordingly
  - Parametric study

- step 6: Modify Boundary conditions to match experimental setups (indentor/uniform pressure – fixed B.C. – introducing dye)

- step 7: Parametric study
Step X: Parametrization of the Curves / Tensile – SPT linkages

In this section will be presented the work done toward the parametrization of the experimental small punch Test results (Load-Displacement curves).

Indeed, the goal in this project is to determine the mechanical properties of a material using SPT testing, or in others words, to be able to retrieve the tensile properties (such as defined in a tensile test) but from the SPT curves. In order to build linkages between the SPT curves and the mechanical properties of a material, the SPT curves have to be simplified to a reduced number of parameters.

Results and discussion

This section is a summary of discussion and feedback that you are getting from each sub-wikis, step1 to step3 or more.

Resources/Data

here provide the repository that your data or codes are stored:

- link 1
- link 2

References
list of all references in this page