Two-Stage SPG Algorithm for Joint Performance Analysis

Youcai Wu

Computational & Multiscale Mechanics Group

Livermore Software Technology, an ANSYS Company

December 28, 2020

**Ansys**

Introduction

- **Objectives**
  - **To perform two-stage analysis with SPG**
    - **First stage: joining process analysis**
      - **Example procedures: Self Piercing Riveting (SPR), Flow Drill Screwing (FDS), Anchor Setting**
    - **Second stage: joint performance analysis**
      - **Can have new: materials, parts and boundary conditions**
      - **Example procedures: Lap Shear, Coach Peel, Cross Tension, Pullout**

- **Approaches**
  - **At the end of 1ˢᵗ stage, write out history variables**
    - ***INTERFACE_SPRINGBACK_LSDYNA**
      - **Output FEM history variables and total displacement of the whole model**
    - ***INTERFACE_SPG_1**
      - **Output SPG variables**
  - **At the beginning of 2ⁿᵈ stage, read and map history variables from 1ˢᵗ stage**
    - ***INTERFACE_SPG_2**
      - **Read SPG variables from 1ˢᵗ stage calculation**
    - **"dynain" generated by "*INTERFACE_SPRINGBACK_LSDYNA" in 1ˢᵗ stage should be used**

Setup for First Stage
Part Set for "dynain"

- **Keyword: *INTERFACE_SPRINGBACK_LSDYNA**

- **Part set for "*INTERFACE_SPRINGBACK_LSDYNA"**
  - **Including all the parts that will be used in the 2nd stage**
    - SPG parts as well, since latest coordinates are needed for all nodes
   - Excluding those are NOT used in the 2nd stage

Rigid driver

Rigid clamp

Excluded from the part set

Rigid die

SPR and strength
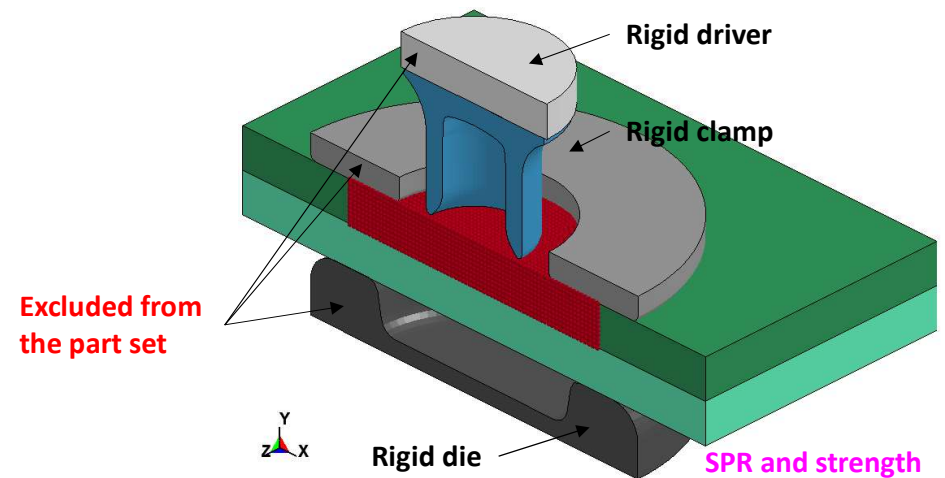
Setup for First Stage
Output SPG Variables

- **Keyword: *INTERFACE_SPG_1**
  - **No parameter, just one line**
  - **An ASCII file named "1234spg" will be generated at termination**
    - **The file contains information of:**
      - **Stress and effective plastic strain at SPG nodes**
      - **SPG NIDs and their corresponding NIDs of neighboring particles, support sizes, IDAM variable, nodal density, nodal active or not**
   - **Total displacement at all nodes**

/Ansys

Setup for Second Stage

- **"dynain" generated at 1st stage must be included as an input file**

- **Copy "1234spg" generated at 1st stage to current folder and rename as "1234spg0"**

- **Add keyword: *INTERFACE_SPG_2**
  - No parameter, just one line
  - **NOT** "*INTERFACE_SPG_1"

- **Tips**
  - Element connectivity and nodal coordinates in 1st stage input files need to be deactivated since the ones in "dynain" are used in 2nd stage
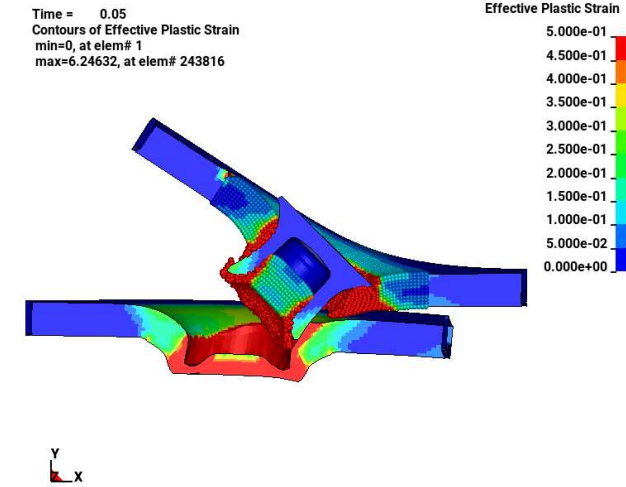
Responses of SPR Joint Lap Shear (Section View)
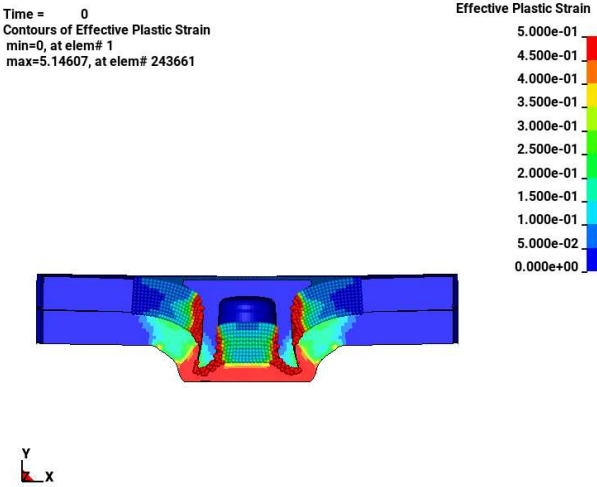
Plastic strain at termination of 1st stage and the beginning of 2nd stage as well
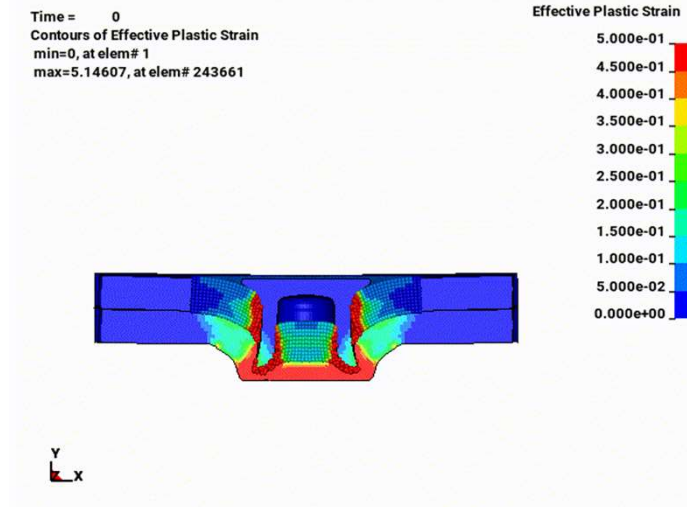
Animation of 2nd stage

Plastic strain at termination of 2nd stage

Summary for Two-Stage SPG Analysis

- **1st stage analysis**
  - *INTERFACE_SPRINGBACK_LSDYNA
    - Define a part set including all parts that are used in 2nd stage
  - *INTERFACE_SPG_1
  - All other keywords as for a regular FEM/SPG analysis

- **2nd stage analysis**
  - *INTERFACE_SPG_2
    - Copy "1234spg" from 1st stage and rename as "1234spg0"
  - Include "dynain" from 1st stage as an input file
  - All other keywords as for a regular FEM/SPG analysis

- **Availability**
  - Both SMP and MPP supported
    - A latest beta version (newer than commit 469ae95c0 on Dec. 18, 2020) should be used
  - 1st stage and 2nd stage are completely independent simulations

Ansys