T003: Decoupled Die Stress Analysis

Introduction

This technical note describes best practices for setting up a decoupled die stress analysis.

Problem Description

Customers often have questions about how to properly set up a decoupled die stress simulation. Failure to properly set up a die stress analysis may result in a lack of stresses, unconstrained parts or inaccurate results.

Background Information

There are several methods that can be used to model die stress in DEFORM.

Coupled die stress solves for the stresses in the tools at the same time as the deformation of the plastic workpiece. This yields tool stresses at each step of the forming simulation. The downside is that the simulation can take a long time to run since multiple deforming objects are involved.

The majority of die stress issues can be resolved using decoupled die stress, which is much faster and less computationally intense. Decoupled die stress solves for the stresses in the tools after the forming simulation has already been completed. A critical step from the forming simulation is identified and then stresses in the tools are studied at that step.

Solution

The following set-up steps are required for a decoupled die stress analysis. Helpful tips are also provided. These procedures are valid for both 2D and 3D simulations.

1. Run the forming simulation in the Integrated Manufacturing (MO) interface.

2. Identify critical step in the forming simulation.

- *Tip: The critical step is typically at the end of the stroke where the load is highest. To get accurate simulated load, do not completely fill corners of the part that do not get filled on the shop floor.*

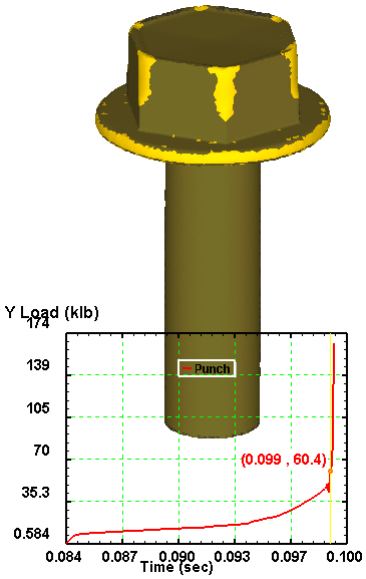
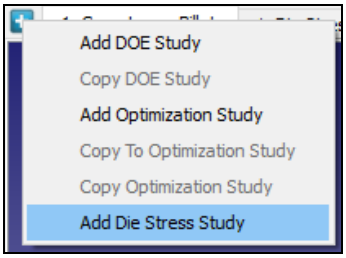
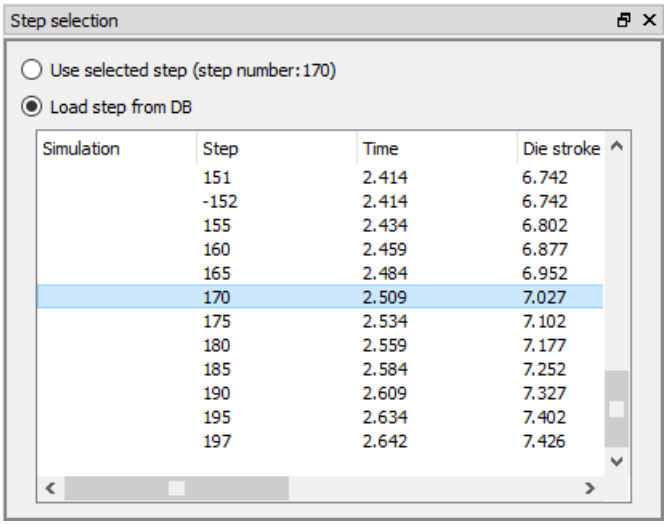


Figure 1: Correct step for die stress based on die fill (shaded area is die contact)

3. In the Operation Editor, click the  in the top left corner to add a Die Stress Study.



In the step selection dialog, select the critical step from the forming simulation.



4. Import support tools and mesh all elastic dies.

- *Tip: When meshing the tools, use enough elements so that the tools look realistic.*
- *Tip: When multiple tools are in contact, it is beneficial to mesh the inner object finer than the outer object.*

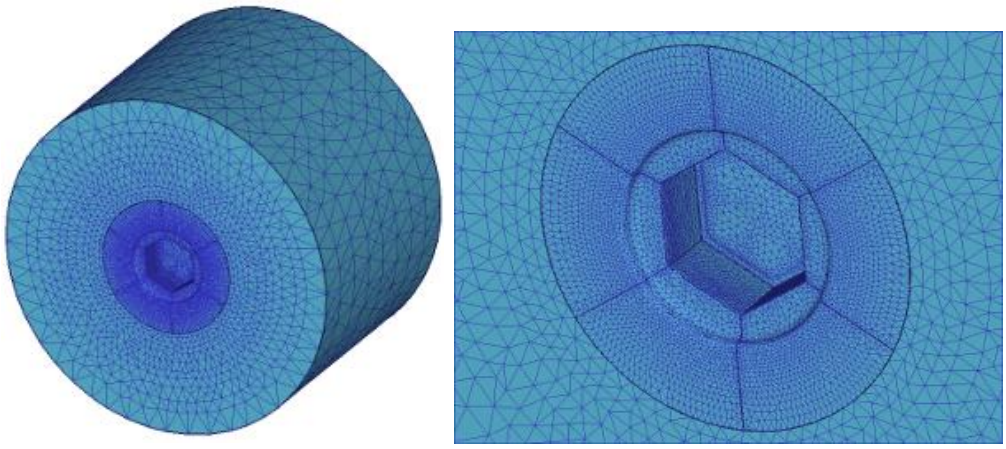


Figure 2: Segmented punch insert is shown - segments are meshed finer than the case.

5. Constrain tools, often by assigning velocity boundary conditions.

6. Interpolate forces from forming simulation workpiece to elastic tools.

- *Tip: Use the default interpolation tolerance of 0. This will allow the program to calculate this tolerance for you. The magnitude of the interpolated force should be very close to that seen in the load-stroke curve.*

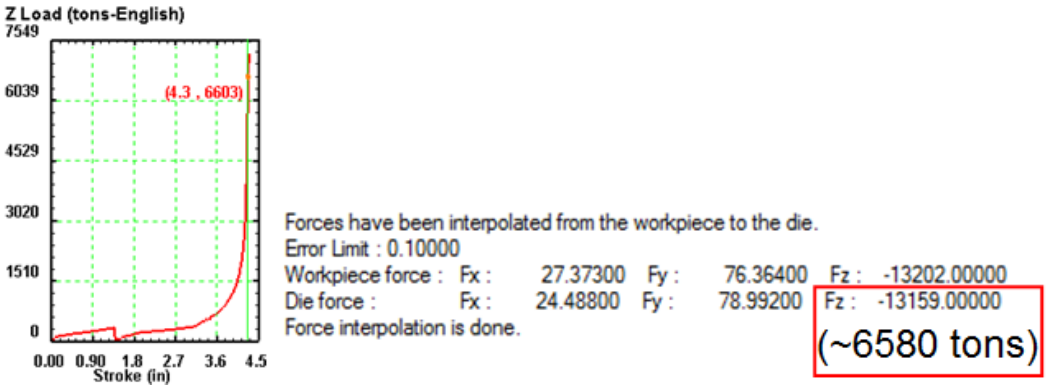


Figure 3: The load on the die in the forming simulation should be very close to the force interpolated onto the object in the die stress analysis.

7. Define shrink fits, if any, between the tool components.

- *Tip: A correctly oriented shrink fit gets applied as a positive number on the outer object.*
- *Tip: It is beneficial to run a simulation with only the applied shrink fit (no force interpolation) to investigate the stresses caused by the shrink fit.*

8. Assign tool master-slave relationships and generate contact.

- *Tip: For die stacks, make the outer object (coarser mesh) the Master to the inner Slave object (finer mesh).*
- *Tip: Try to generate complete contact at mating die surfaces. If needed, gradually increase the contact tolerance to accomplish this.*

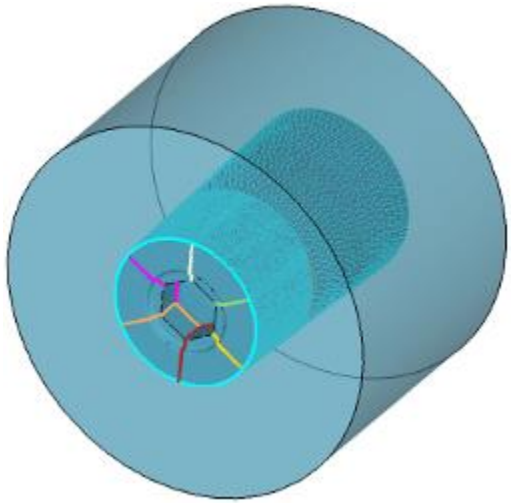


Figure 4: Punch assembly shown with complete contact at all mating surfaces.

9. Write a new die stress database and run the simulation.

- *Tip: Use one simulation step if modeling only one die component. For multiple contacting die components in 3D, use 10 steps so equilibrium is obtained. If modeling multiple contacting die components in 2D, just one step will suffice since 2D utilizes contact substepping by default.*

Related Information

An in-depth PowerPoint presentation regarding die stress analysis is provided. Details are given about the various modes of die failure, as well as how to use DEFORM to troubleshoot die cracking issues.