SORPAS® 3D – Case #4

Welding and testing
Peel-Strength

For more information

www.swantec.com

Support@swantec.com

How to read the document?

Bold: Position, which window is open and which category the action is.

Underline: Action

Created in SORPAS® 3D.welding+testing V5.00

Introductions:

There are a lot of different parameters to test to find out if the formed nugget is acceptable for a certain application. Very often, welding engineers investigate the parameters related to the mechanical properties of the nugget.

To measure the mechanical properties of the nugget, different mechanical tests are used such as tensile strength test, cross tension test, hardness measurements etc. These tests can help to evaluate if the weld nugget has an acceptable strength for the application.

SORPAS® 3D.testing can help to evaluate the weld strength based on simulations with actual welding and joining conditions. SORPAS® 3D.testing has also integrated standard templates for workpieces which makes it easier for the user to set up the welding design.

In this tutorial you will see how to make a welding simulation and how to make a Peel strength test on the welded workpieces.

This tutorial demonstrates how to build a model with one spot weld and test the weld strength of the nugget. It shows how to add testing tools to the workpieces. To demonstrate this case simple, no change of materials and process parameter is done, we use default settings.

The example provides an excellent overview of building a model and doing weld strength testing. It demonstrates how to build a model, adding contact interface and adding testing tools.

More information about Weld Strength Test is in the **help** menu, -> Weld Strength Test.

Model definitions:

The model's geometry is shown below. Due to symmetry conditions only half of the model is necessary to create, see Fig 1, which reduces the computation time.

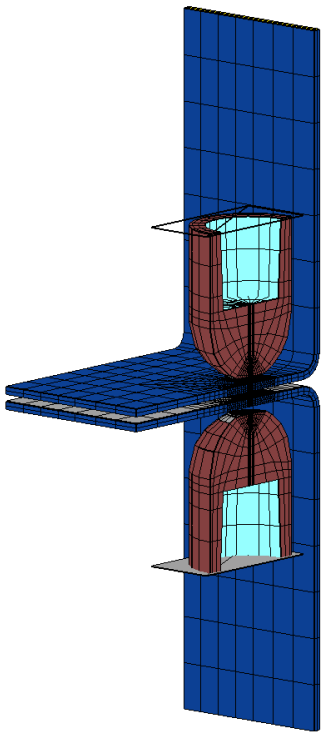


Fig.1 Model

Model instructions

Build/Define model

1) From the **Libraries** menu, click Workpiece library.
2) In the **Workpiece library** window, under **Category** select New-workpiece. *(If you haven't created a new category then under **workpiece category** click Add new. Name it New-workpiece and click Add category.)*
3) Under **Object within selected category**, click Add new.
4) In the **Add workpiece object** window, name the workpiece Peel-strength-sheet, select Define standard workpiece by geometric parameters and click Add object.
5) In the **Define standard workpiece by geometric parameters** window, select Peel test sheet with round mesh (4-2) and click Save to library.
6) In the **Workpiece library** window, click OK.
7) From the **File** menu, click new data file.
8) In the **New data file** window, under SORPAS3D/work folder, click Special cases. (If the folder doesn´t exist – then create it).
9) In the **Special cases** folder, name the file 2-sheets-Peel-strength-test and click Save.

10) In the **Input wizard step 1** window, select Welding simulation and weld strength testing and click Next.
11) In the **Input wizard step 2** window, click Add object.
12) In the **Add object** window, under **Select object from library**, select Workpiece library.
13) Under **Object category,** select New-workpiece and under **Object name**, select Peel-strength-sheet.
14) Under **Position** select Original object position and click Add object.
15) In the **Input wizard step 2** window, click Add object.
16) In the **Add object** window, under **Position** select Original object position.
17) under **Orientation**, select Flip in z and click Add object.
18) In the **Input wizard step 2** window, under **position object**, change **Translation** in z to 47mm and click Move- (Move minus).
19) Click Add object.
20) In the **Add object** window, under **Select object from library**, select Electrode library.
21) Under **Object category**, select Type F1 and under **Object name** select Type F1; d2=6,0, d1=16; R=40.
22) Under **Position**, select On top of all objects and click Add object.
23) In the **Input wizard step 2** window, under **position object**, click Mirror in yz-plane.
24) Click Add object.
25) In the **Add object** window, under **Position**, select Under bottom of all objects.
26) Under **Orientation**, select Flip in z (vertical) and click Add object.
27) In the **Input wizard step 2** window, under **position object**, click Mirror in yz-plane.

The model should look like on the Fig 1.1. To finished the model, contact interface must be added to all objects within contact with another object.

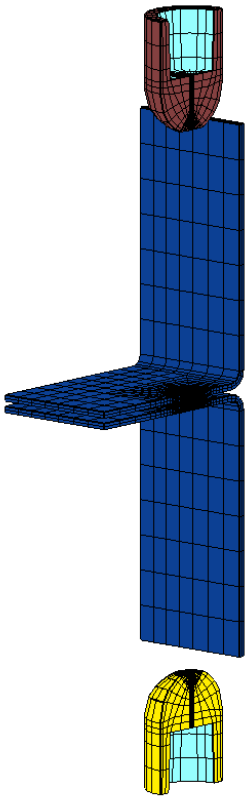


Fig.1.1 model with no contact interface

Adding Contact interface

1) Under **Range of faces**, change to 30 deg.
2) Under **Add thin layer for coating or interface**, under **Thickness**, change to 0.01mm and click Add upper layer.
3) Under **Object type and material** change sliding to sticking.
4) Under **Select object**, select **Active object** no.3
5) Under **Range of faces**, change to 30 deg
6) Under **Add thin layer for coating or interface**, under **Thickness**, change to 0.01mm and click Add lower layer.
7) Under **Object type and material** change sliding to sticking.
8) Under **Select object**, select **Active object** no.2
9) Under **Add thin layer for coating or interface**, click Add upper layer. Now the contact interface is added in the required places and the model building is finished, see Fig 1.2
10) In the **Input wizard step 2** window, click Next.

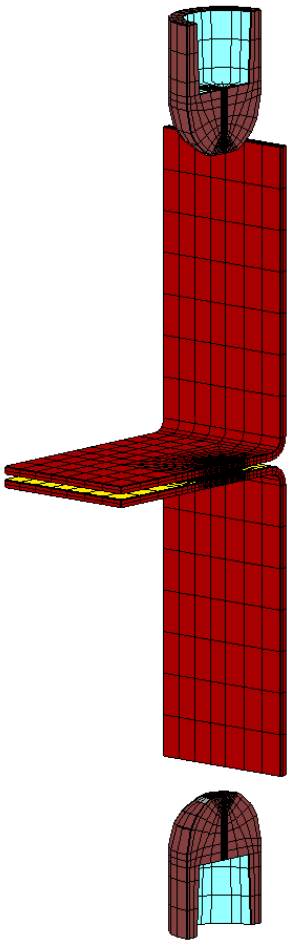


Fig.1.2 model with contact interface

Connecting to the machine

1) In the **Input wizard step 3** window, under **Add tool**, click Add to top and click Add to bottom.
2) Click Next.
3) In the **Input wizard step 4** window, under **Process time**, change **Off** to 1600ms or 80Cy.
4) Click Next.
5) In the **Input wizard step 5** window, under **Movement within active weld**, click Add movement.
6) Select **Electrodes** 4 and **Tools** 1.
7) In the **Translation** change z-direction to 5mm and click Add movement.
8) Select **Electrodes** 5 and **Tools** 2.
9) In the **Translation** change z-direction to -5mm and click Next.

Connecting to weld strength testing

1) In the **Input wizard step 6** window, under **Weld strength testing**, click Bottom (-) and Top (+).
2) Under **Tool velocity**, change **z** to 2 [mm/s] and at **Duration of test** change to 20000 [ms], see Fig 1.3
3) Click Start simulation…

Note: When starting simulation, the first step closes all gaps between objects, for example the electrodes move to the weld point before mechanical interaction.

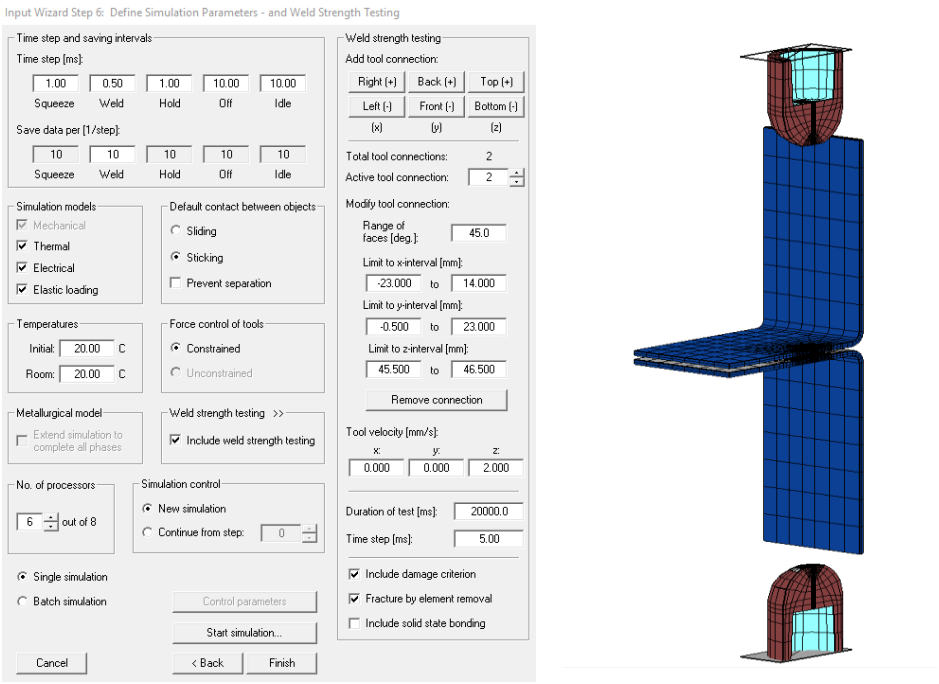


Fig.1.3 Starting simulation

Results

The results of the simulation are shown on the Fig 1.4.

1) From the **Results** menu, click Damage factor distribution.
2) From **View** menu, click Options.
3) In the **Options** window, under **Mirror**, click ZX-mirror and click OK.

The location with higher values of damage factor is the place where fracture would occur. On the Figure 1.4 you see that the fracture occurred in the sheet which results in a plug or button failure. The load-elongation curve of the weld strength test is displayed as well.

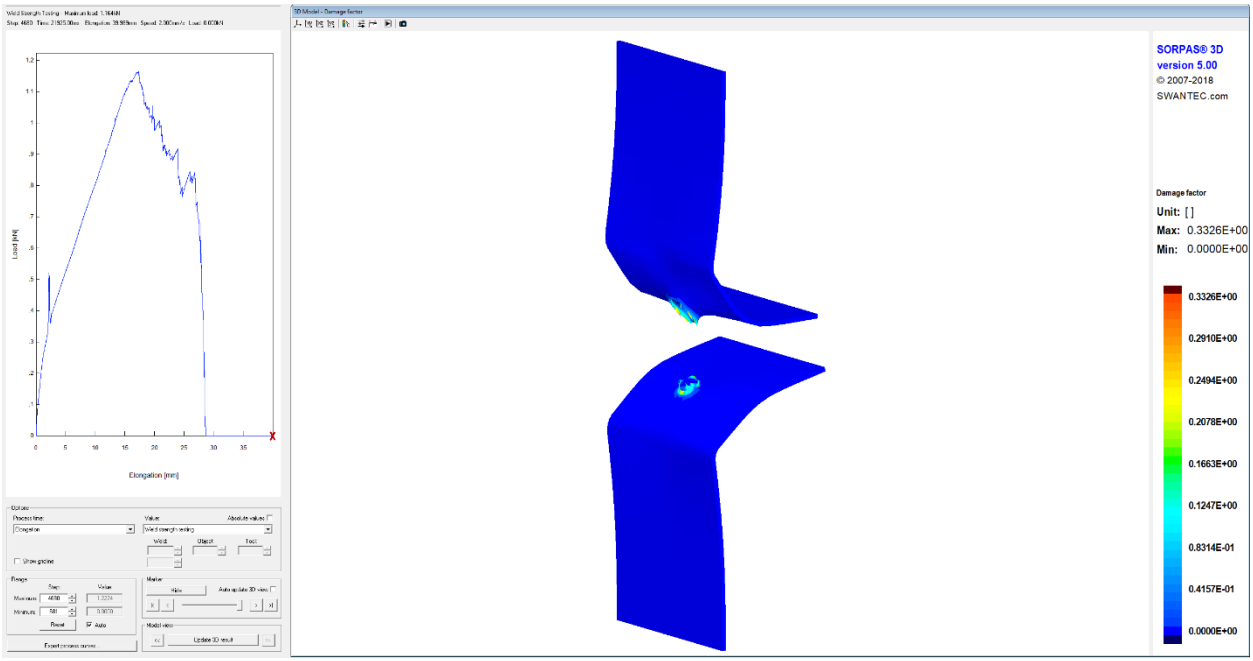


Figure.1.4 Results, Plug failure