SORPAS® 3D – Case #3

Testing Cross-Tension-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 V5.50

Introductions

There are lot of different parameters to test regarding if the formed nugget is acceptable for the certain application. Parameters welding engineers often investigates are 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 the weld nugget has an acceptable strength for the certain application. With SORPAS 3D testing it can help to evaluate the weld strength based on simulations with actual welding and joining conditions. SORPAS 3D testing have also integrating standard templates for workpieces which makes it easier for the user to set up the welding design. In this following tutorial it will show how to make a welding simulation and how to make a Cross tension 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 tool to the workpiece. 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 quarter of the model is necessary to create, see Fig 1, which reduces the computation time.

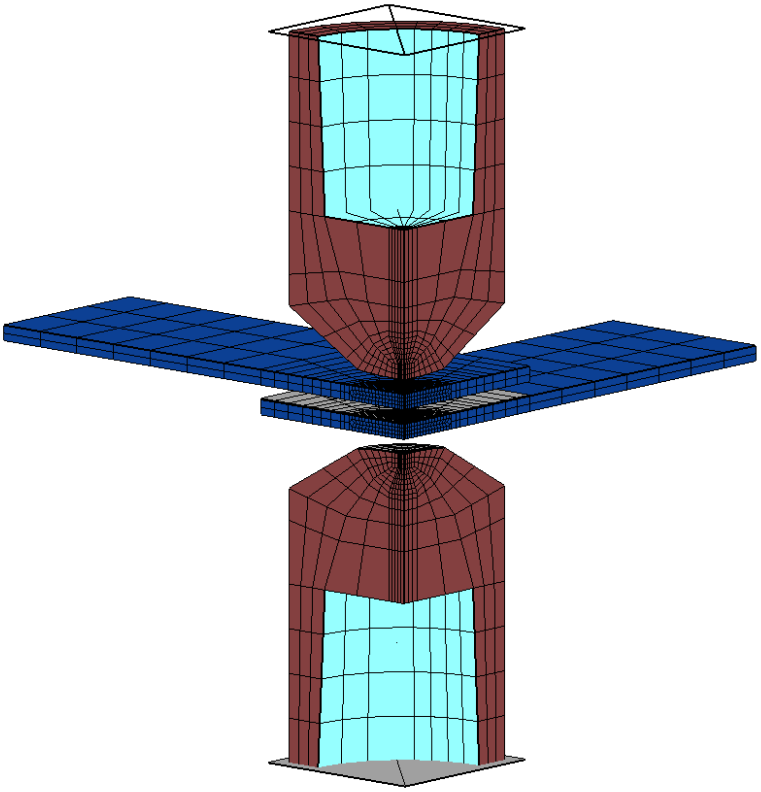


Fig.1.1 Model

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 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 Cross-tension-sheet, select Define standard workpiece by geometric parameters and click Add object.
5) In the **Define standard workpiece by geometric parameters** window, select Cross-tension 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-Cross-tension-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 Cross-tension-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 On top of all objects**,** under **Orientation**, select Flip in y and click Add object.
17) In the **Input wizard step 2** window, under **position object**, change **Rotation** in z-axis to 90 deg change **Translation** in x-direction to 12.5mm and click Move+ (Move plus).
18) Click Add object.
19) In the **Add object** window, under **Select object from library**, select Electrode library.
20) Under **Object category**, select Type B0 and under **Object name** select Type B0; d2=8, d1=20; R=50 – ISO 5821;2009.
21) Under **Position**, select on top of all objects and click Add object.
22) In the **Input wizard step 2** window, click Add object.
23) In the **Add object** window, under **Position**, select Under bottom of all objects.
24) Under **Orientation**, select Flip in z (vertical) and click Add object.
25) Under symmetry conditions, select yz and zx-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.

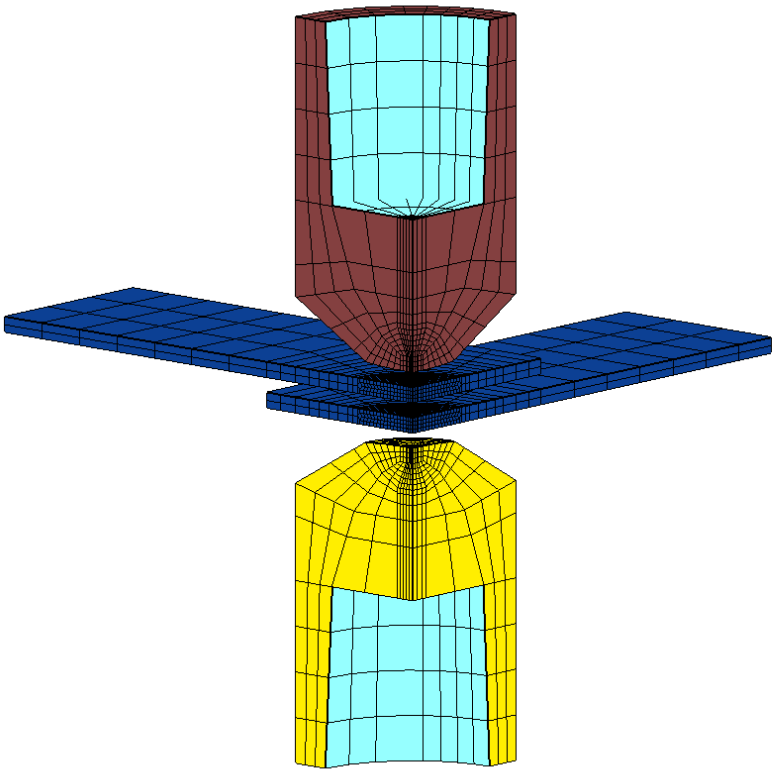
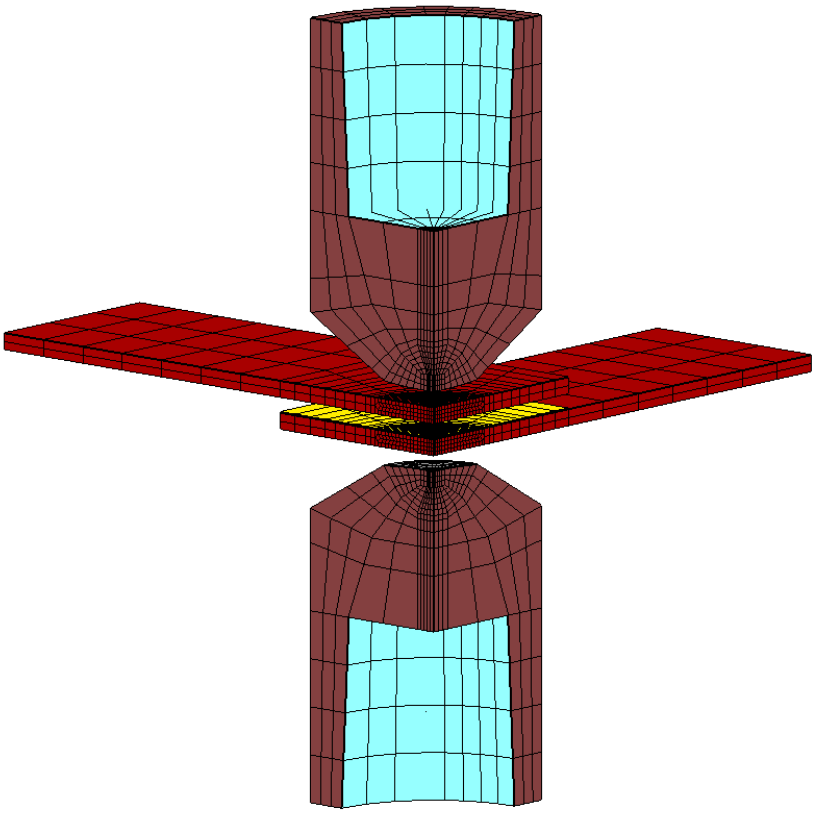


*Fig.1**Error! No text of specified style in document.**.1 model with no contact interface*

Adding Contact interface

1) Under **Add thin layer for coating or interface**, under **Range of faces**, change to 30deg.
2) Under **Thickness**, change to 0.01mm and click **Limit layer to z-interval**.
3) Under **Limit layer to z-interval**, change interval from -2.0mm to -1.0mm and click Add upper layer.
4) Under **Object type and material** change sliding to sticking.
5) Under **Select object,** select Electrode [3]
6) Under **Add thin layer for coating or interface**, under **Range of faces**, change to 30deg.
7) Under **Thickness**, change to 0.01mm and **Limit layer to z-interval**.
8) Under **Limit layer to z-interval**, change interval from 4.0mm to 5.0mm and click Add lower layer.
9) Under **Object type and material** change sliding to sticking.
10) Under **Select object**, select Workpiece [1]
11) Under **Add thin layer for coating or interface**, click Limit layer to x-interval.
12) Change **x**-interval from 0 to 12.5mm and click Add upper layer. Now the contact interface is added in the necessary places and the model building is finished, see fig 1.3
13) In the **Input wizard step 2** window, click Next.



14) *Fig.1**Error! No text of specified style in document.**..2 model with no contact interface*

Connecting to the machine

1) In the I**nput wizard step 3** window, under Tool list, 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.
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.
10) In the **Input wizard step 6** window, under **Time step,** change weld to 0.50ms.
11) Under **Default contact between objects,** select sticking.
12) Click Next.

Connecting to weld strength testing

1) In the **Input wizard step 7** window, under **Add tool connection**, click Right (+) and Back (+).
2) Under **Tool movement**, change Velocity z to 2 [mm/s] and under **Simulation parameters** in **Duration of test** change to 20000 [ms].
3) Click Start simulation…

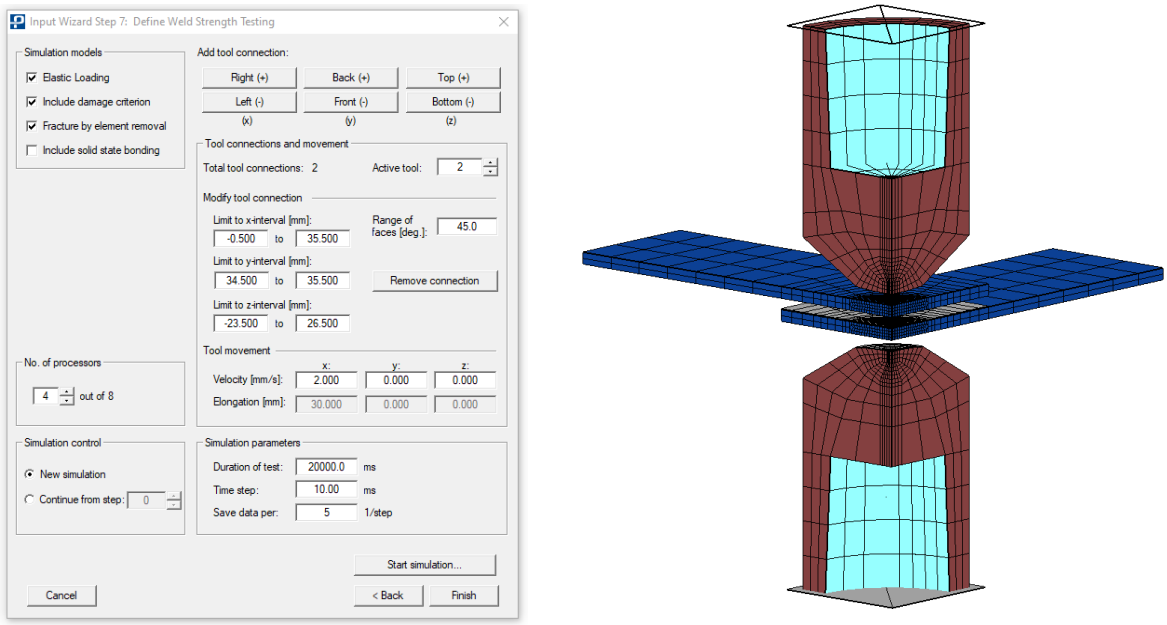
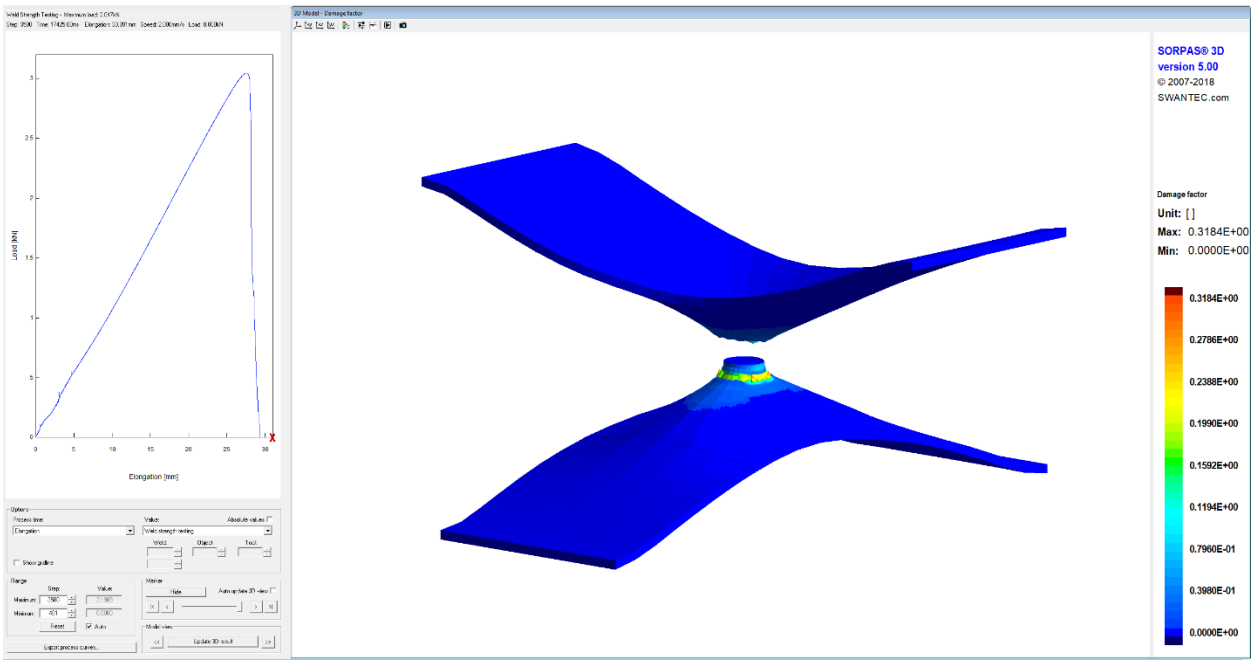


Fig.1.3 Starting simulation

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.



1.4 Results. On the left Load-elongation curve of the cross-tension test.