SORPAS® 3D – Case #1

Creating a 2-sheet spot weld model

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

This tutorial demonstrates how to build a model with one spot weld. It shows how to build a 2 sheets model. 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. It demonstrates how to build a model and adding contact interface.

More information about build/define model is in the **help** menu, -> Input wizard -> Step 2: Define model.

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.

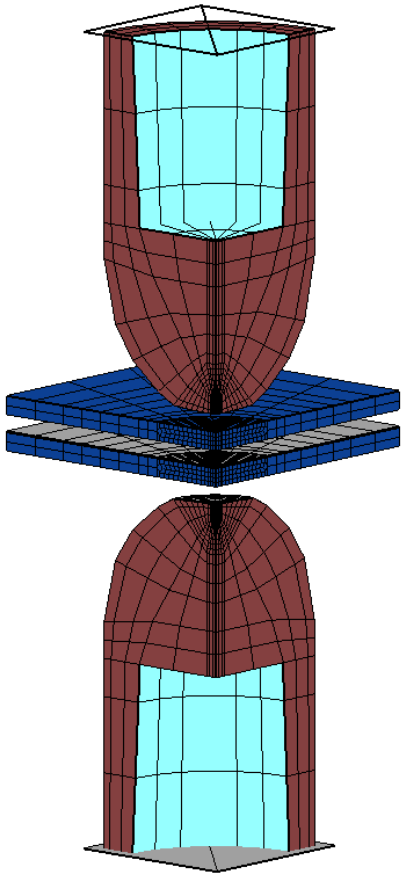


Fig.1 Model

Model instructions

Build/Define model

1) From the **File** menu, click New data file.
2) In the **New data file** window, under SORPAS3D/work/folder, click Special cases. (If the folder doesn´t exist – then create it).
3) In the **Special cases** folder, name the file 2-sheet-spot-welding and click Save.
4) In the **Input wizard step 1** window, select Welding simulation only and click Next.
5) In the **Input wizard step 2** window, click Add object
6) In the **Add object** window, under **Select object from library**, select Workpiece library.
7) Under **Object category,** select Flat-sheet_round-mesh and under **Object name**, select 15mm x 15mm x 1.0mm.
8) Under **Position** select Original object position and click Add object.
9) In the **Input wizard step 2** window, click Add object.
10) In the **Add object** window, click Add object.
11) Click Add object.
12) In the **Add object** window, under **Select object from library**, select Electrode library.
13) Under **Object category**, select Type F1 and under **Object name** select Type F1; d2=6,0, d1=16; R=50 – ISO 5821;2009.
14) Under **Position**, select on top of all objects and click Add object.
15) In the **Input wizard step 2** window, click Add object.
16) In the **Add object** window, under **Position**, select Under bottom of all objects.
17) Under **Orientation**, select Flip in z (vertical) and click Add object.

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.

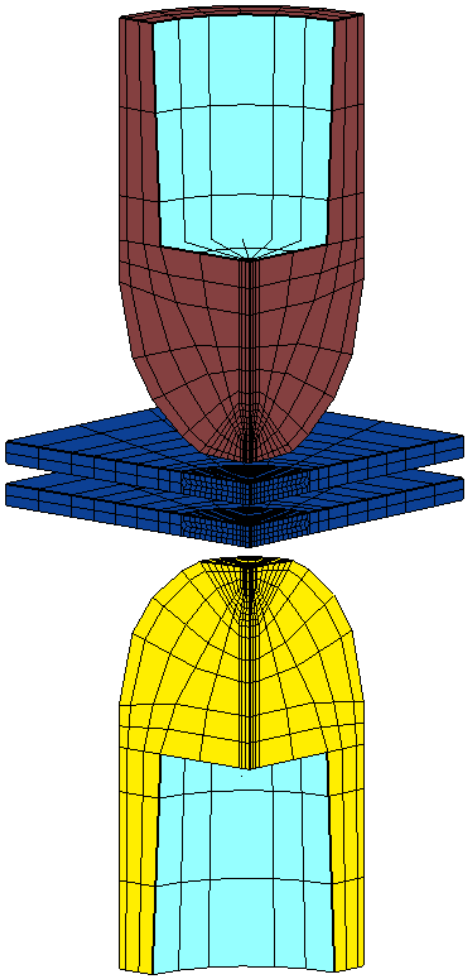
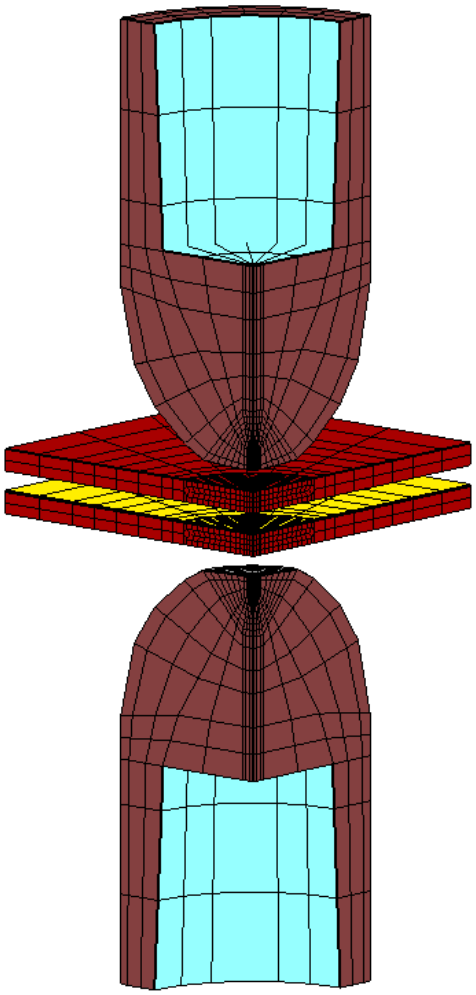


Fig.1.1 model with no contact interface

Adding Contact interface

1) Under **Add thin layer for coating or interface**, under **Range of faces**, change to 30 deg.
2) Under **Thickness**, change to 0.01mm and click Add upper layer.
3) Under **Select object**, select Electrode [3]
4) Under **Add thin layer for coating or interface**, under **Range of faces**, change to 30 deg.
5) Under **Thickness**, change to 0.01mm and click Add lower layer.
6) Under **Select object**, select Workpiece [1]
7) Under **Add thin layer for coating or interface**, click Add upper layer. Now the contact interface is added in the necessary places and the model building is finished, see fig 1.3
8) In the **Input wizard step 2** window, click Next.



9) Fig.1.2 model with contact interface

Connecting to the machine

1) In the **Input wizard step 3** window, under **Machine tool setting**, click Add to top and click Add to bottom.
2) click Next.
3) In the **Input wizard step 4** window, click Next.
4) In the **Input wizard step 5** window, under **Movement within active weld**, click Add movement.
5) Select **Electrodes** 4 and **Tools** 1.
6) In the **Translation** change **z**-direction to 5mm and click Add movement.
7) Select **Electrodes** 5 and **Tools** 2.
8) In the **Translation** change **z**-direction to -5mm and click Next.

Running the simulation

1) In the **Input wizard step 6** window, click Start simulation. For this simulation default settings were used as seen on the grey box see Fig 1.3.

Results of the simulation is shown on Fig 1.4.

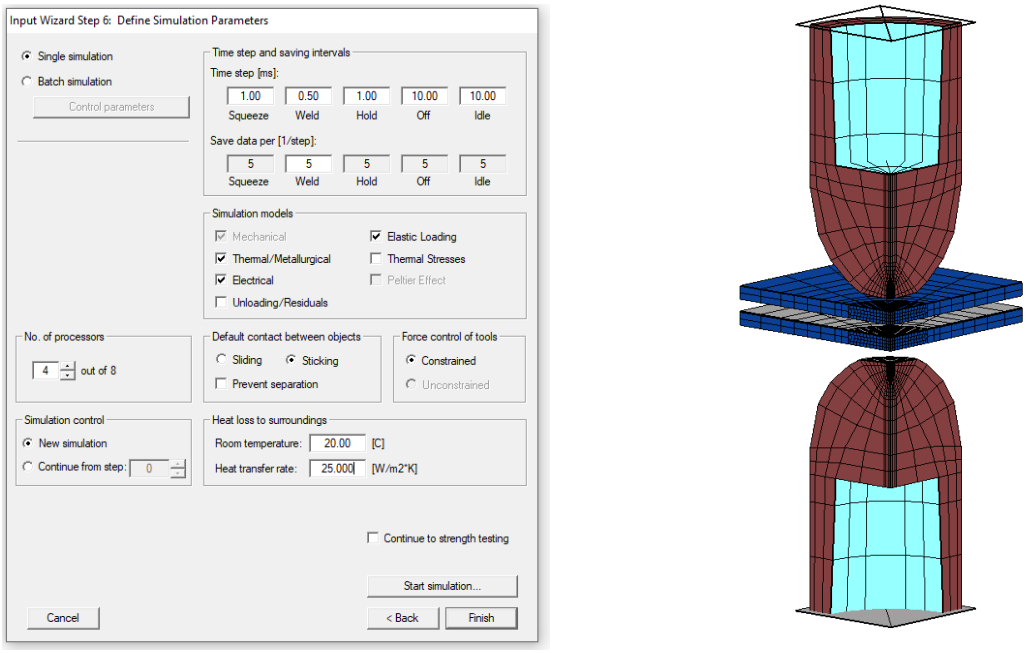


Fig.1.3 Starting simulation

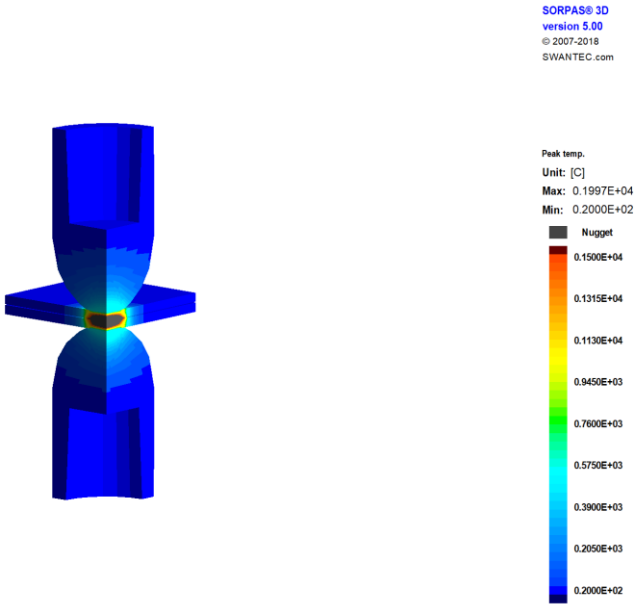


Fig.1.4 Peak temperature