ANSYS®
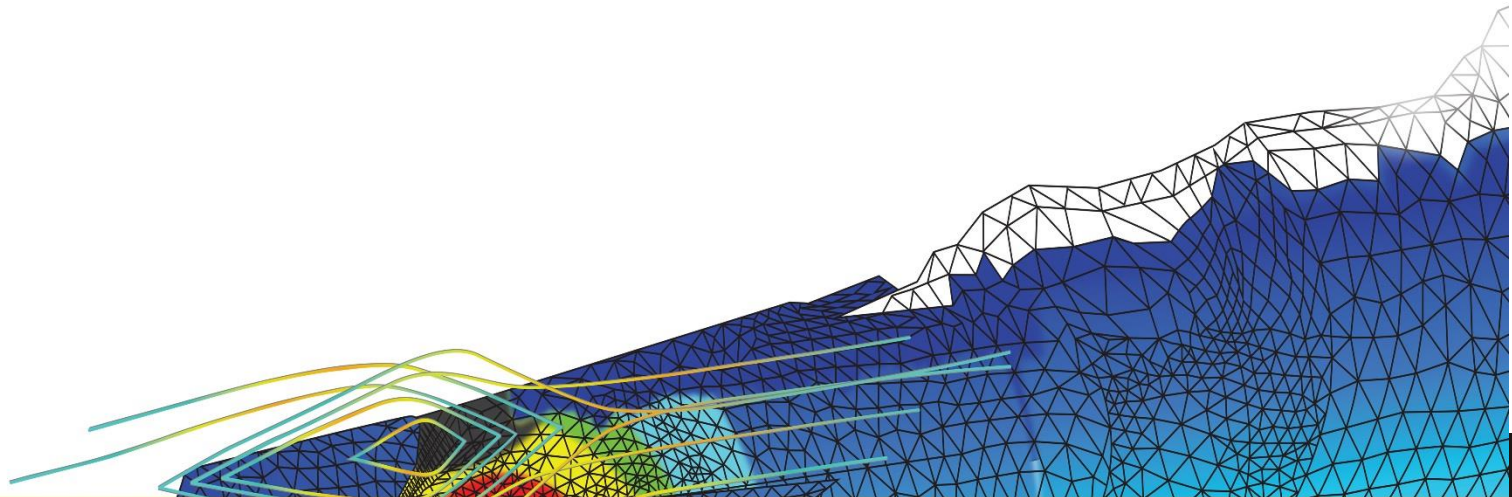
Welding Distortion Wizard

abel.ramos@ansys.com

Copyright and Trademark Information

Copyright and Trademark Information

Disclaimer Notice

ANSYS®

Welding Distortion - Overview

- Welding Distortion:

- Target application: Mechanical

- Description: Predict distortion in metal components connected by welding toes.

The version of the App and the supported versions of ANSYS are the ones indicated on the App Store.

Copyright and Trademark Information

© 2018 ANSYS, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS®

ACT App Store

Welcome to the
ANSYS App Store

- *https://appstore.ansys.com/shop/ACTApps_act%20apps*

- Great place to get started
  - A library of helpful applications available to any ANSYS customer
  - New apps added regularly
  - Applications made available in either binary format (.wbex file) or binary plus scripted format (Python and XML files)
  - Scripted extensions are great examples
  - Documentation and training materials available on the ANSYS Customer Portal: https://support.ansys.com/AnsysCustomerPortal/en_us/Downloads/ACT+Resources

Information

- Please pay attention to paragraph 9 of the CLICKWRAP SOFTWARE LICENSE AGREEMENT FOR ACS EXTENSIONS regarding TECHNICAL ENHANCEMENTS AND CUSTOMER SUPPORT (TECS): "TECS is not included with the Program(s)"

- Report any issue or provide feedback related to this app please contact:

  Contact email address: abel.ramos@ansys.com
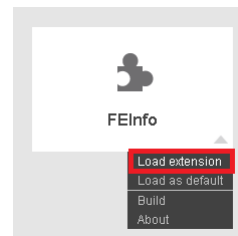
Binary App Installation (1)

Installing from the ACT Start Page:

1. From the project page, select the "*ACT Start Page*" option
2. Click on "Extension Manager"
3. Press "+" symbol in the top right corner
4. It will open a file dialog to select the appropriate "*.wbex*" binary file
5. The extension is installed
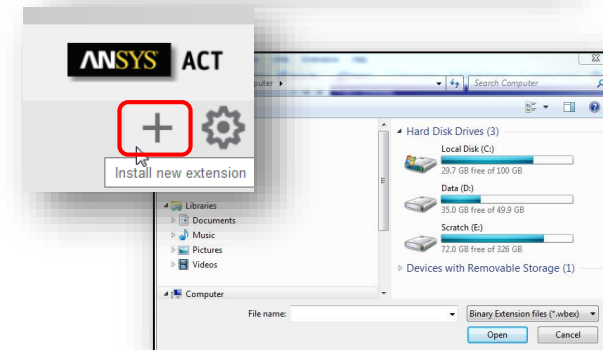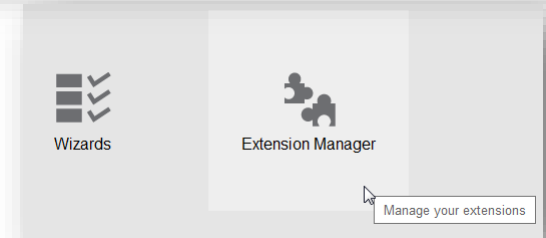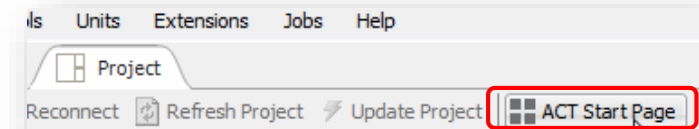
Loading the extension:

1. From the Extension Manager, click on your extension and choose 'Load Extension'
2. The extension is loaded

Notes:
- The extension to be installed will be stored in the following location: %AppData%\Ansys\[version]\ACT\extensions
- The installation will create a folder in this location, in addition to the .wbex file
- Example for [version]: v180

ANSYS

Binary App Installation (2)

Installing from the Extensions menu:

1. From the Extensions menu, select the "*Install Extension…*" option

2. It will open a file dialog to select the appropriate "*.wbex*" binary file

3. Click "Open" to install the extension

Loading the extension:

1. From the Extension Manager,
   click on your extension and choose 'Load Extension'
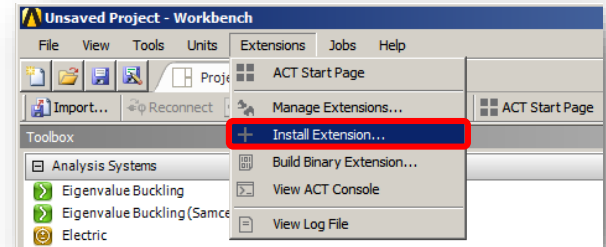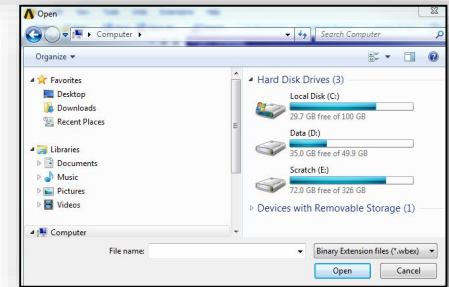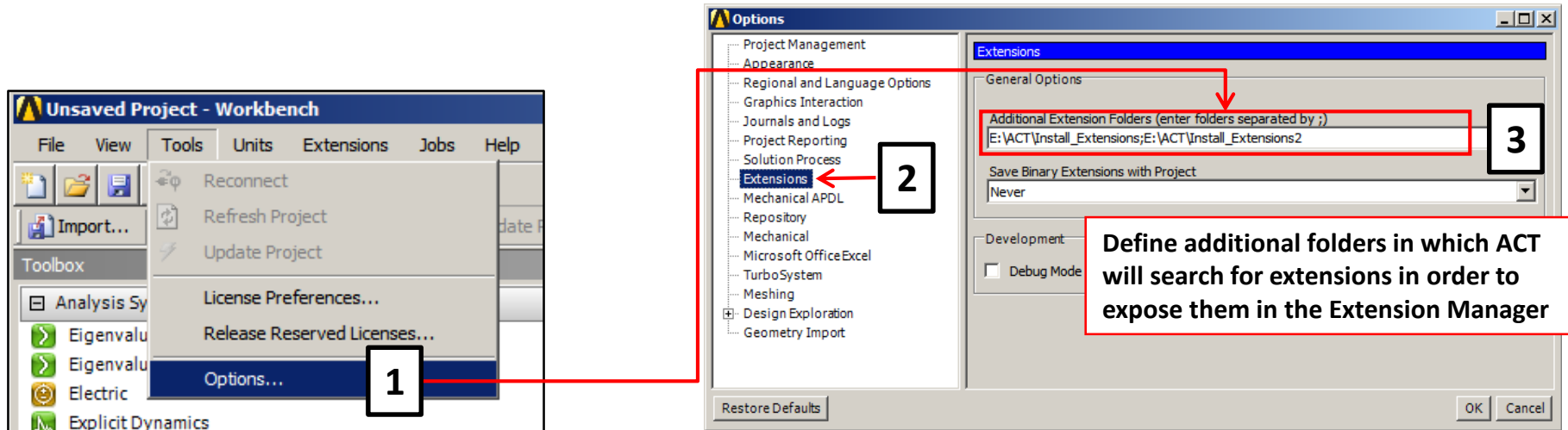
2. The extension is loaded

Notes:
- The extension to be installed will be stored in the following location: %AppData%\Ansys\[version]\ACT\extensions
- The installation will create a folder in this location, in addition to the .wbex file

ANSYS®

Binary App Installation (3)

- Once the binary extension is installed at default location, one can <u>move the *.wbex and the folder</u> to any other location
  - Default path: **%AppData%\Ansys\[version]\ACT\extensions**
  - New path: Any location on your machine, shared drive etc.

- All users interested in using the extension need to include that path in their Workbench Options
  - In the **"Tools"** menu, select the **"Options…"**
  - Select **"Extensions"** in the pop up panel
  - Add the path under **"Additional Extensions Folder …"**

Define additional folders in which ACT will search for extensions in order to expose them in the Extension Manager

Notes:
- During the scan of the available extensions, the folders will be analyzed according to the following order:
  1. The application data folder(e.g. *%AppData%\Ansys\[version]\ACT\extensions)*
  2. The additional folders defined in the "Additional Extension Folders" property
  3. The installation folder
  4. The "extensions" folder part of the current Workbench project (if the project was previously saved with the extension)
- If an extension is available in more than one of these locations, the 1st one according to the scan order is used

Documentation

The objective of this wizard is to automate the set up creation to simulate the welding process in ANSYS Mechanical in order to predict welding distortion
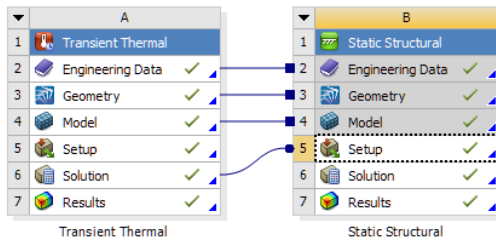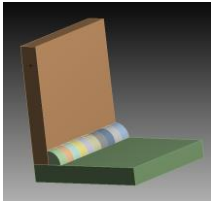

Assumptions:

- Weld geometry will be manually prepared on SpaceClaim
- Mechanical Script will automatically create all the contacts for the welding, the analysis settings time stepping and ekill/ealive controls needed
- Weld is assumed to be deposited at a user defined temperature and cooled down to room temperature
- Material Properties defined by user on Engineering Data. Properties dependant on Temperature, but not phase changes defined.

ANSYS®
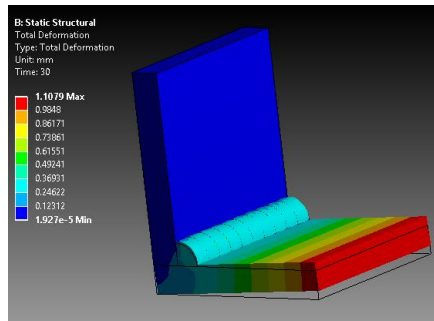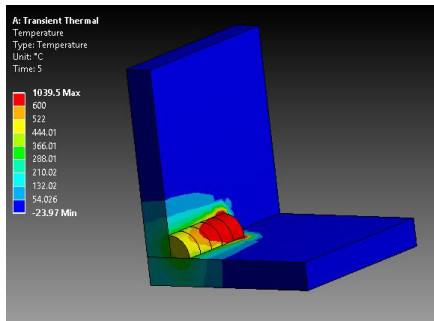
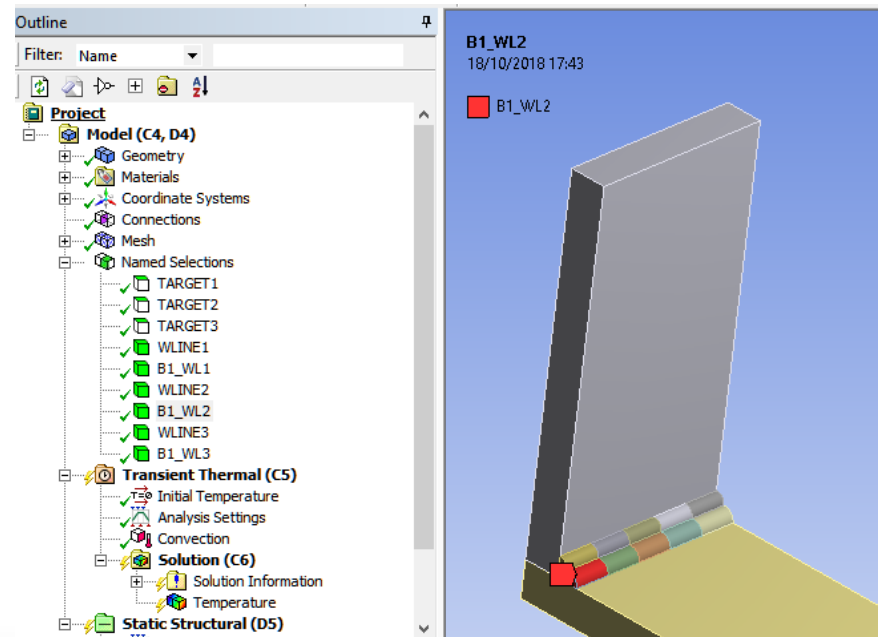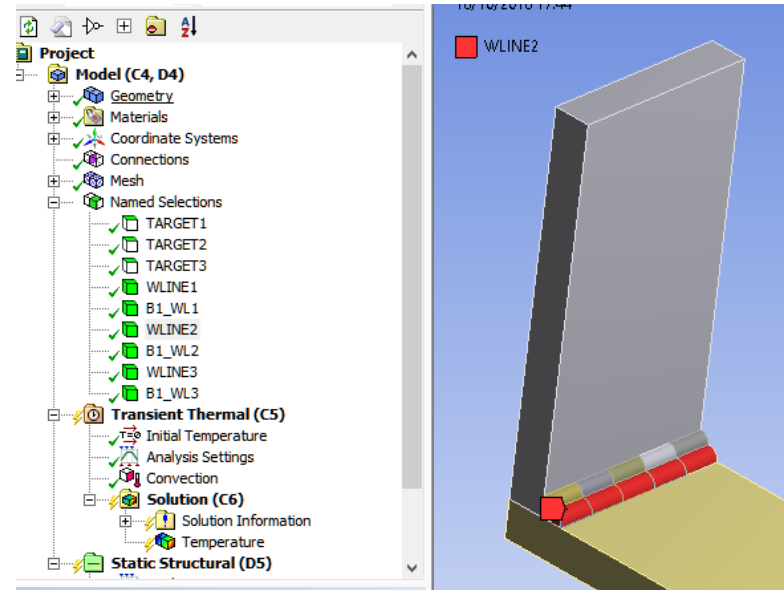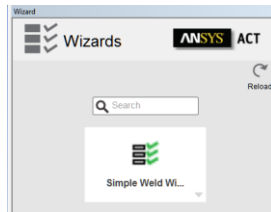# Simulating welding process - Wizard
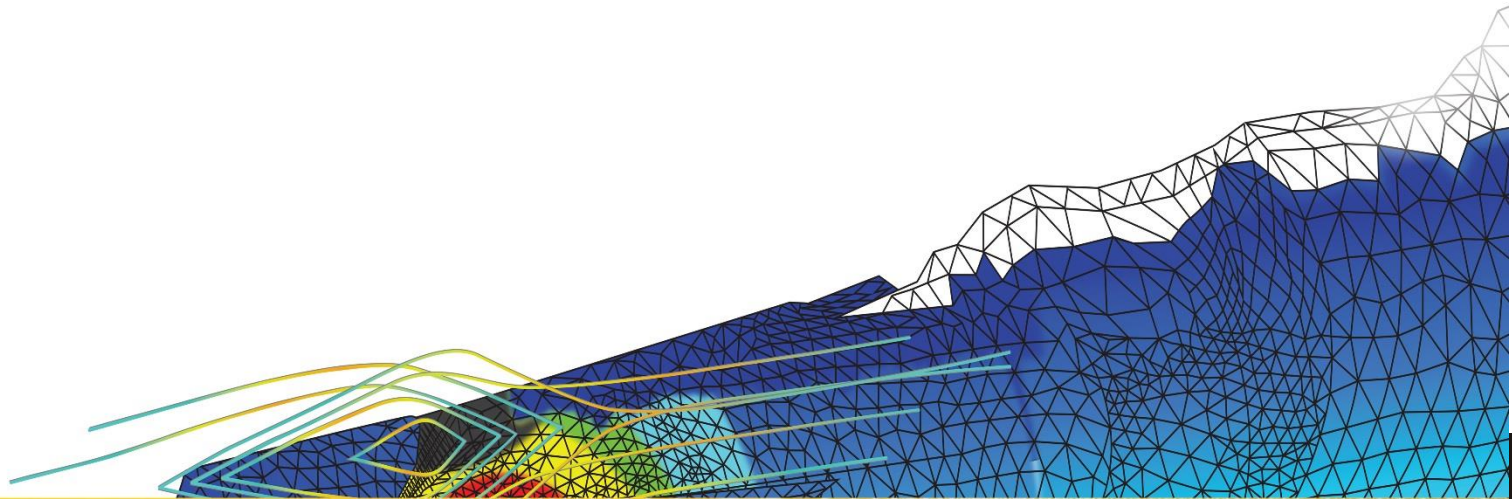
Wizard

**Name Selection Automatic Creation
If WELD1, WELD2, …. , WELD3 wants
to be automatically created, you can
use two name selections**

- **WLINEi: Cointaining all the bodies
  in the welding line**
- **B1_WLi: Defining the first body in
  the welding line**

**Where "i" corresponds to the number
of the welding line 1,2,3….,n**

Wizard

Install: S_WeldWizard.wbex

Simple Weld Wizard — ANSYS ACT

Mechanical
Model Setup:
Welding
Process

Field	Value	Unit
NS 1-Skip / 0-Create	Input required.	
N welding toes	Input required.	
N of welding bodies on…	Tabular Data [Edit] — Input required.	
Total welding time	Input required.	sec
Total cooling time	Input required.	sec
Melting Temperature	Input required.	C
0-Contacts/1-Share topo	Input required.	

- **Automatic Creation of NS:WELD1, WELD2, …. , WELD3**
- **Number of welding lines**
- **Table with the number of bodies on each welding line**
- **Total welding time**
- **Total cooling time**
- **Reference temperature**
- **Create contacts (0) or use Shared topolgy (1)**

Help

- Static Structural (B5)
  - Analysis Settings
  - Imported Load (A6)
    - Imported Body Temp…
  - Element Birth and Death
  - Element Birth and Death 2
  - Element Birth and Death 3
  - Element Birth and Death 4

Simulating welding process

Workflow
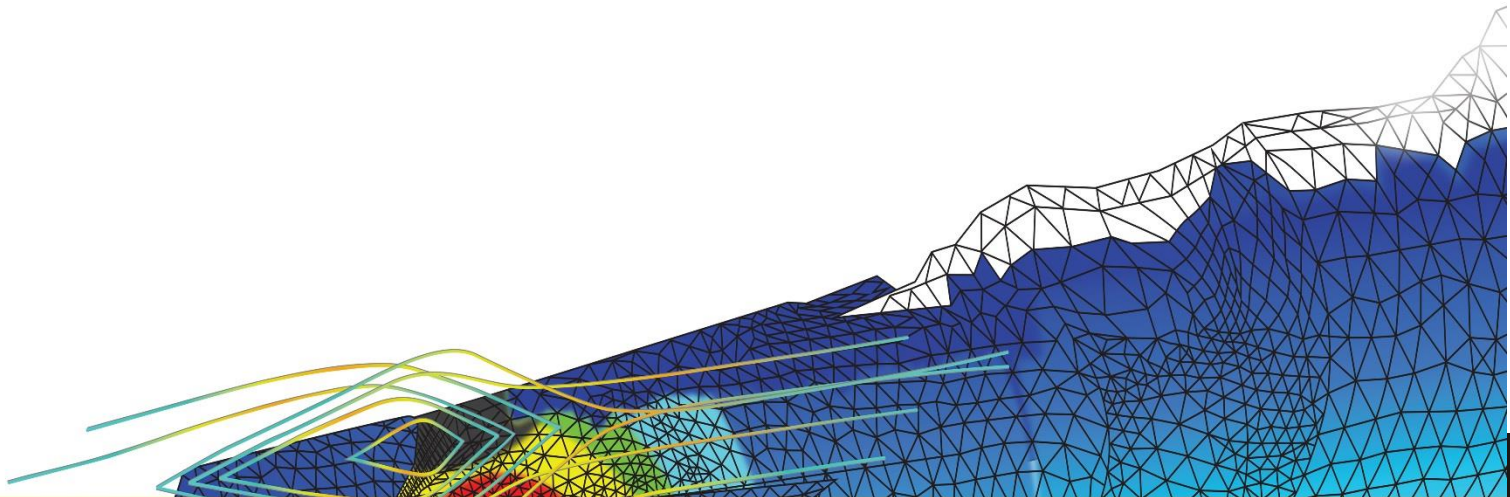
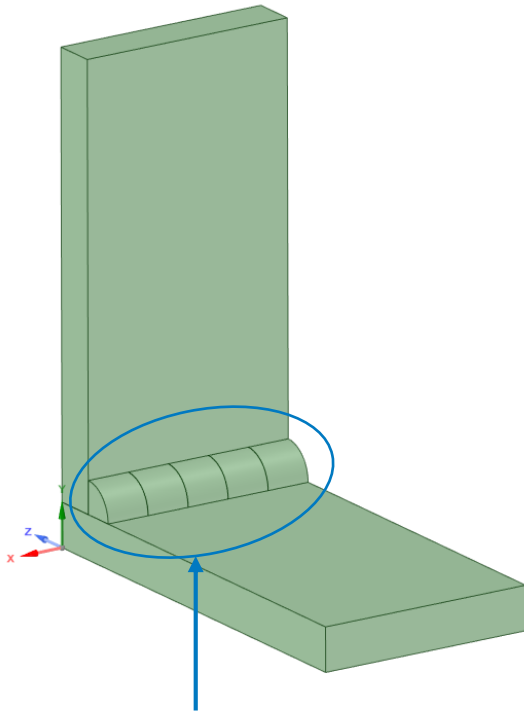# Welding Process Project Example
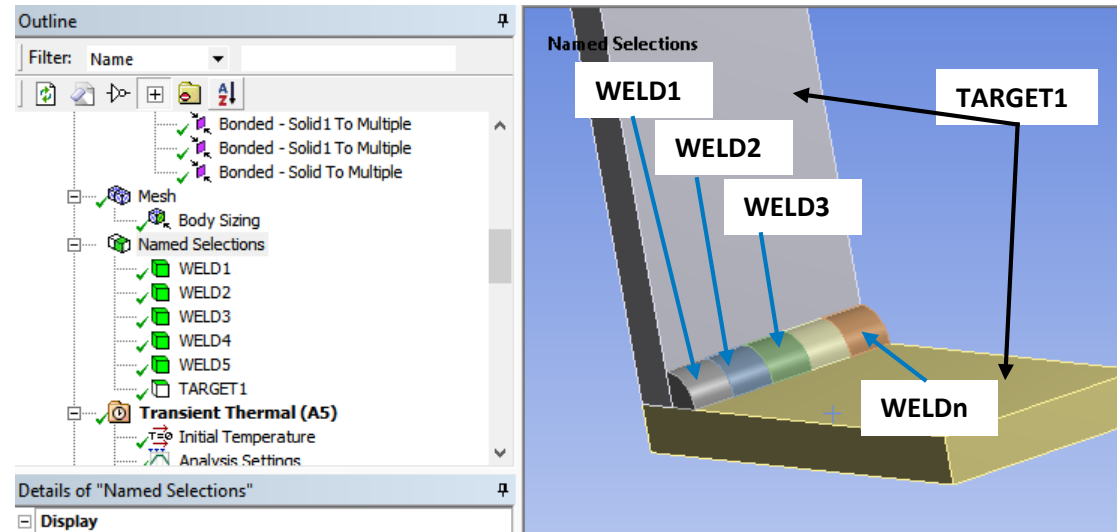
| Geometry Preparation | Material Properties | Meshing | Mechanical Set Up | Post-Processing |

Create welding geometry on SpaceClaim and divide it in "n" bodies depending on resolution required during simulation. Use share topology to obtain a conformal mesh in the weld

User needs to define manually the order on which each body will be alive in the simulation based on a Name Selection, starting form WELD1, WELD2, … up to WELDn
If you are not using shared topology, contacts will be automatically created. In this case, for each weld, some targets faces need to be defined using a Named Selection: TARGET1, TARGET2, .. TARGETm

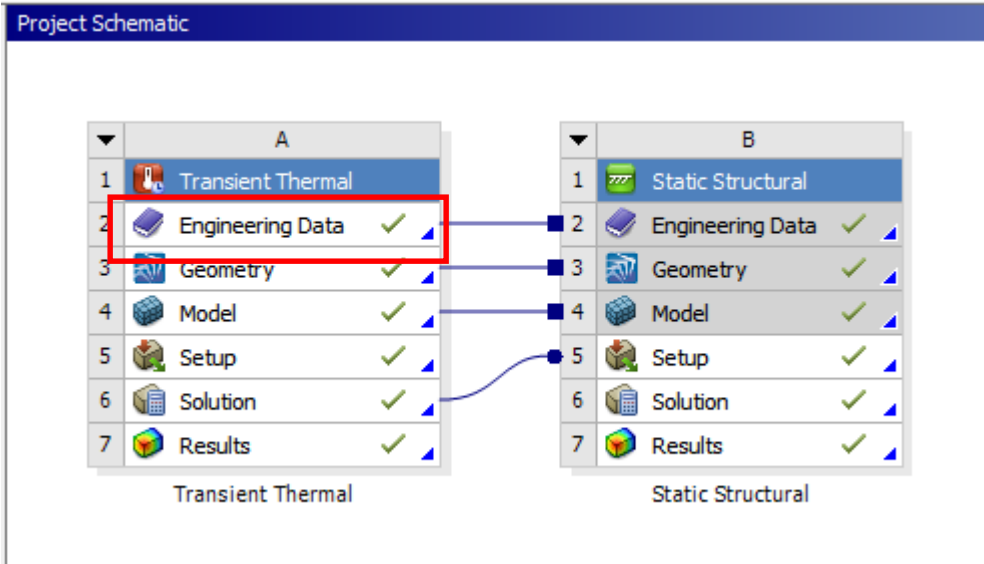# Welding Process Project Example

| Geometry Preparation | Material Properties | Meshing | Mechanical Set Up | Post-Processing |

User can use standard Engineering data to define the material properties used in the simulation

Welding Process Project Example

Geometry Preparation	Material Properties	Meshing	Mechanical Set Up	Post-Processing

- **Automatic Creation of NS:WELD1, WELD2, …. , WELD3. If 1, you need to use WLINEi, B1_WLi as explained on slide 13**
- **Number of welding lines**
- **Table with the number of bodies on each welding line**
- **Total welding time**
- **Total cooling time**
- **Reference temperature**
- **Create contacts (0) or use Shared topolgy (1)**

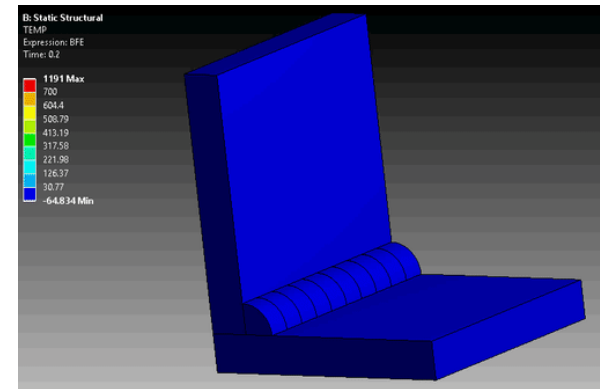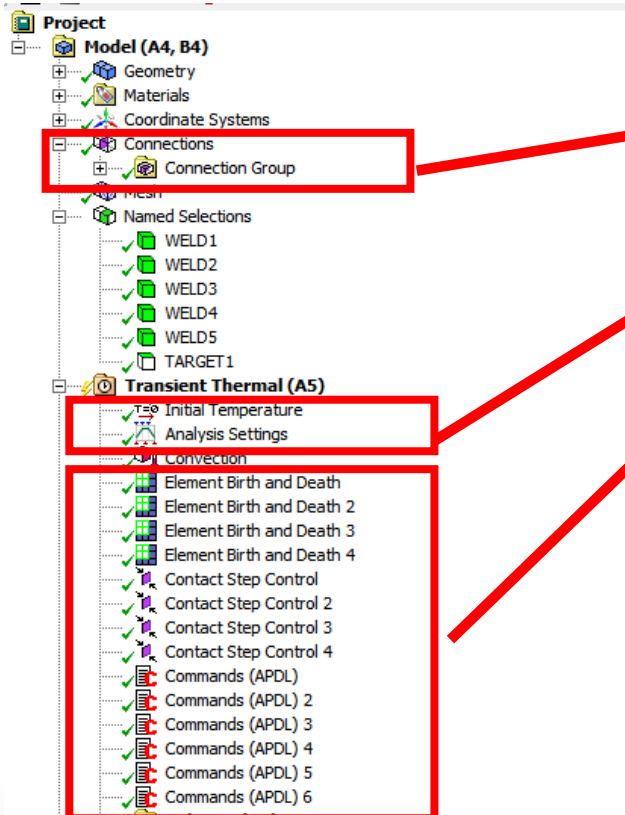# Welding Process Script Example

Geometry Preparation	Material Properties	Meshing	Mechanical Set Up	Post-Processing

Automatically created with the wizard

User can input additional boundary conditions and contacts in the tree as needed
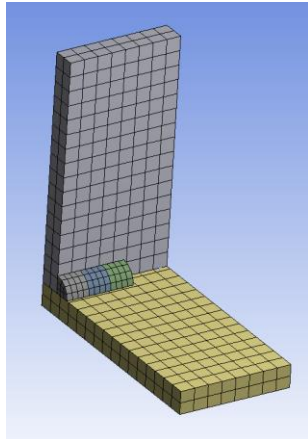
ANSYS®

Welding Process Script Example

Geometry Preparation	Material Properties	Meshing	Mechanical Set Up	Post-Processing

**Procedure**

- The number of total Load Steps Automatically defined is the number of welding bodies plus one additional to cool the model down
- The time defined for each WELDn body is TWtime/(number of welding bodies)
- Each new WELDn body is activated at T=Melt Temperature defined by the user. Temp is maintained during the load step and is freed to cool down in the nexst load step

Example: Load Step 3
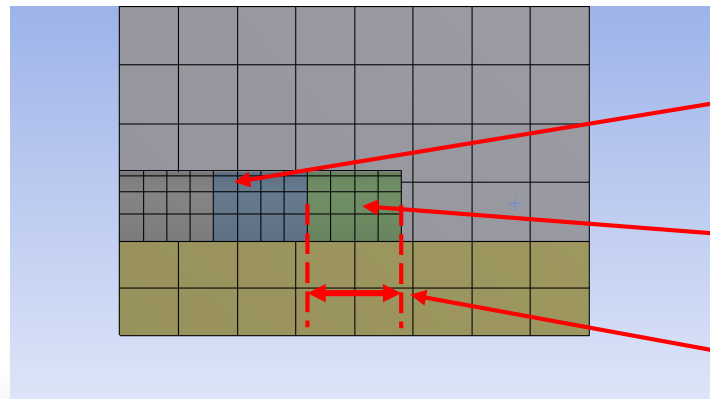
Temp = Cooling down depending on B.C.

Temp = Melt_temp [ºC]

Time = Twtime/5

Welding Process Script Example

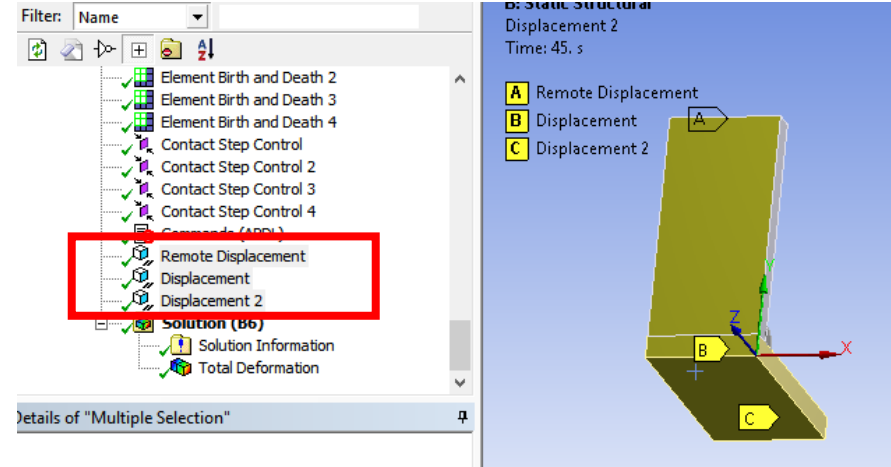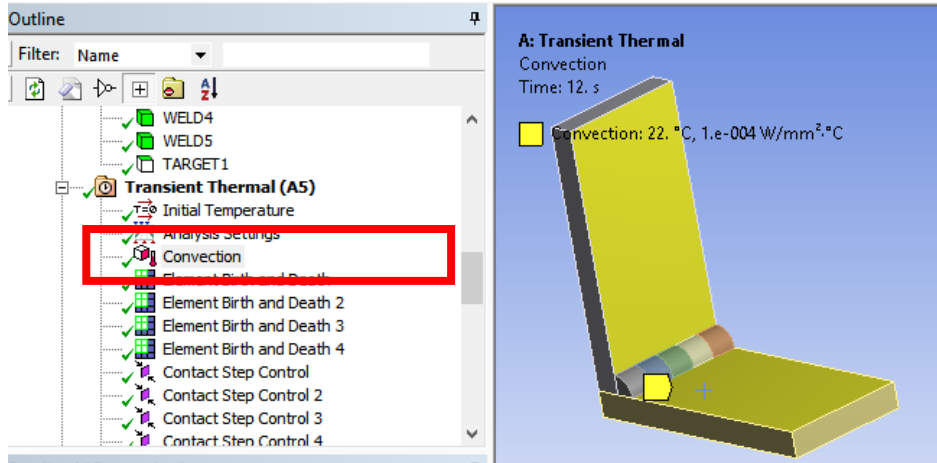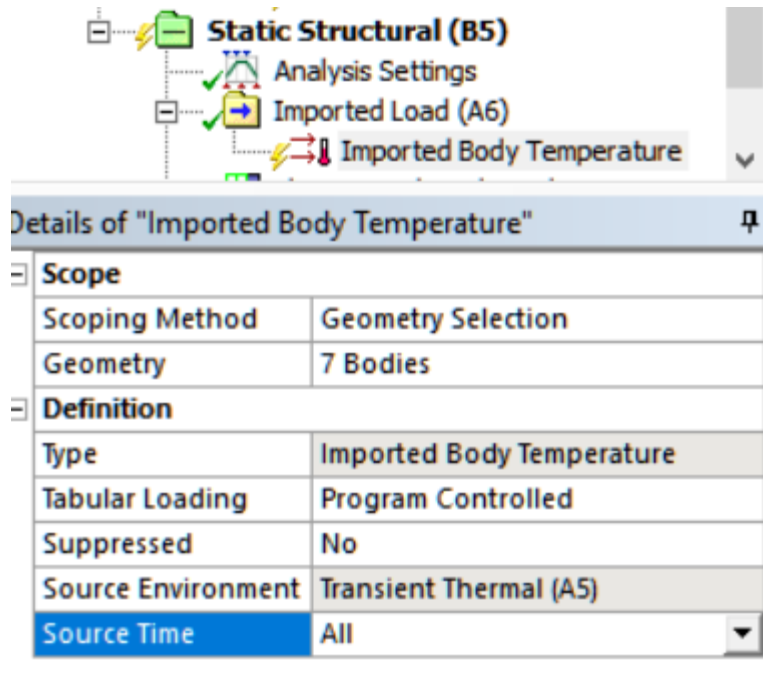| Geometry Preparation	Material Properties	Meshing	Mechanical Set Up	Post-Processing

Example additional boundary condition suggested

Thermal: Convection

Structural: Displacements

Displacements can be deactivated in the last Load Step to calculate spring back due to thermal loads

ANSYS®

Boundary Conditions - Tip

- **Under StaticStructural Analysis Systems:**
  - **Define Source Time: All on the Imported Body Temperature**
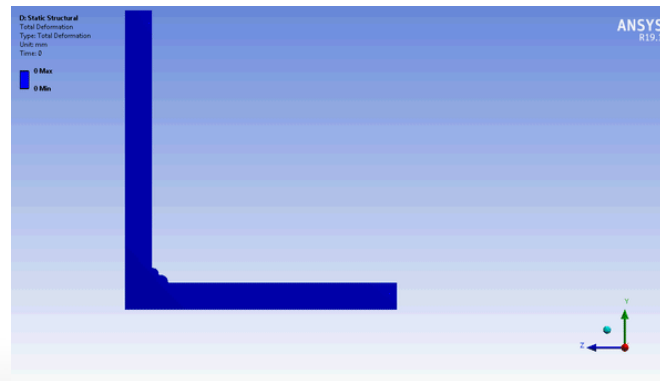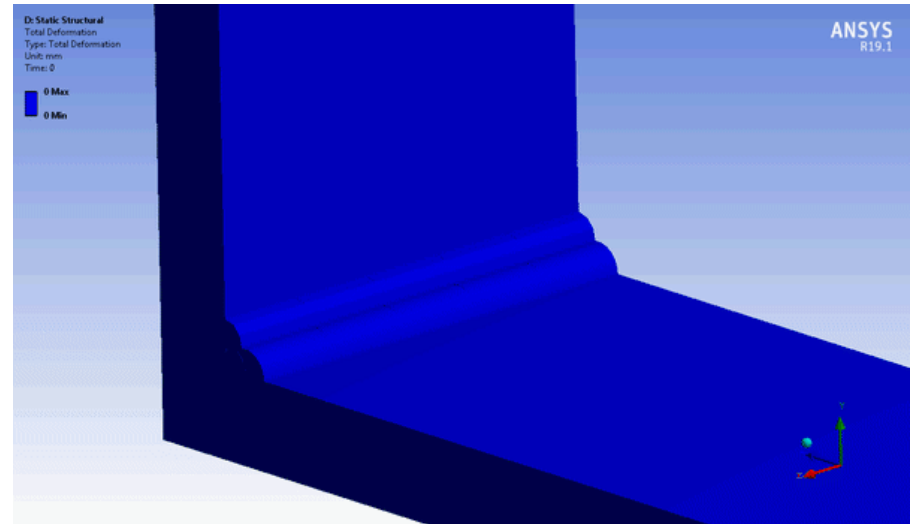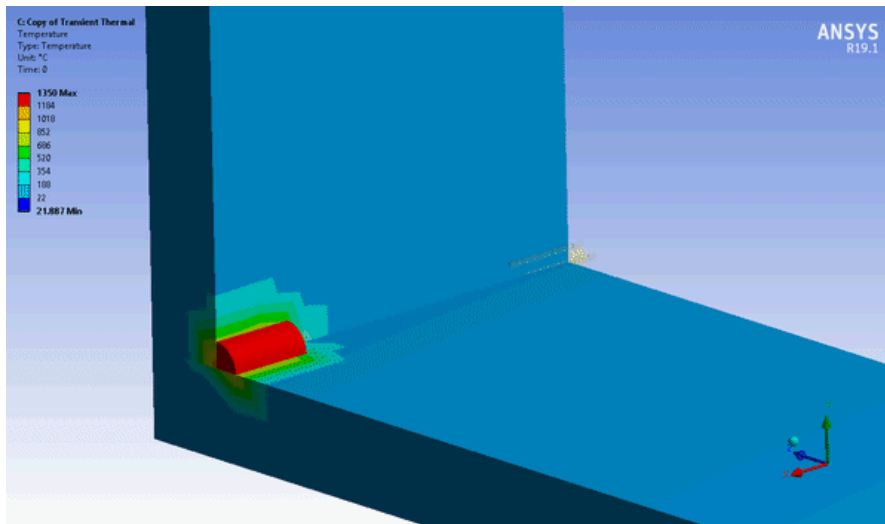
Welding Process Script Example

Geometry Preparation	Material Properties	Meshing	Mechanical Set Up	Post-Processing

Thank you

- Abel Ramos

- abel.ramos@ansys.com

Copyright and Trademark Information

ANSYS®