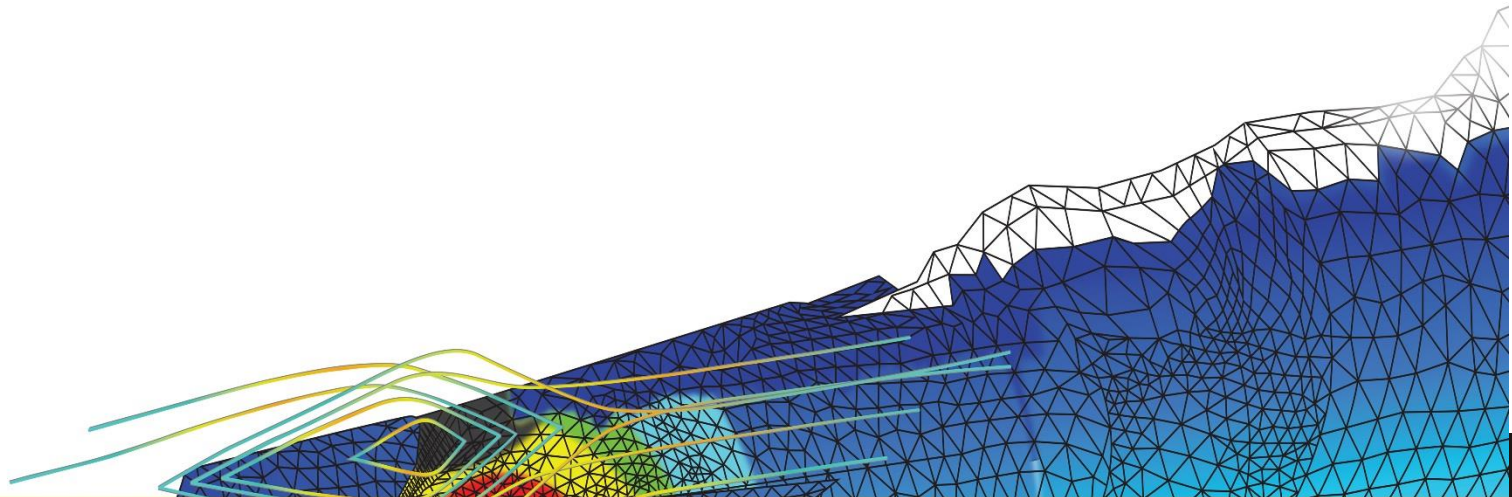


ANSYS®

Welding Distortion Wizard

abel.ramos@ansys.com



Copyright and Trademark Information

Copyright and Trademark Information

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

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS, AIM and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

Welcome to the
ANSYS App Store



- 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)”

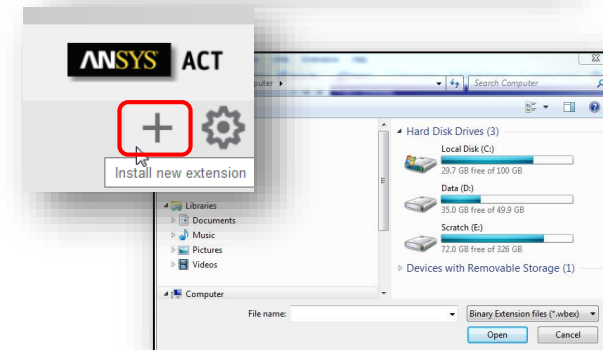
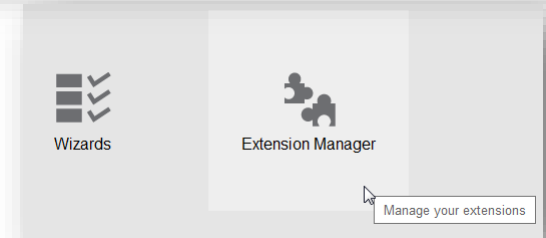
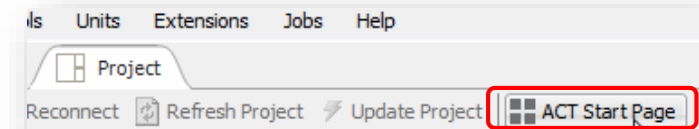
- Report any issue or provide feedback related to this app please contact:

Contact email address: abel.ramos@ansys.com

Binary App Installation (1)

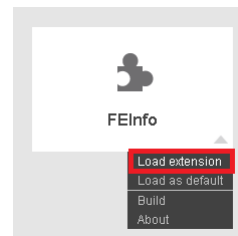
Installing from the ACT Start Page:

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



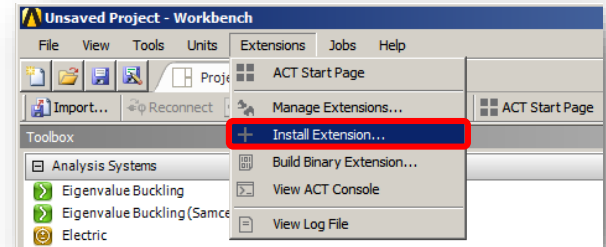
Notes:

- 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

Binary App Installation (2)

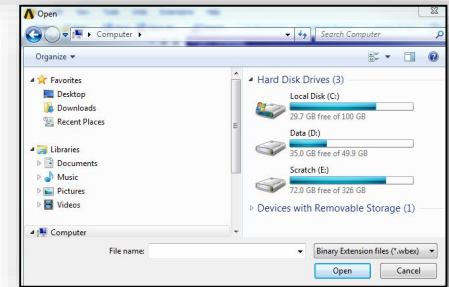
Installing from the Extensions menu:

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

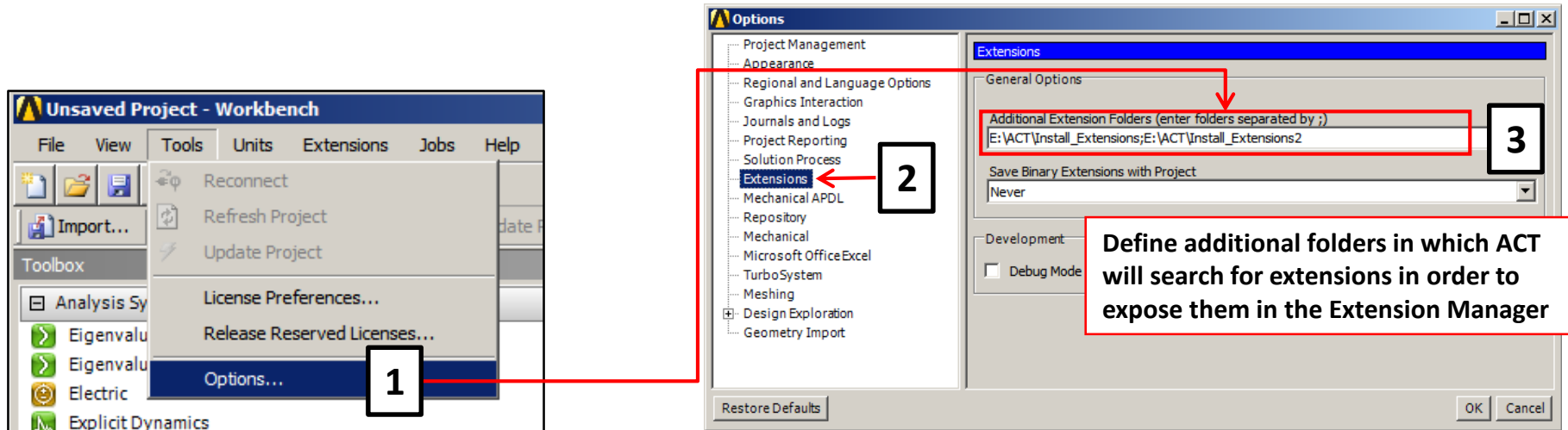


Notes:

- 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

Binary App Installation (3)

- Once the binary extension is installed at default location, one can move the *.wbex and the folder 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 ...”

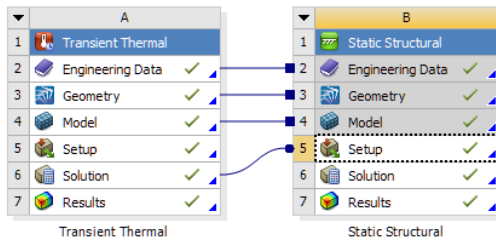
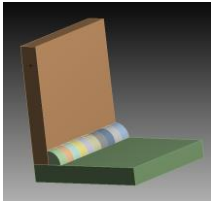


Notes:

- 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

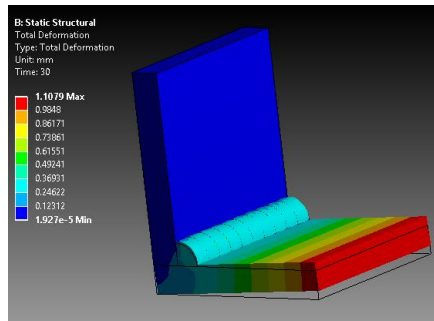
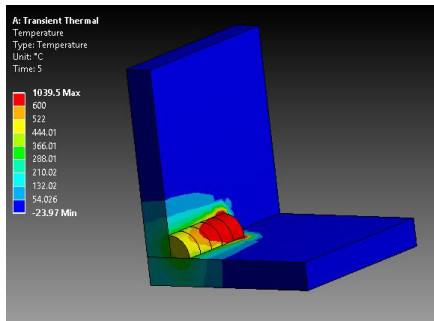
Documentation

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



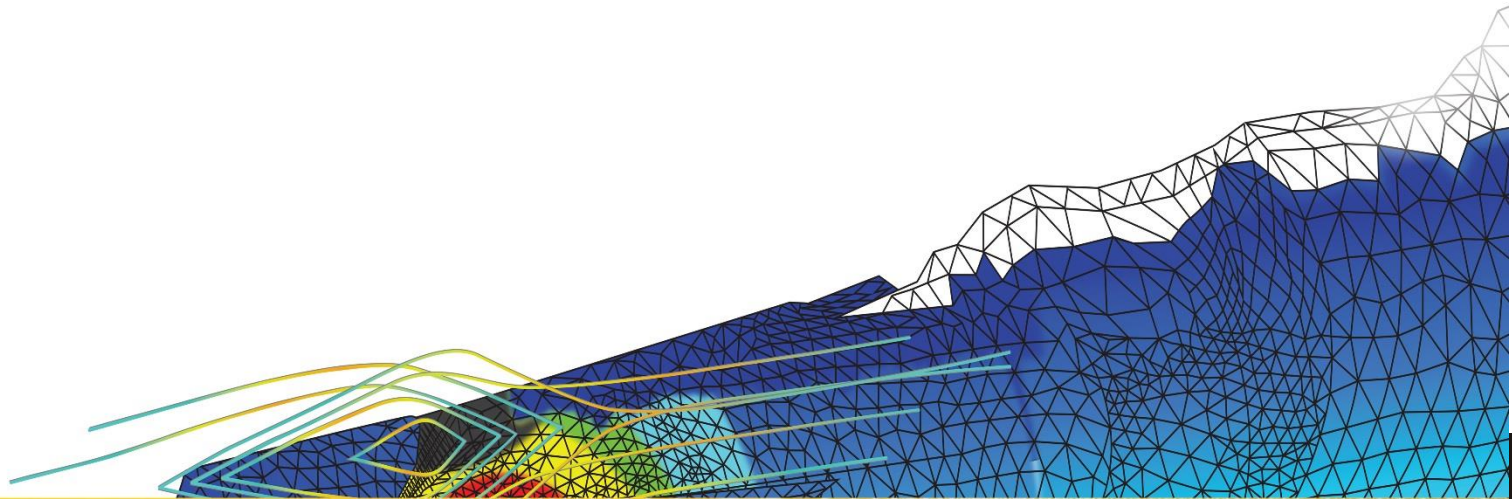
Assumptions:

- 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.





Simulating welding process - Wizard

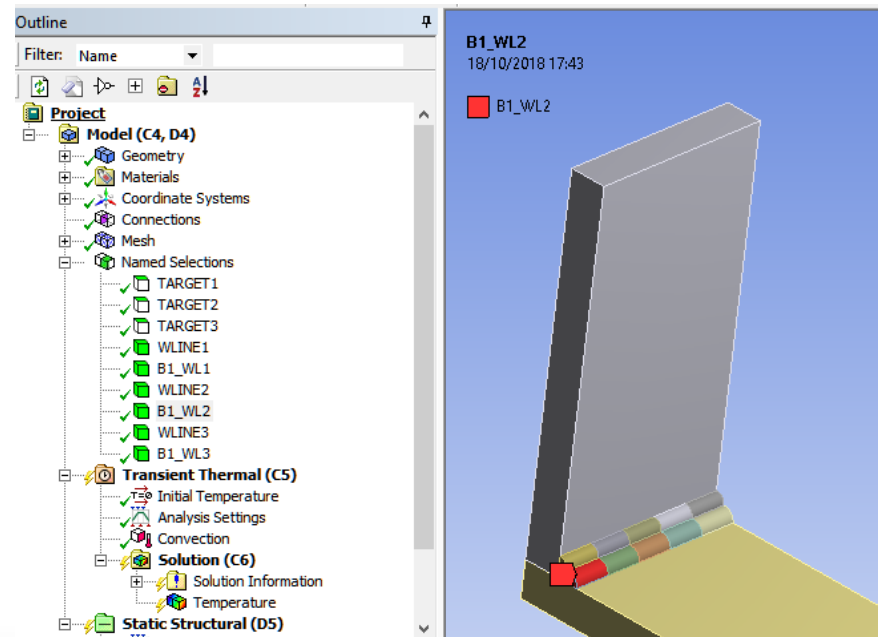
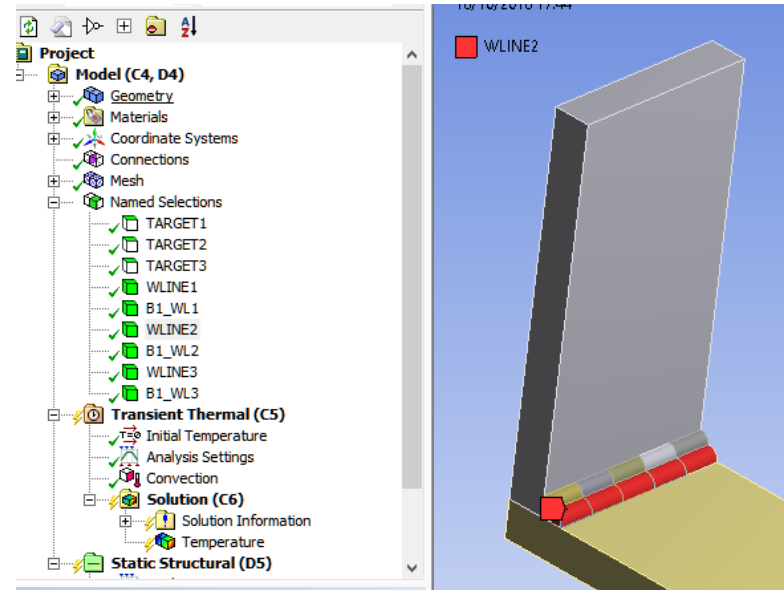


Wizard

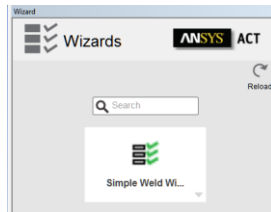
Name Selection Automatic Creation
If WELD1, WELD2, , WELD3 wants to be automatically created, you can use two name selections

- **WLINEi**: Containing 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

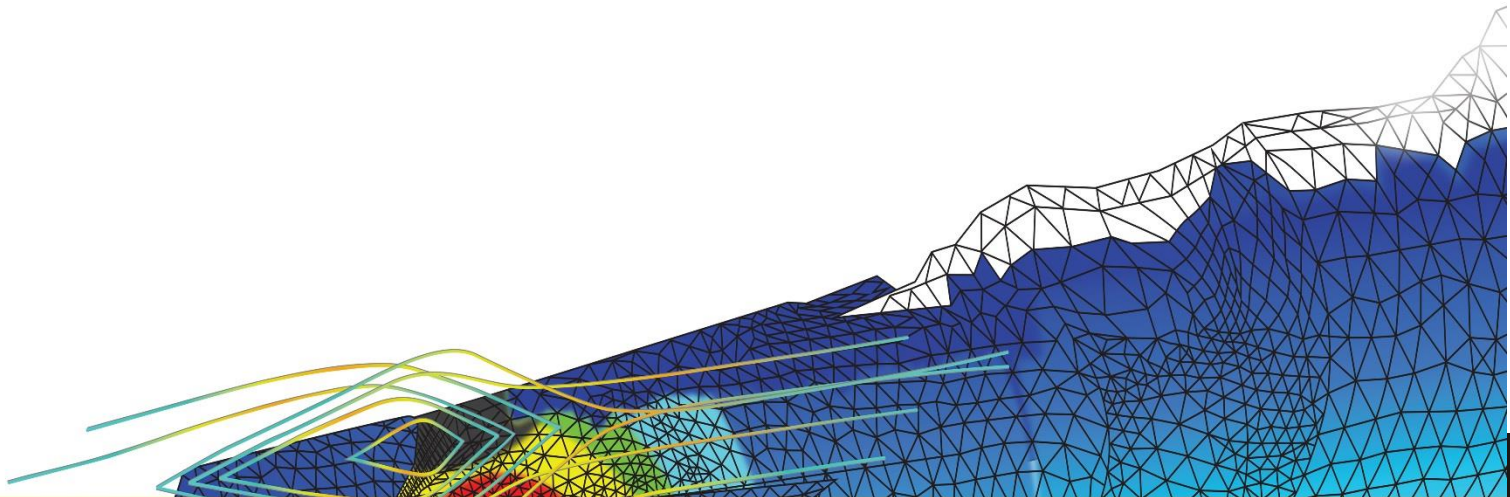
A screenshot of the 'Simple Weld Wizard' configuration window. The window has a title bar 'Wizard' and a header with the ANSYS logo and 'ACT'. The main area is divided into a left sidebar with 'Mechanical' selected, and a main panel with various input fields. The inputs include: 'NS 1-Skip / 0-Create' (input required), 'N welding toes' (input required), 'N of welding bodies on...' (Tabular Data, Edit button, input required), 'Total welding time' (input required, sec), 'Total cooling time' (input required, sec), 'Melting Temperature' (input required, C), and '0-Contacts/1-Share topo' (input required). A 'Help' sidebar is visible at the bottom left, showing a tree view of the project structure including 'Static Structural (B5)', 'Analysis Settings', 'Imported Load (A6)', 'Imported Body Temp', and four 'Element Birth and Death' items.

- 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 topology (1)

ANSYS®

Simulating welding process

Workflow



Welding Process Project Example

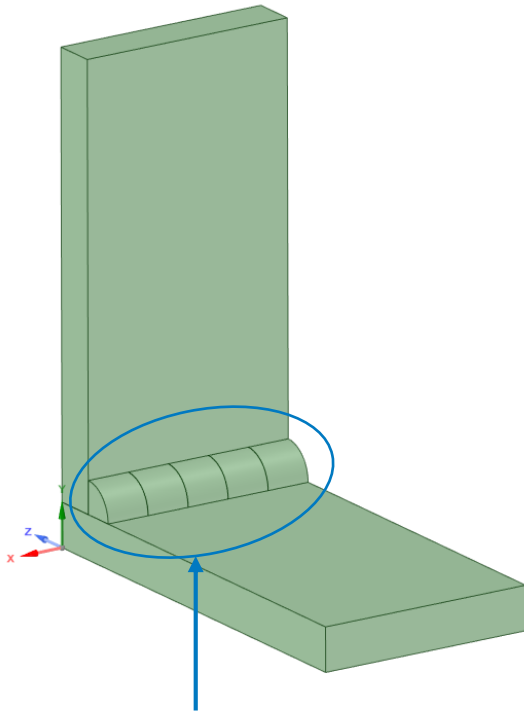
Geometry Preparation

Material Properties

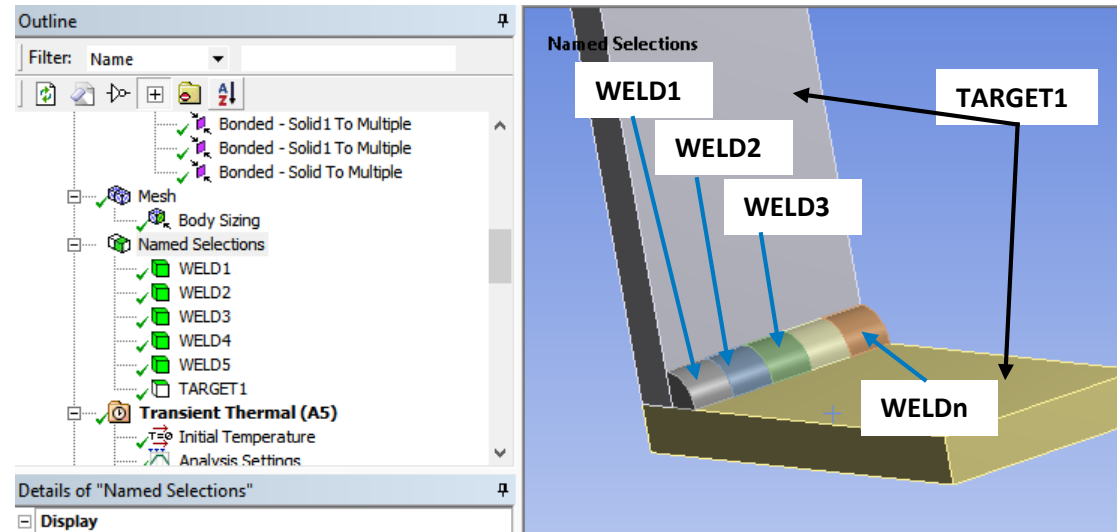
Meshing

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

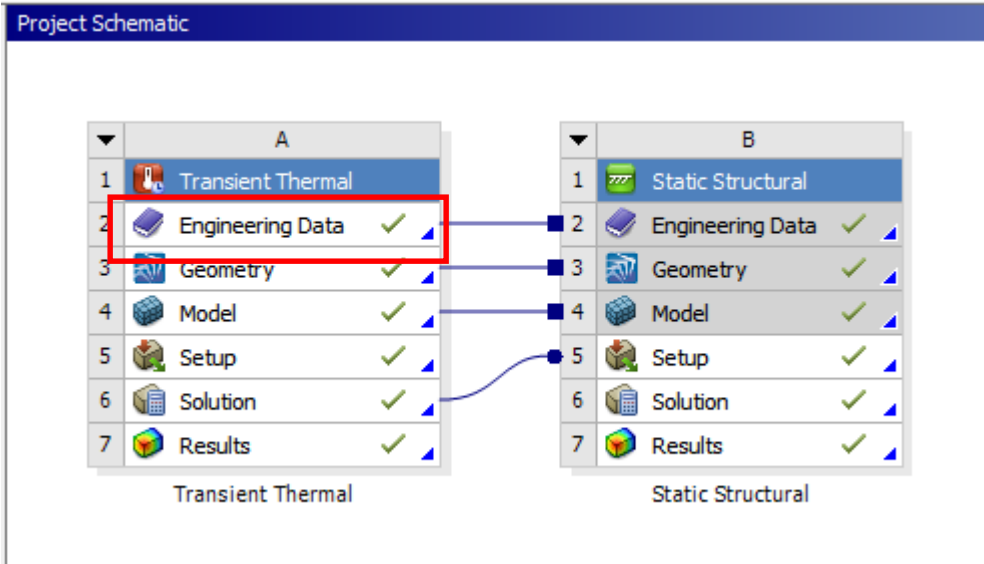
Material
Properties

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

Wizard

Simple Weld Wizard ANSYS ACT

Mechanical
Model Setup:
Welding
Process

NS 1-Skip / 0-Create	<input type="text"/>	Input required.
N welding toes	<input type="text"/>	Input required.
N of welding bodies on...	<input type="text" value="Tabular Data"/>	<input type="button" value="Edit"/> Input required.
Total welding time	<input type="text"/>	sec Input required.
Total cooling time	<input type="text"/>	sec Input required.
Melting Temperature	<input type="text"/>	C Input required.
0-Contacts/1-Share topo	<input type="text"/>	Input required.

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

- 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 topology (1)

Welding Process Script Example

Geometry
Preparation

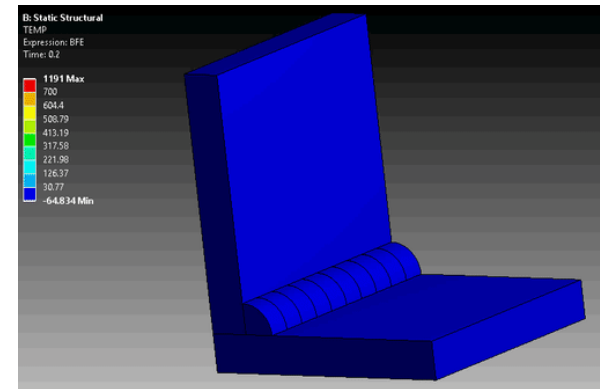
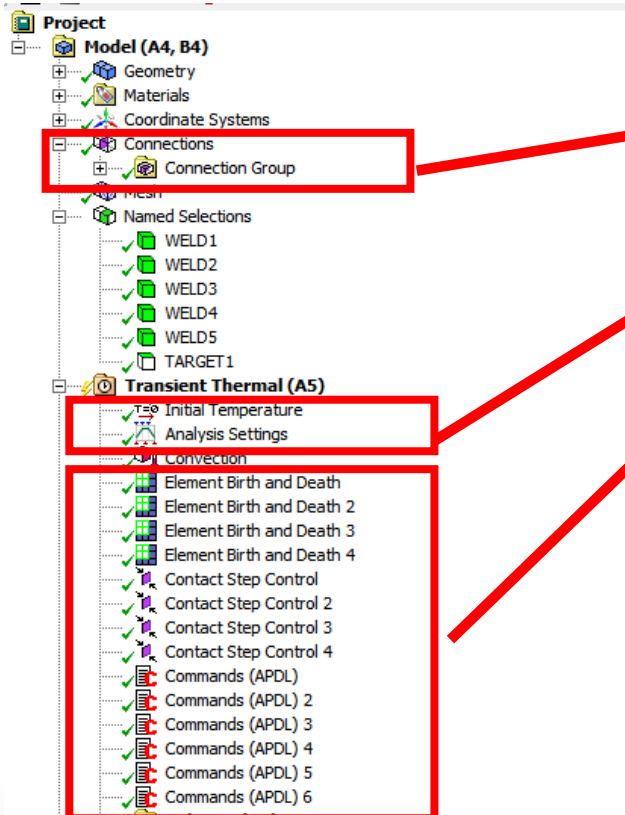
Material
Properties

Meshing

Mechanical
Set Up

Post-Processing

Automatically
created with the
wizard



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

Welding Process Script Example

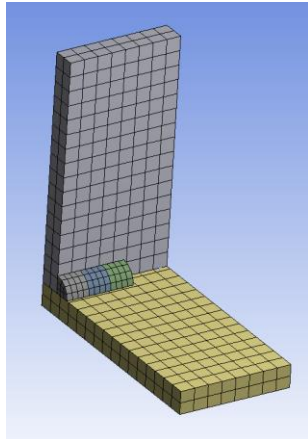
Geometry
Preparation

Material
Properties

Meshing

Mechanical
Set Up

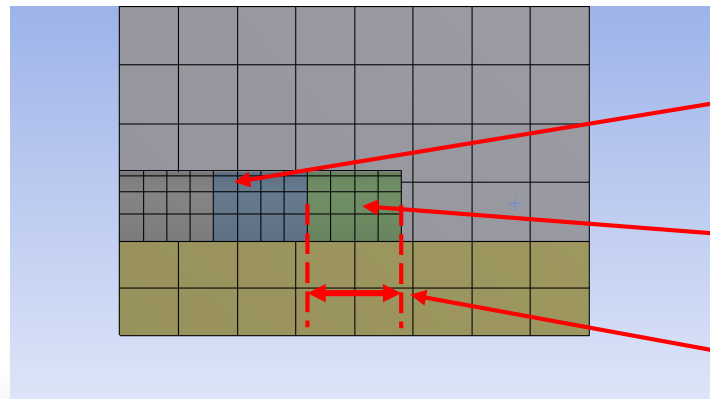
Post-Processing



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 $TWtime / (\text{number of welding bodies})$
- Each new WELDn body is activated at $T = \text{Melt Temperature}$ defined by the user. Temp is maintained during the load step and is freed to cool down in the next load step

Example: Load Step 3



Temp = Cooling down
depending on B.C.

Temp = Melt_temp [°C]

Time = $Twtime/5$

Welding Process Script Example

Geometry
Preparation

Material
Properties

Meshing

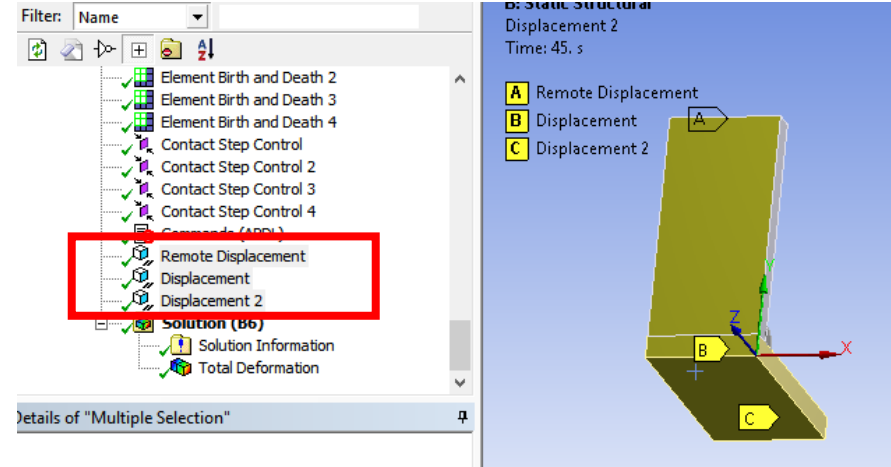
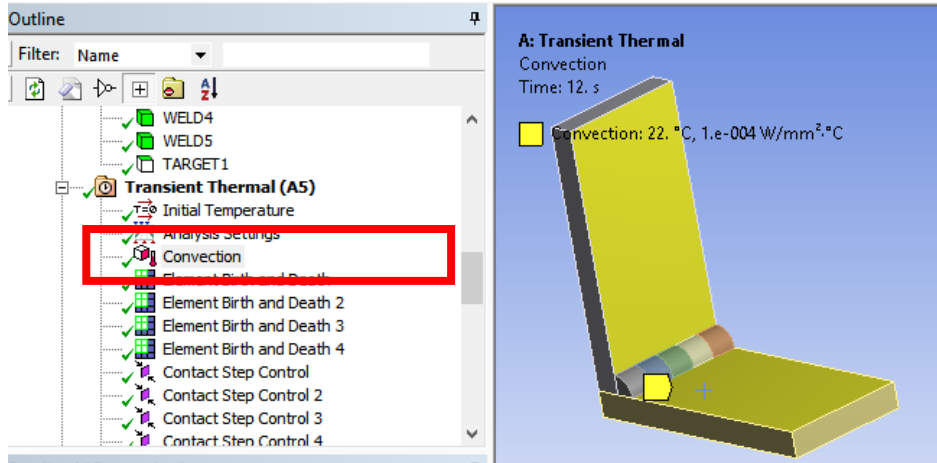
Mechanical
Set Up

Post-Processing

Example additional boundary condition suggested

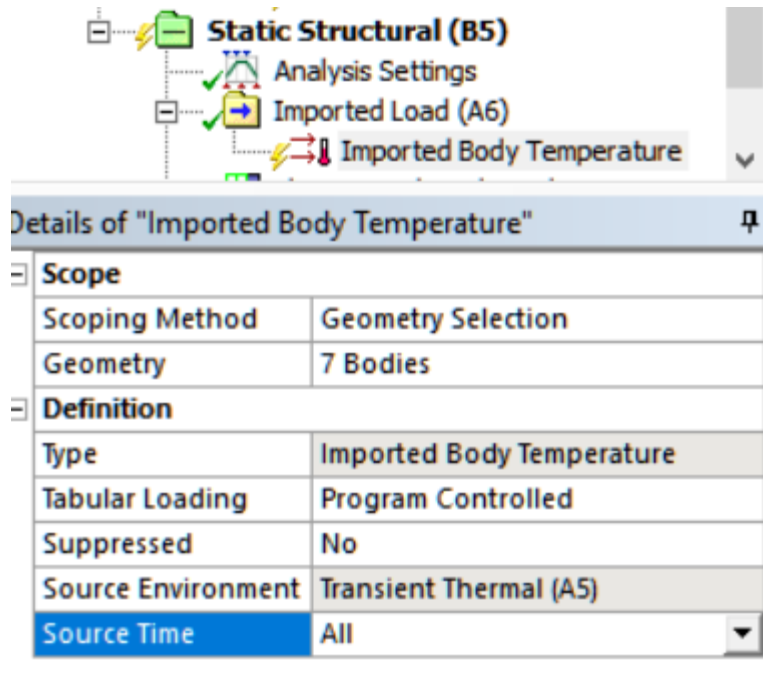
Thermal: Convection

Structural: Displacements



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

Boundary Conditions - Tip



- Under StaticStructural Analysis Systems:
 - Define Source Time: All on the Imported Body Temperature

Welding Process Script Example

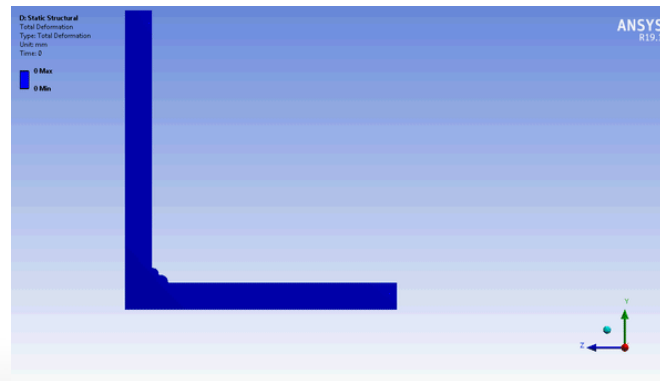
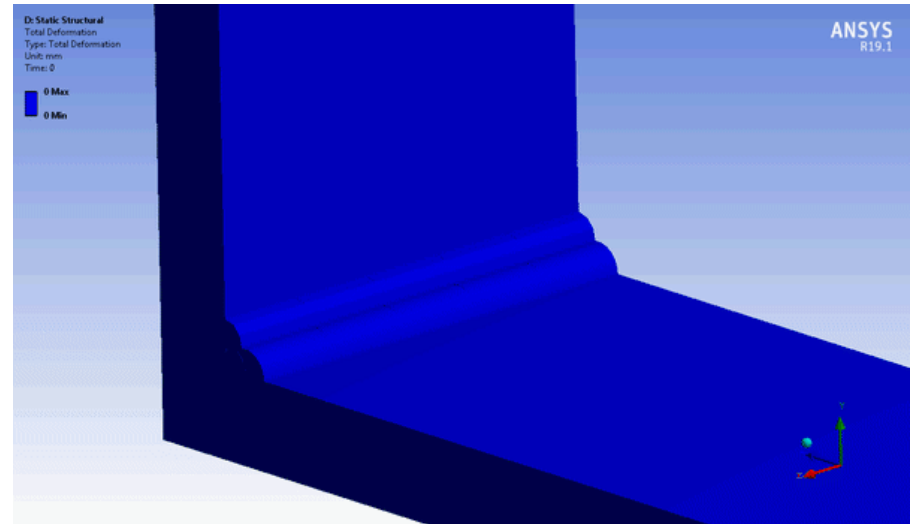
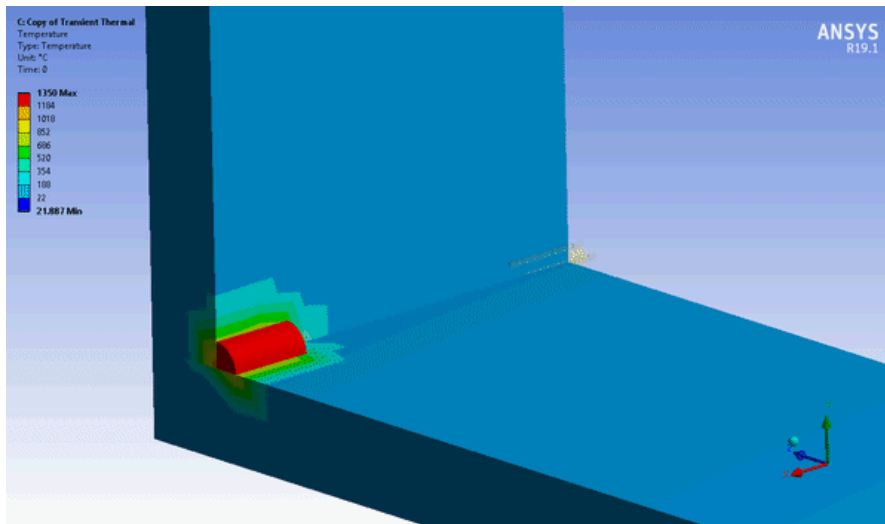
Geometry
Preparation

Material
Properties

Meshing

Mechanical
Set Up

Post-Processing



Thank you

- Abel Ramos
- abel.ramos@ansys.com

Join the ACT Group
on LinkedIn:

"Customization ACTors
for Engineering
Simulation"



Copyright and Trademark Information

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