

An Abaqus Extension for Welding Simulations

M. Shubert¹, M. Pandheeradi¹, F. Arnold¹, and C. Habura²

¹Dassault Systèmes SIMULIA Corporation

²Bechtel Marine Propulsion Corporation

Abstract: An Abaqus Extension, in the form of an Abaqus/CAE Plug-In, has been developed for modeling welding. The Plug-In enables an easy, efficient, model-tree based approach to setup all aspects of a welding model from within Abaqus/CAE by automating most of the repetitive, time-consuming tasks associated with building a welding model in a traditional CAE environment. Examples of tasks supported by the Plug-In include weld-bead and pass definitions, automatic definition of steps and boundary conditions for each pass, automatic (but editable) creation of node sets for each pass to enable definition of temperature sensors, and set-up of both the thermal and mechanical analysis models. The Plug-In, together with the user-defined amplitude subroutine (UAMP), provides support for temperature sensors to automate the termination of the heatup and cooldown steps within the thermal analysis. Presently, the Plug-In is limited to 2-dimensional (2D) welding analyses, but SIMULIA anticipates that it will be extended to 3-dimensional (3D) weld modeling where the benefits over the current manual approach are expected to be even more significant.

Keywords: Welding, Abaqus Plug-In, Residual Stress, AWI, GUI

1. Introduction

Welding simulations are complicated, often requiring accurate representation of geometric features, multiple weld passes, non-linear temperature dependent material properties, thermal and structural boundary conditions, etc. To capture all of these details, significant user effort is required. While analysis capabilities for modeling welding processes have always been available in Abaqus (e.g. robust nonlinear solver, capability to handle large deformations and finite strains, and a huge selection of material models complemented by the extensive user subroutines feature), the setup of weld models – especially the preprocessing aspect – remained a time-consuming part of the weld modeling workflow. Abaqus was used (either exclusively or in conjunction with other software) in the work presented in (Arnold, 2005), (Yaghi, 2006), (Warren, 2006), and (Dawson, 2008). The discussion of the relative merits of the various computational approaches to modeling the welding process is not the subject of this paper, but interested readers should refer to (Goldak, 2005) and (Lindgren, 2007).

While Abaqus/CAE has the capabilities to setup an accurate welding simulation, the process is tedious and time-intensive. For example, an 80 pass welding simulation when using Abaqus/CAE as it is today would require an analyst to create 240 steps (assuming an average of 3 steps per pass), where each step requires insertion of the step time, initial time increment, maximum time increment, maximum allowable temperature, etc. Energy transport mechanisms must be captured

such as convection, radiation, and energy input boundary conditions for the thermal analyses. Each of these requires several data inputs. Additionally, surfaces need to be manually defined to prescribe surface interactions and surface-based thermal loads. The model setup may take several days.

The Abaqus Welding Interface (AWI), introduced in this paper, exists in the form of an Abaqus/CAE Plug-In. It provides the capability to significantly improve two-dimensional welding simulations by automating most of the time consuming tasks associated with building a welding finite element model. In the Plug-In environment the analyst completes a few dialogue boxes with some basic information applicable to the entire welding process being analyzed, selects the beads for each weld in the model from the viewport, and optionally sets up a custom weld pass sequence (or lets the Plug-In automatically define the passes, assuming the passes are consistent with the weld bead ordering). The user can also select machining regions (if applicable) to simulate the removal of weld/base material following the completion of the welding simulation. The Plug-In creates all the required analysis steps with appropriate data, creates energy transport boundary conditions for each step, creates sets for sensor output requests, and builds the entire thermal model followed by the automatic generation of the corresponding mechanical model for the stress (structural) analysis. The setup time for the 80 pass model referred to earlier would be reduced to approximately an hour (with actual user time being only a fraction of this time), as compared to days with the traditional approach.

The remainder of the paper is organized as follows. The AWI overview is described in Section 2, with figures of the main dialogue boxes and options associated with this new capability. Section 3 is devoted to an initial validation problem that demonstrates the use of the new Plug-In with a comparison of results to a validated weld residual stress (WRS) simulation approach. The conclusions appear in Section 4.

2. Abaqus Welding Interface

2.1 Overview

A convenient, model tree-based approach familiar to Abaqus/CAE users was adopted for this welding graphical user interface (GUI). As shown in Figure 1, a tab named “**Weld Modeler**” appears in the model tree when the Plug-In is enabled (via Plug-In menu in Abaqus/CAE), in addition to the standard Abaqus/CAE model tree tabs “**Model**” and “**Results**” (and any other custom tabs). It is in this new tab that all aspects of setting up the welding analysis are accomplished as described later. The standard Abaqus/CAE “**Model**” tab is used for aspects of the model setup that are not unique to the welding analysis, such as:

Part Module:

- Geometry import or Abaqus/CAE geometry generation
 - Create part(s) in the modeling space currently supported by the Plug-In (2D planar or axisymmetric)
- Partitioning
 - Partitions to represent individual beads (lumping of beads handled in AWI)

- Partitions to represent machining regions, if applicable

Property Module:

- Creation of material model, followed by assignment using typical Abaqus/CAE methods (section properties and definitions can be defined but are not necessary)

Assembly Module:

- Assembly of the current weld model
- Positioning of the model (if necessary) using current Abaqus/CAE capability

Mesh Module:

- Creation of mesh on all components of the model
- Assignment of appropriate thermal element types (equivalent structural elements created by AWI in the structural model)

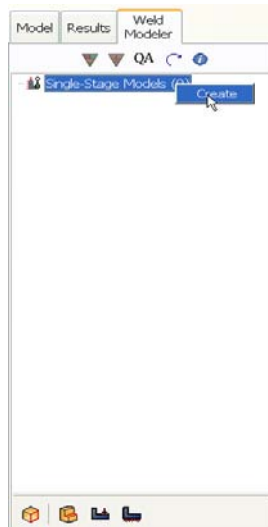


Figure 1 The "Weld Modeler" tab in model tree.

The AWI currently assumes a sequentially coupled approach for the thermal stress analysis associated with the welding simulation. This means the thermal analysis is performed first, followed by the mechanical analysis to compute the residual stresses and deformations resulting from the welding process. The AWI constructs both the thermal and mechanical models, including the necessary step definitions, step-dependent boundary conditions (film, radiation, temperature/flux etc. for the thermal analysis), and step-dependent temperature field specifications (for the stress analysis) accounting for the evolving geometry from one weld pass to the next, based on the user selections and input through the GUI.

2.2 Analysis methodology

Presently, the AWI is setup assuming the following methodology for performing a 2D welding simulation:

1. The weld torch is simulated by applying a prescribed temperature (of magnitude higher than the melting temperature) at the boundary between the beads involved in the current weld pass and the neighboring region (which could be base material and/or weld beads already deposited in previous passes).
2. The beads associated with the current pass and subsequent passes (i.e., beads yet to be deposited) are not present in the model during the weld torch application step for that pass. Beads for a pass are activated through the “model change” capability in a separate step just prior to the start of the cool down step for that pass.
3. The torch application and cool down are both analyzed through transient heat transfer steps. An optional “reset temperature” step that ramps the whole model to a constant temperature prior to the next pass is performed in a steady-state heat transfer step.
4. Temperature sensors defined at or near the user-defined depth are utilized to turn off energy input (and proceed to the cool down step) within the torch application step. Similarly, sensors are used to control the cool down process by monitoring the temperatures at the weld bead free surfaces of the already deposited weld bead. Note, sensors can be turned off for all or individual passes by the user.
5. Weld beads are always active during the stress analysis. However, to ensure the yet to be deposited beads do not influence the deformation, these un-deposited beads have nodal temperatures near the melting temperature as specified by the initial conditions. This leads to very soft or compliant mechanical material properties. Having all beads present from the start of the analysis (but, with melt-like properties) allows for the beads to move with the deformation in the weld zone while not affecting the overall response until they are “active” or deposited. This is necessary for large deformation problems in welding simulations. In order to ensure a totally strain-free (including zero elastic strains) “activation”, beads corresponding to a particular pass are removed and re-introduced just prior to the cool down step for that pass.
6. Machining operations are modeled using the “model change” technique during the mechanical analysis to remove finite elements associated with any machined region.

2.3 Weld definitions and welding pass setup

The AWI allows for the simulation of multiple welds at the same time, but assumes that they are all located on the same geometric part which is selected when a new weld model is created (Figure 2). Other input required to set up a new weld model includes selection of the base material name and region as well as selection of a unit system (British or SI) so that modifiable defaults can be set for certain input parameters such as the initial temperature of the base metal. Note, as with any Abaqus analysis, the user is still required to supply all the rest of the parameters and material properties in consistent units.

The dialogue box for creating a new weld (within the weld model) is shown in Figure 3, which also has an option to set defaults for certain parameters in the selected unit system. Each weld is assumed to be partitioned into the maximum number of individual weld beads that are to be deposited. Once all of the welds have been defined (i.e. the beads have been selected for each weld (Figure 4)), the user can proceed to define the weld passes. The user has complete control over the beads that go into each weld pass definition, thus enabling “what-if” studies for weld pass

sequence design or lumping studies. The user creates weld passes manually by choosing the ‘manual pass creation’ option and selecting the beads associated with a pass from the list of beads in the pass creation dialogue box (Figure 5). The Plug-In automatically updates the bead list to include only beads (from all welds) that are yet to be associated with a pass. Alternatively, the user can select the automatic pass creation option. Here, the Plug-In creates passes with exactly one bead per pass, resulting in the same number of passes as the total number of beads in all the welds (Figure 6).

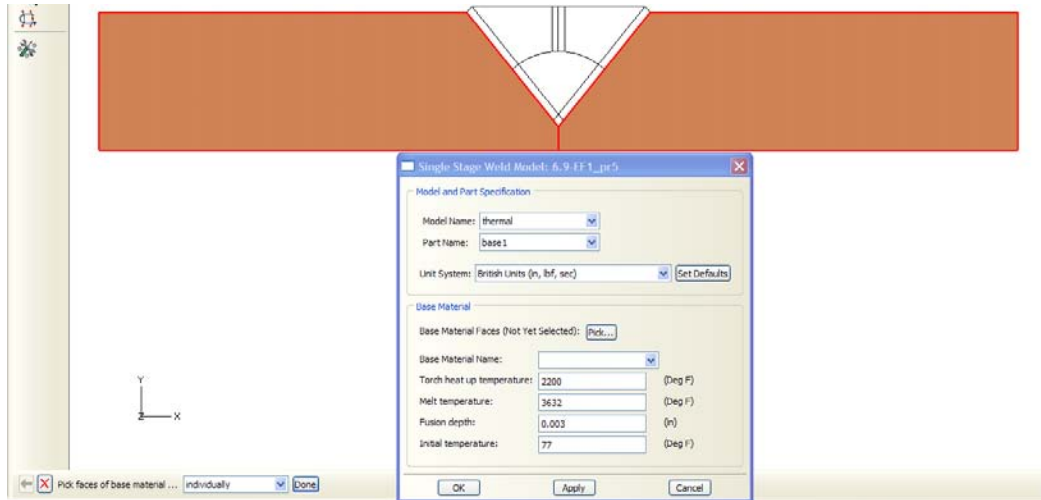


Figure 2. Main dialogue box for weld model setup.

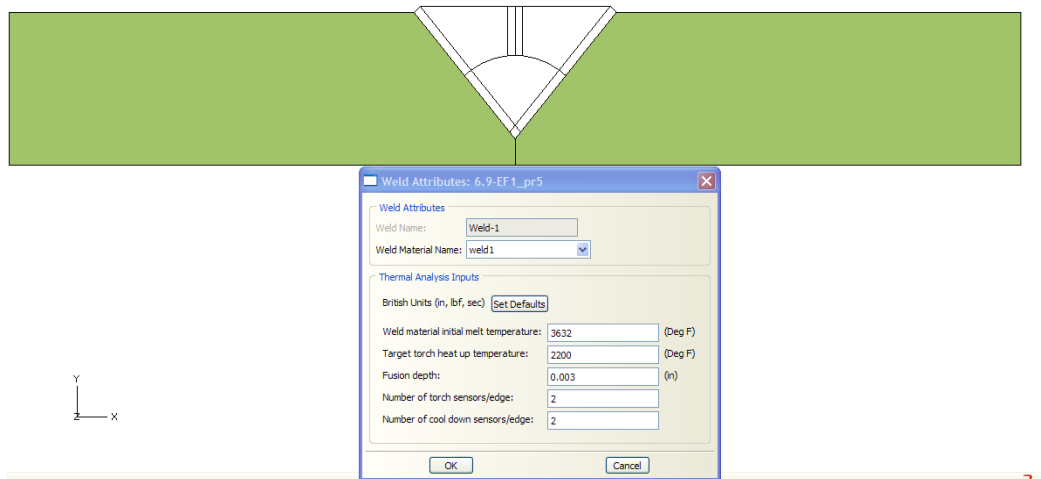


Figure 3. Weld creation dialogue box.

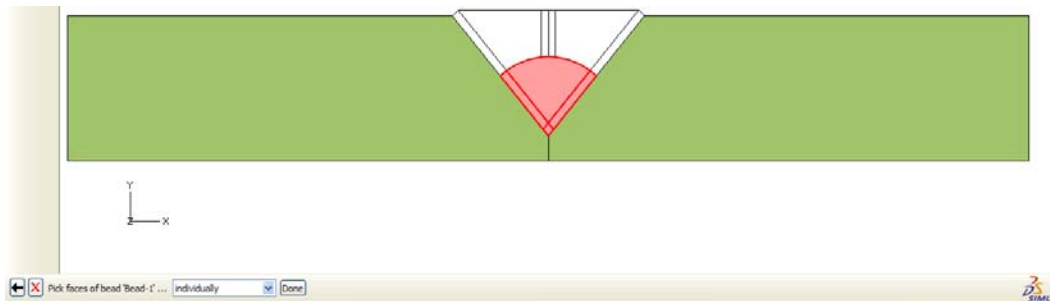


Figure 4. Selection of weld beads.

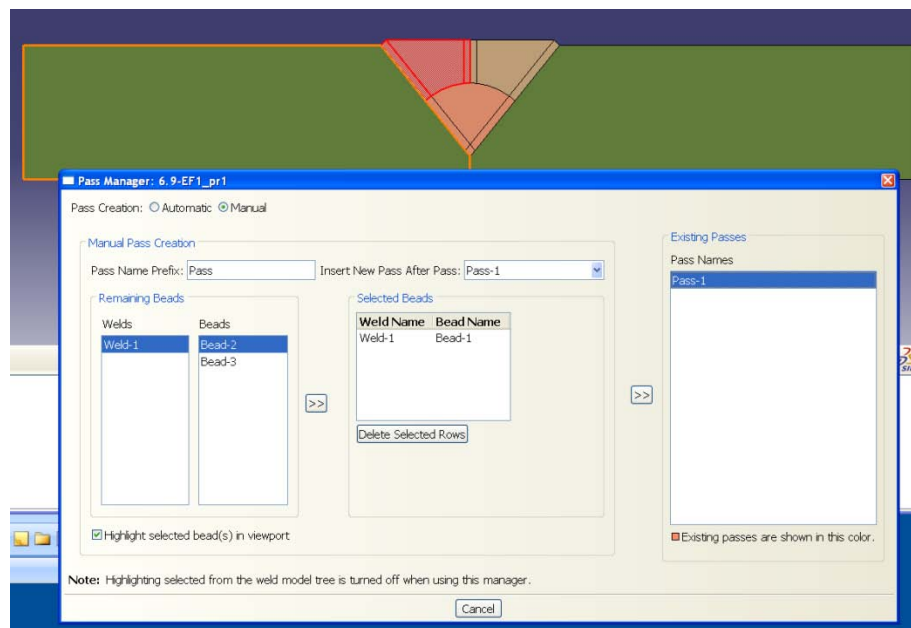


Figure 5. Creation of passes using the “Manual” option.

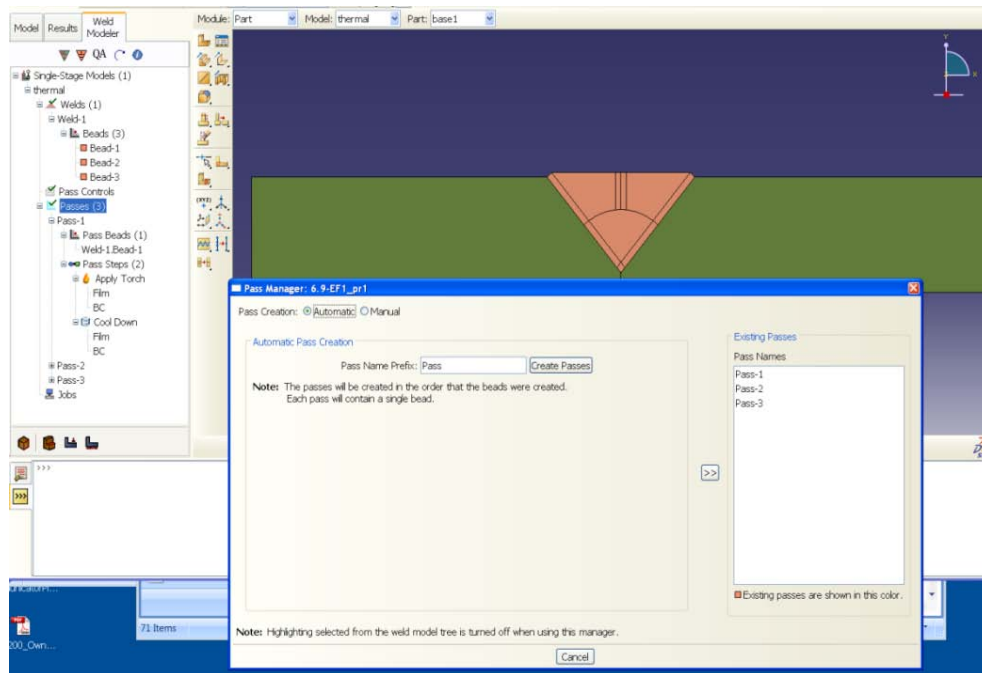


Figure 6. Creation of passes using the automatic option.

The Pass Controls dialogue box (Figure 7) serves as the master controller for all weld passes. The analyst can supply the AWI the information on film load definitions, radiation conditions, and the analysis steps that would be included for all weld passes. It is noted that all of the options available from the standard Abaqus/CAE GUI for film and radiation interaction definitions are supported in the AWI via the “Film Properties” and “Radiation Properties” tabs, respectively. Film conditions can be defined on an edge-by-edge basis or a simple default film condition can be applied to all edges of the base material (Figure 7). Parameters for radiation boundary conditions are currently assumed to be constant for the entire model. The default film coefficient (and radiation if activated) is applied to all weld bead surfaces exposed to the “environment” during the welding process. The Plug-In keeps track of the active, evolving boundary edges for each weld pass and appropriately enables or disables the film and radiation conditions on each edge accordingly. The pass control dialogue box for film definition also includes convenience features such as “copy”, “highlight edges”, and “activate”.

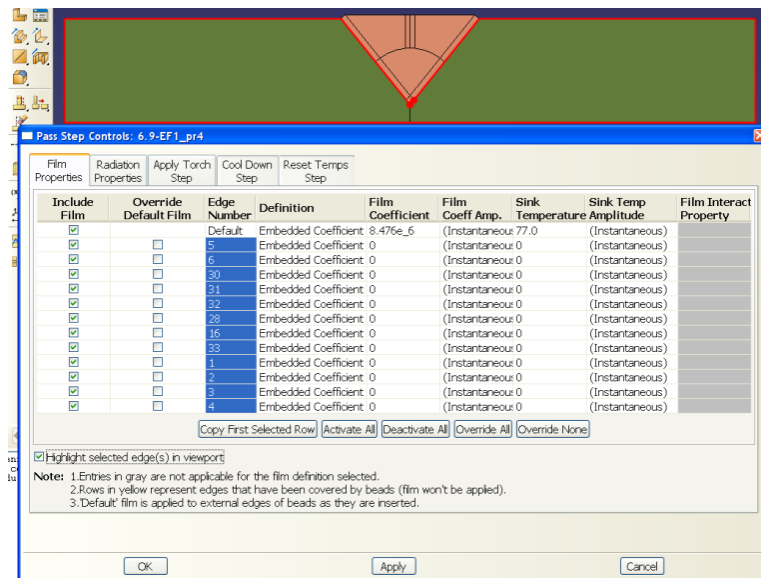


Figure 7. Pass controls for film interactions.

The other tabs in the pass control dialogue box allow the user to enter the essential information for the analysis steps that are common for all passes. The “Apply Torch” step tab is associated with the required step information for simulating the weld torch application (resulting in bead deposition) for a particular pass by applying a temperature boundary condition at the boundary/interface between the beads that are part of the current pass and the adjoining material. The “Cool Down” step tab allows for the step definition information needed to define the cool-down phase following the bead deposition. The “Reset Temps” step tab allows for a step to be defined that simply ramps the current temperature state of the model to a constant temperature before the next weld pass begins.

Figure 8 shows the contents of the “Apply Torch” tab, illustrating the necessary inputs to the analysis steps. Although not shown here, the other two tabs have a similar appearance to the illustration in Figure 8. Note, the main difference between the “Reset Temps” tab and the other two step related tabs is the availability of an option to terminate the step for both the “Apply Torch” and “Cool Down” steps.

If selected, step termination is accomplished using temperature sensors. Sensors defined at a user-defined (e.g., fusion) depth below the surface (for the “Apply Torch” step) and at weld bead free surface locations (for the “Cool Down” step) “turn off” the analysis step when the user-specified sensor temperature (e.g., fusion or interpass) is reached. The Plug-In creates the necessary node sets and sensor definitions for both types of sensors for each weld pass. These sensors are accessed during the thermal solution from within the user subroutine UAMP to control the heat input and cooling time. The part is required to be meshed by the user prior to sensors being defined. The sensors can be regenerated as needed (e.g., when the part mesh changes) as shown in Figure 9 and can be reselected (i.e., edited) at the pass level (Figure 9, Figure 10).

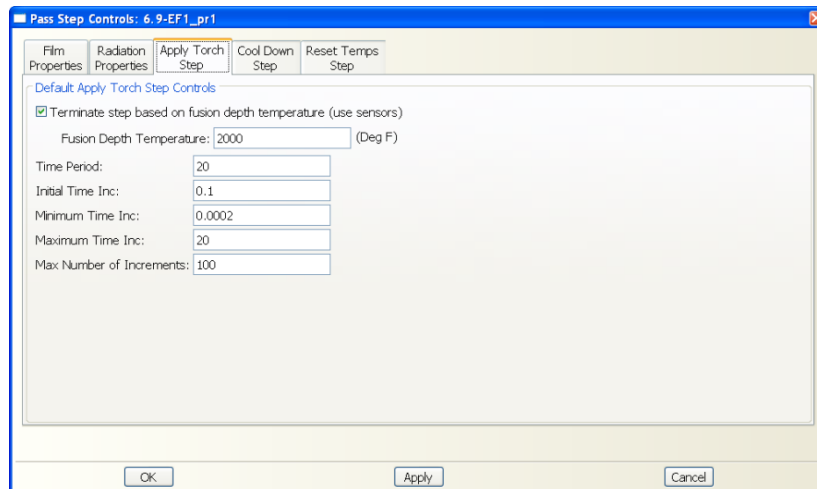


Figure 8. Pass controls for "Apply Torch" step.



Figure 9. Sensor Options (a) regenerate sensors for the whole model (left) and (b) reselection of sensors (right) at pass level, if needed.

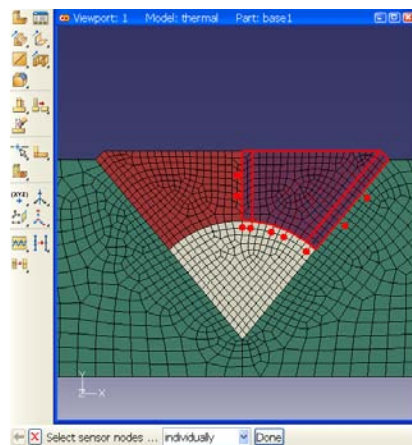


Figure 10. Reselection of sensors at the pass level.

A snapshot of an expanded Model Tree is shown in Figure 11. Here, the various containers have been expanded to show the hierarchy. It is possible to highlight the active edges where film or boundary conditions have been applied for a particular step in a pass. This makes it easy to visually verify that they have been applied correctly (edges are highlighted in red in Figure 12, Figure 13, Figure 14). The Model Tree also allows the editing of attributes (e.g. film conditions, step information etc.) of a particular pass and/or step as well as insertion of analysis steps in addition to the default steps common to all passes (Figure 15). For example, there may be an intermediate creep hold step after a pass which needs to be present in the mechanical (stress) model. Any machining steps are also incorporated in this way by selecting “Model Change” as the step type. The choice of a “User-Defined Step” type allows even more flexibility. For example, the user can insert a “preheating” step anywhere during the process.

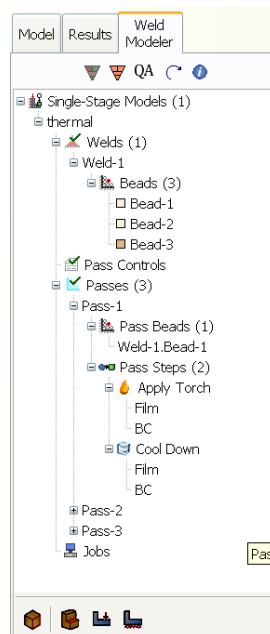


Figure 11. Expanded model tree illustration.

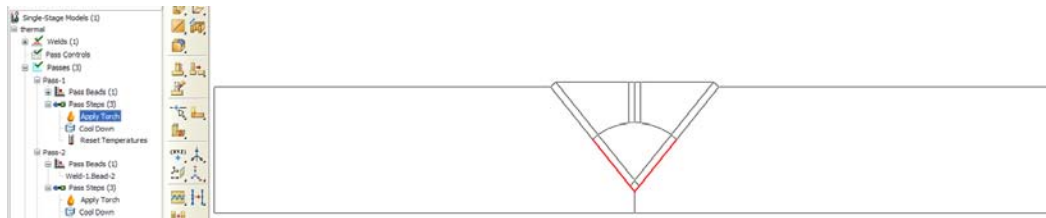


Figure 12. Boundary condition edges for simulating torch application for the selected pass.

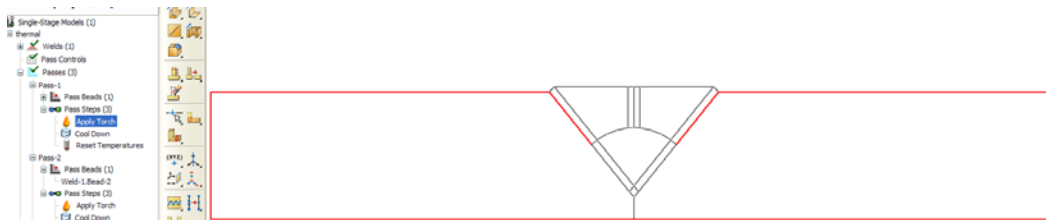


Figure 13. Film load (heat removal) edges during torch application for the selected pass.

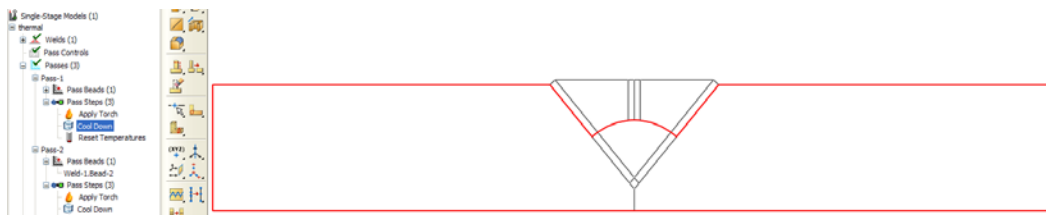


Figure 14. Film load (heat removal) edges during cool down for the selected pass.

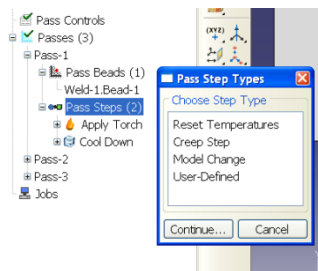


Figure 15. Option to insert a new analysis step.

2.4 Creation of steps and analysis models

Once the passes have been created, it is trivial (from the user's perspective) to generate both the thermal and mechanical models. These models are created automatically and simultaneously by the Plug-In. Options exist to control settings before the models are generated (Figure 16).

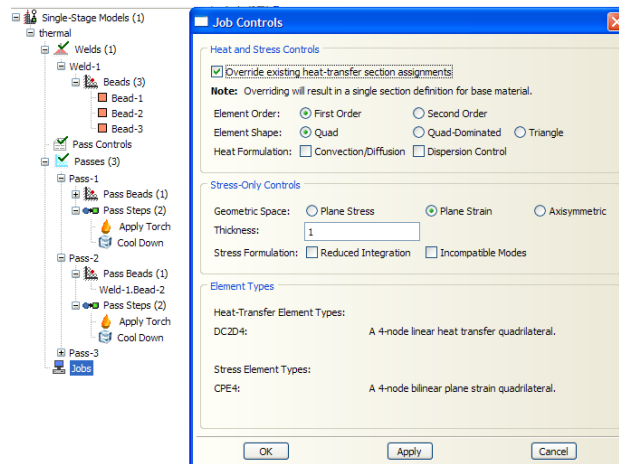


Figure 16. Generation of thermal and mechanical models.

3. Example/Validation of analysis approach

The complete validation of a welding solution is difficult to obtain. In general, thermal solutions can be compared to microstructure data or accompanying welding thermocouple data to verify the heat input and that the thermal solution is predicting the correct temperatures within the welded part. The process of validating the structural solution can be even more difficult and expensive. Surface displacement measurements can be used to validate finite element analysis (FEA) deformations. More importantly, surface and through-thickness stress measurements can be performed so that a direct comparison can be made between experimental and analytical solutions. While the AWI mainly aids in the tedious and time-consuming setup of WRS simulations, it does impose a certain direction in the simulation of welding processes. For instance, the heat source modeling for the current 2D AWI is performed using a linear temperature ramp as opposed to a heat flux (presently not included). The AWI has an option that makes use of the UAMP user subroutine to monitor sensors (nodes) to determine when the heat source should be turned off or a cooldown cycle terminated. For these reasons as well as others not mentioned here, validation of the AWI is necessary so that a reasonable WRS solution is ensured.

While validation of a welding simulation can be difficult to obtain, comparisons to measured temperature histories and stresses have been performed. For example, in (Katsareas, 2004) comparisons to thermocouple readings as well as various stress measurements (neutron diffraction, sectioning, hole drilling) were used in order to validate the FEA weld residual stress solution. Other references, such as (Ogawa, 2008), present comparisons of FEA predictions to either temperature or stress measurements. However, much of the time it is difficult to use these references in a validation effort. This is mainly due to the fact that typically papers that exist on this subject do not have all-inclusive information needed for a thorough validation. For instance, material property data and stress measurement results may exist, but the article may be lacking proper thermal analysis validation. Alternatively, there may not be enough information to properly reproduce the model. Based on this, validation of the AWI to results in open source

literature has not been performed. However, a comparison was made between results produced by the AWI GUI and an in-house GUI that Bechtel Marine Propulsion Corporation (BMPC) uses to support welding simulations. A simple 3-bead V-groove model (Figure 17) was used.

Analysis specifics common for both the BMPC and AWI analyses were: temperature dependent thermal and structural material properties, linear kinematic hardening, small deformation, plane stress (CPS8R elements), and sequentially coupled thermal-structural solutions. Both the AWI and BMPC welding simulation techniques provide essentially a temperature ramp on the interface nodes between the weld bead and base metal to simulate the application of a torch being applied to the welded region. Differences between the AWI and the BMPC in-house simulation techniques do exist with the main difference associated with the thermal solution weld deposition sequence.

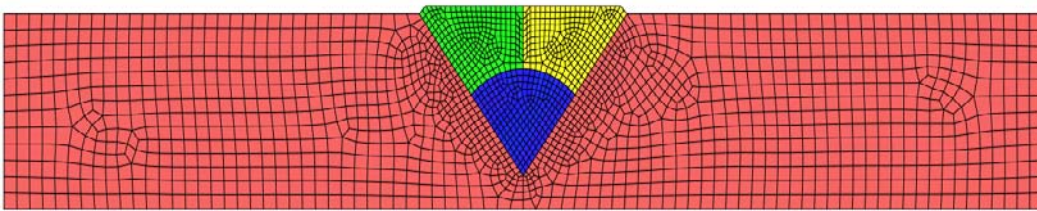


Figure 17. 3-bead V-groove model.

For the comparisons, an assumed fusion depth of one element ($\sim 0.080''$) was used for both analyses. The linear temperature ramp (heat input) was varied for both analyses until the fusion depth requirement was met. In both analyses the time-temperature ramp had a maximum temperature of 300°F - 500°F above the assumed melting temperature. Figure 18 shows the resulting room temperature Von Mises stress contours from the study at the conclusion of the welding process simulation. This figure provides a qualitative comparison of the stress distributions between the two analysis results. While it is evident that there are peak stress magnitude differences between the analysis results, as an initial comparison it appears the AWI and its imposed welding simulation methodology is capturing the proper physics of the problem. It is noted here that “sensors” were not used in the thermal analysis for the AWI simulation for this initial study, instead a manual check was performed for the first weld bead of both analyses to ensure a consistent fusion zone was achieved.

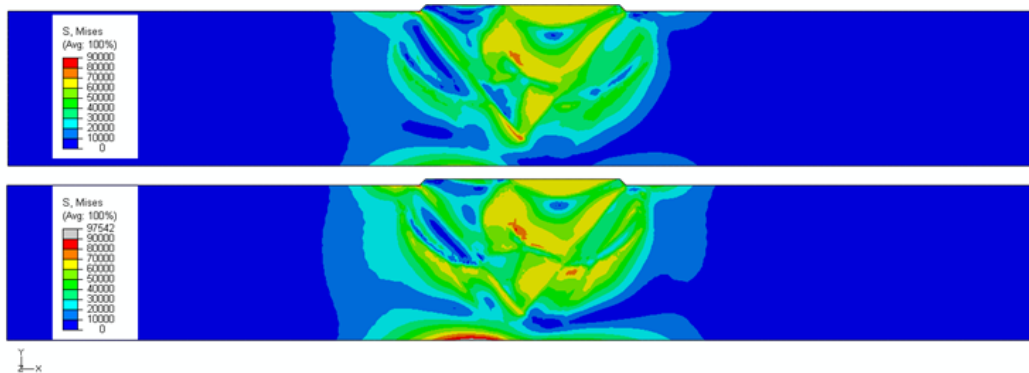


Figure 18. Von Mises stress (psi) result comparison; BMPC(Top), Abaqus(Bottom).

4. Conclusions and future work

A feature-rich, custom graphical user interface (termed AWI) has been developed for setting up welding models within Abaqus/CAE. It is expected that using the AWI will result in significant time savings in the modeling workflow for welding processes. As highlighted in the paper, the custom GUI allows easy setup of a multi-pass welding model by automating most of the time-consuming aspects of the model construction such as setup of pass sequence, creation of appropriate analysis steps, and specification of loads and boundary conditions (thermal analysis only) for both the thermal and stress analyses. Presently, the GUI is only applicable to 2D welding models, but SIMULIA anticipates that it will be extended to 3D modeling where the benefits over the current manual approach are expected to be even more significant.

5. Acknowledgement

The authors gratefully acknowledge Mr. Randall Mabe, Mr. Wayne Long, Mr. Fred Bahr, and Mr. Michael Teague from BMPC and Mr. Kris Altiero from BPMI for their contributions to the AWI effort.

6. References

1. Arnold, F., and M. Pandheeradi, "Low Stress Welding Simulations," Desktop Engineering, October 2005.
2. Dawson, M.I. "Modeling and Analysis of Welding Processes in Abaqus using the Virtual Fabrication Technology (VFT) Analysis Software developed by Battelle and Caterpillar Inc.," 2008 Abaqus Users' Conference Proceedings, pp. 576-589, 2008.
3. Goldak, J. A., and Akhlaghi, M., "Computational Welding Mechanics," Springer Science+Business Media, Inc., New York, USA, 2005.

4. Katsareas, D.E., Ohms, C., and Youtsos, A. G., "On the Performance of a Commercial Finite Element Code in Multi-Pass Welding Simulation," PVP-Vol. 477, pp. 29-37, ASME, New York, 2004.
5. Lindgren, L. E., "Computational Welding Mechanics – Thermomechanical and Microstructural Simulations," Woodhead Publishing Limited, Cambridge, England, 2007.
6. Ogawa, K., Okuda, Y., Saito, T., Hayashi, T., and Sumiya, R., "Welding Residual Stress Analysis Using Axisymmetric Modeling for Shroud Support Structure," Proceedings of the 2008 ASME Pressure Vessels and Piping Division Conference, PVP2008-61148, Chicago, IL, USA, July 27-31, 2008.
7. Warren, A.P., Bate, S. K., Charles, R., O’Gara, D. M., Wood, P. M., and Gregg, A., "The Effect of Modeling Simplifications on the Prediction of Residual Stresses in Thin-walled Pipe Butt Welds," Proceedings of the 2006 ASME Pressure Vessels and Piping Division Conference, PVP2006-ICPVT-11-93384, Vancouver, BC, Canada, July 23-27, 2006.
8. Yaghi, A., Hyde, T. H., Becker, A. A., Sun, W., and Williams, J. A., "Residual Stress Simulation in Thin and Thick-walled Stainless Steel Pipe Welds including Pipe Diameter Effects," International Journal of Pressure Vessels and Piping, vol. 83, pp. 864-874, 2006.