Multi-pass Pipe Welding Analysis using the Abaqus Welding Interface

Abaqus Technology Brief

Summary

Pipes and pipelines play a fundamental role in a wide variety of industries. The cost effective operation of plants and processes depends on the integrity of the necessary piping systems. The ability to accurately determine the residual stress state in a welded pipe joint is thus important; with a predictive capability, engineers are better able to reduce the possibility of in-service failures.

In this Technology Brief we discuss the application of the Abaqus Welding Interface (AWI) [1,2] to model the multi-pass welding of an SUS304 stainless steel pipe joint. We will demonstrate that the AWI analysis methodology provides reasonably accurate predictions of the resulting temperature history and residual stresses, with significant time savings in the construction of complex 3-D weld models. The Abaqus analysis results show a positive comparison with the experimental data reported in [3].

Background

Circumferential butt welds are commonly used to join pipes in various industries, including such critical applications as nuclear power plants [4]. Tensile residual stresses from welding play a major role in the integrity of pipe joints because they increase the susceptibility to fracture, fatigue failure, and stress corrosion cracking.

The residual stresses have a complex dependence on many variables, including the geometry of the structure, temperature-dependent thermal and mechanical properties of the base and weld metals, the sequencing in multi-pass welding scenarios, inter-pass temperature conditions, the boundary constraints on the parts being welded, and the energy input from the heat source.

The ability to accurately predict welding residual stresses during the design process offers cost advantages not only by saving on prototyping and experiments, but by reducing the chance of costly in-service failures. Abaqus, with its advanced thermo-mechanical analysis capabilities, material models, fracture capabilities, and the innovative Abaqus Welding Interface (AWI) [1,2] extension to Abaqus/CAE, is well-suited to meet this analysis need.

While Abaqus is widely used in welding simulations, the model development can be time consuming because weld bead deposition necessitates evolving geometry. Further, this affects related modeling aspects such as loads, boundary conditions, and convective film interactions—each of which must be specified in a potentially large number of steps.

Additionally, if the energy input from the torch is modeled using a flux-based approach [3, 5, 6, 7], user subroutines are needed to implement the chosen heat input model. AWI automates most of the time consuming steps associated with the welding model development, saving valuable analyst time.

Key Abaqus Features and Benefits

- The Abaqus Welding Interface extends the capabilities of Abaqus/CAE for efficient multi-pass welding simulation
  - Axisymmetric and three-dimensional weld deposition simulation
  - Sequentially coupled thermal-stress analysis computes residual stresses and deformations after each weld pass
  - Automated definition of steps, loads, and boundary conditions for highly efficient model building
- SIMULIA capabilities in fracture, damage, fatigue, and optimization that provide a broad analysis framework for comprehensive workflow solutions

Welding simulations built using AWI use a sequentially coupled thermal-stress analysis, where the temperature history from the welding is first determined using a pure heat transfer analysis. This is then followed by static stress analysis that uses the temperatures from the heat transfer analysis in the loading history. The temperature loading, together with the temperature-dependent elasto-plastic and thermal expansion material properties and boundary restraints on the part generate the residual stress distribution.

AWI assumes a prescribed temperature approach to represent the heat input from the welding source. Specifically, for the given "torch step," the interface between the appropriate bead material volume and the surrounding base is assigned a prescribed temperature boundary condition. This is typically at a value higher than the melt temperature.

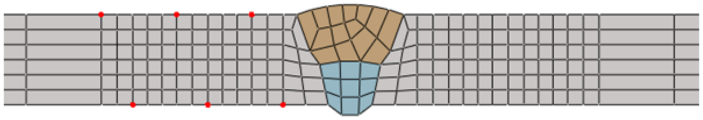

Figure 1: Pipe cross section with thermocouple locations highlighted

Finite Element Analysis

AWI Modeling Approach

The weld beads for each pass are assumed to be deposited in discrete chunks at the target torch temperature. AWI supports both cell-based and element-based deposition in 3-D. For the present 3-D pipe model each bead is subdivided circumferentially into 22 element based chunks. When the model is constructed, all of the weld beads are meshed. The bead chunks are then selectively activated during the analysis sequence via the "model change" feature in Abaqus.

Geometry and Material Properties

The pipe segments being welded are each 400 mm long, with an outer diameter of 114.3 mm and wall thickness of 6 mm [3]. The longitudinal pipe cross section showing the two-pass weld geometry of the 3-D model is shown in Figure 1, along with the locations of the experimental thermocouples on the inside and outside surfaces. The properties of the base metal (SUS304) are described in [3]. The weld metal properties are assumed to be same as those of the base metal (see [5] for a discussion of the effect of different plasticity parameters for the base and weld metal). Linear kinematic hardening plasticity is used in the simulations.

Sequentially Coupled Thermal-Stress Analysis

The heat transfer model is completed in AWI in a few easy steps. AWI creates all the needed interactions (such as film coefficients and step dependent model changes), analysis steps, and temperature boundary conditions. The user-defined film coefficient option was used with a simple FILM subroutine to implement the film coefficients for convective cooling discussed in [3]. A torch step time of 11.6 seconds was estimated for each bead chunk, based on the chunk size and the welding speed (80 mm/min, [3]).

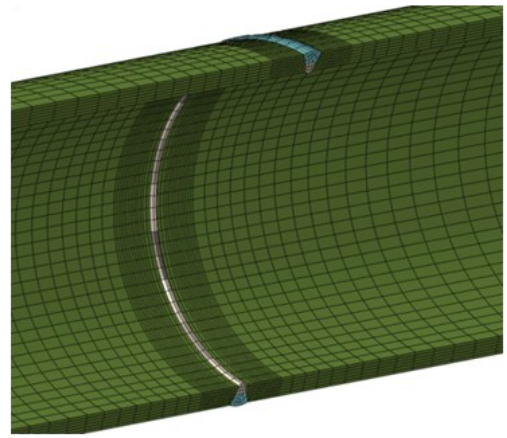

Figure 2: Half-pipe showing bead chunks

AWI also creates the corresponding stress analysis model from scratch, with the user simply adding adequate boundary conditions to represent the restraints on the parts. For the present problem, boundary conditions were applied only to prevent rigid body motion. The predefined temperature field from the previous heat transfer analysis is used by AWI automatically. The mesh of the 3-D model is shown in Figure 2. The AWI GUI is shown in Figure 3.

Results

All predictions from the axisymmetric and 3-D AWI-generated models are compared with the experimental data reported in [3]. Further, the comparisons are made at 180° from the weld start point. The axisymmetric model, shown in Figure 3, used a finer mesh.

Figures 4 and 5 show the temperature histories from the 3-D model on the inside and outside surfaces, respectively. The peak temperatures and cooling rates predicted by

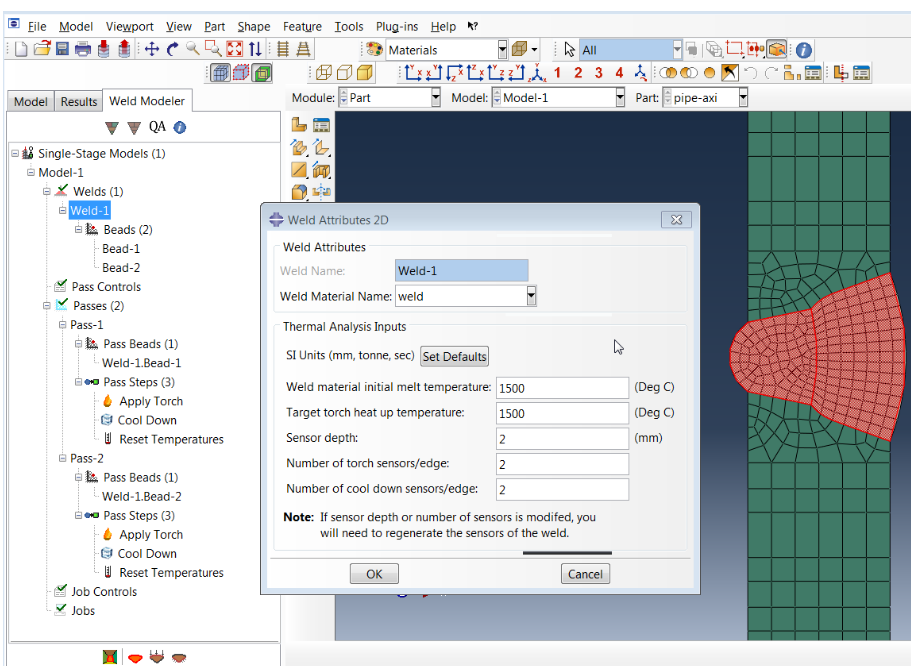

Figure 3: Abaqus Weld Interface dialog, axisymmetric model definition

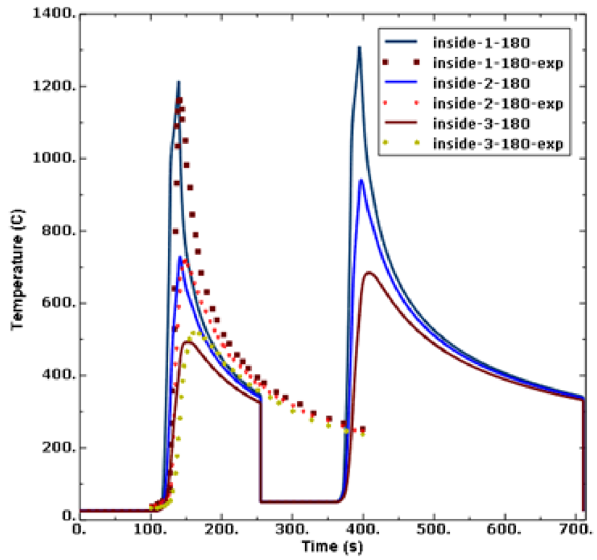

Figure 4: Temperature history, inner surface

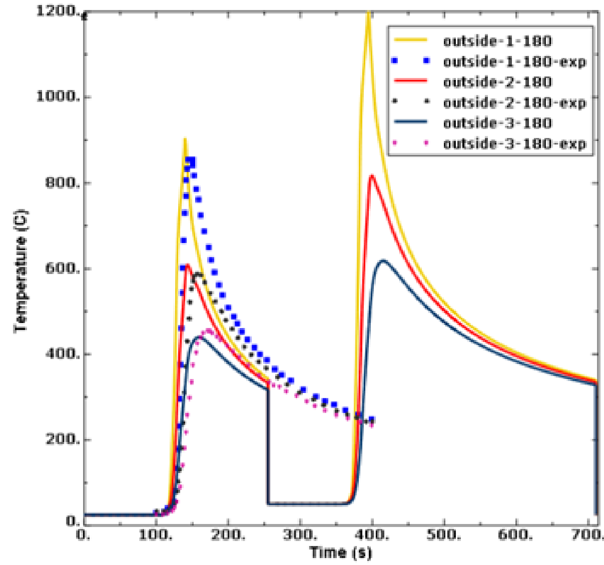

Figure 5: Temperature history, outer surface

Abaqus match well with the thermocouple data; note that the model results include both weld passes, with a reset of the temperatures to inter-pass conditions following the deposition of the last chunk of the first pass. It was also observed that there was no significant difference between the temperature histories at different angular locations around the circumference of the pipe in the 3-D model, confirming that there is relatively little heat conduction occurring in the hoop direction for the given welding speed.

Figures 6 and 7 show the axial and hoop stresses, respectively, on the inside surface of the pipe across the weld centerline. As with the temperatures, the 3-D stress results were found to be very similar at different angular locations.

While the inside axial stress is in fairly good agreement with the experimental values, the peak values of the hoop stress are under predicted. As noted in [3] and [5], there is a particular sensitivity of the circumferential analysis results to the weld metal constitutive model and properties. The current analyses assume the same plasticity model and properties for both the base and weld metals. Even with this approximation, the character of the residual stress pattern is captured.

The axial and hoop stresses on the outside surface of the pipe are shown in Figures 8 and 9 respectively. As with the inner surface results, the predicted hoop stress distribution shows a wider variation with respect to the experimental data than the axial stress distribution. The disagreement in the hoop stress magnitudes away from the weld zone follows the pattern reported in [3, 5] for flux-based simulation results. There, the authors postulated that the experimental values are much higher due to the initial residual stress introduced by the manufacturing process.

Conclusions

The Abaqus Welding Interface extends the capabilities of Abaqus/CAE to allow very rapid development of welding simulations. Multi-pass welding analyses can often in-

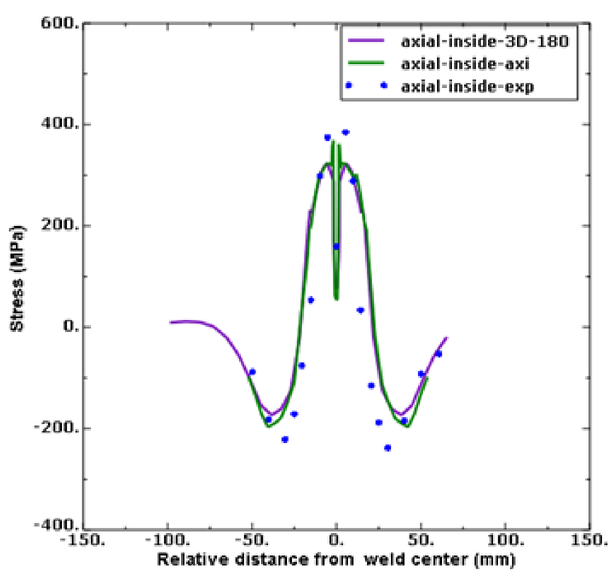

Figure 6: Axial stress, inner surface

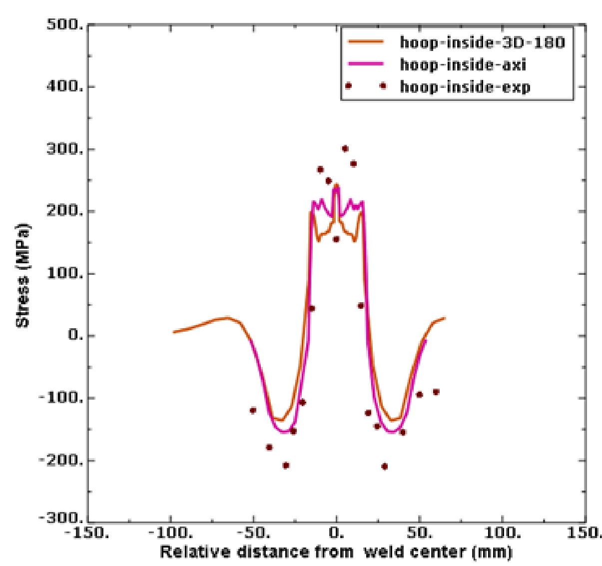

Figure 7: Hoop stress, inner surface

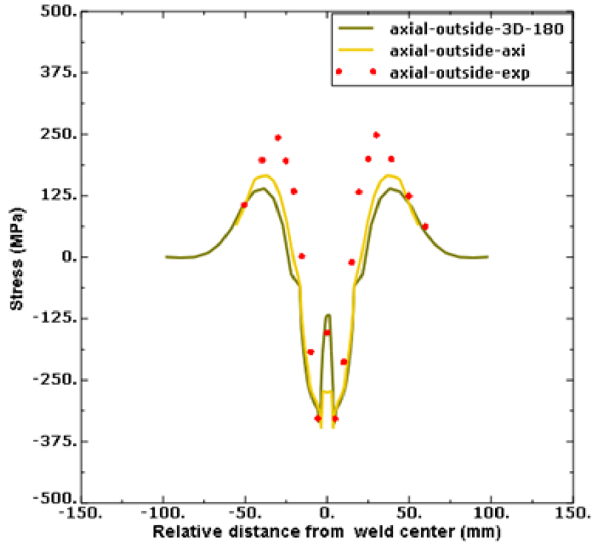

Figure 8: Axial stress, outer surface

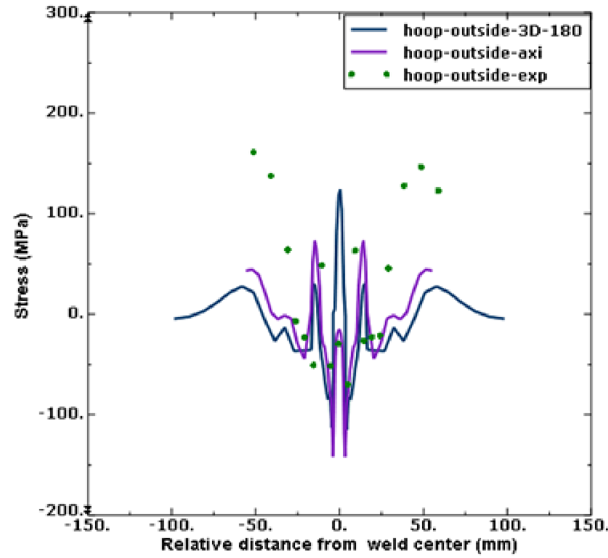

Figure 9: Hoop stress, outer surface

volve dozens of weld depositions; the AWI tools for automatically defining the associated Abaqus models, steps, loads, and boundary conditions afford significant time savings.

The capabilities of AWI to generate residual stress results have been presented in the context of a highly simplified model. The reasonable accuracy points to the utility of AWI as a means of efficiently performing design and trade-off studies, and with proper benchmarking and calibration, AWI may also be used for final design simulations.

Additional SIMULIA analysis tools such as Isight can be used in conjunction with AWI to study the various modeling parameters—torch temperature, temperature ramping options, number of weld bead chunks, torch speed, etc. With this approach, different classes of welding analyses can be calibrated and optimized; particular time savings can be realized with axisymmetric and plane strain models.

References

1. M. Shubert and M. Pandheeradi, ""An Abaqus Extension for 3-D Welding Simulations", to appear in the Proc. of the 9th International Conference on Residual Stresses (ICRS 9), October 7 - 9, 2012, Garmisch-Partenkirchen, Germany
2. M. Shubert, M. Pandheeradi, F. Arnold, and C. Habura, An Abaqus Extension for Welding Simulations, SIMULIA Customer Conference 2010, Providence, RI, USA (2010)
3. D. Deng and H. Murakawa, "Numerical simulation of temperature field and residual stress in multi-pass welds in stainless steel pipe and comparison with experimental measurements", Computational Materials Science 37, 269–277 (2006)
4. H. J. Rathbun, L. F. Fredette, P. M. Scott, A. A. Csontos, and D. L. Rudland, "NRC Welding Residual Stress Validation Program International Round Robin Program and Findings", Proceedings of 2011 ASME Pressure Vessels and Piping Division Conference (PVP2011), July 17-21, 2011, Baltimore, Maryland
5. D. Deng, H. Murakawa and W. Liang, "Numerical and experimental investigations on welding residual stress in multi-pass butt-welded austenitic stainless steel pipe", Computational Materials Science 42 234–244 (2008)
6. D. J. Dewees, "Comparison of 2d and 3d welding simulations of a simple plate", Proceedings of the ASME 2012 Pressure Vessels & Piping Division Conference, July 15-19 (2012), Toronto, Ontario, Canada
7. F. Arnold, and M. Pandheeradi, "Low Stress Welding Simulations," Desktop Engineering, October 2005

Abaqus References

For additional information on the Abaqus capabilities referred to in this document please see the following:

- Abaqus extensions: http://www.3ds.com/products/simulia/portfolio/abaqus/abaqus-portfolio/abaqus-add-ons/extensions

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors.

Delivering Best-in-Class Products

CATIA
Virtual Product Design

EXALEAD
Information Intelligence

SOLIDWORKS
3D for Professionals

NETVIBES
Dashboard Intelligence

SIMULIA
Realistic Simulation

3DSWYM
Social Innovation

DELMIA
Virtual Production

3DVIA
Online 3D Lifelike Experiences

ENOVIA
Global Collaborative Lifecycle Management

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes in all industries in more than 80 countries. For more information, visit www.3ds.com.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA