Welding Simulation with Abaqus

Summary

Metal welding processes are employed in various industries. Gas welding techniques use the heat from a flame to melt the parts to be joined and a filler material simultaneously. Extreme thermal loading is applied to the parts being joined, and complex material responses are initiated. The steep, localized thermal gradients result in stress concentrations in the welding zone. Consequently, modeling and simulation of welding processes are often complex and challenging. In this technology brief the use of Abaqus for this class of problems is discussed and an example analysis is presented.

Background

Welding is a fundamental manufacturing technique used to join metal components. While there are a wide variety of welding processes, most involve the application of heat to induce coalescence of the metal in the adjoining parts.

The gas welding technique uses the heat from a gas flame to melt together the contacting edges of the parts being joined. A filler material may or may not be used.

The introduction of very high temperatures in the region of the welded joint causes steep temperature gradients in the structure. As a weld cools, residual stresses may be produced in the weld zone; such stresses may cause the structure to distort. The ability to predict the residual stress state allows for the prediction of final part shapes and a more complete understanding of how residual stresses may affect the load capacity of a structure.

The finite element models described in this brief represent structural beams. These structures are formed by welding plate sections together, rather than producing the section directly in a mill. The cross-sectional size of these components can make direct manufacture impractical.

The techniques discussed herein are based on a generic model but are transferable to a range of applications.

Finite Element Analysis Approach

The simulation uses a sequentially coupled approach in which a thermal analysis is followed by a stress analysis. The temperature results from the thermal analysis are read into the stress analysis as loading to calculate the thermal stress effects. The thermal analysis makes use of Abaqus user subroutines **DFLUX**, **GAPCON**, and **FILM**. The objective of the simulation is to predict post-weld deformation and residual stress distribution.
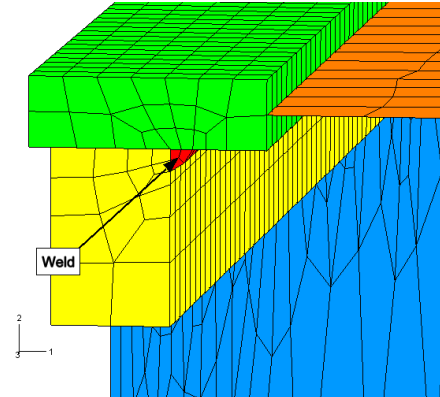
Key Abaqus Features and Benefits

- Combined thermal-mechanical analysis procedures, in sequentially coupled or fully coupled form

- Definition of transient heat sources and weld material deposition via user subroutines

- Ability to specify complex thermal and mechanical material definitions, including temperature and field variable dependence and latent heat effects

The model is constructed using solid and shell elements. A solid representation is used in the weld area to ensure more accurate capture of the high solution gradients. Regions outside the weld zone, where thermal gradients are not as severe, are modeled with shell elements to reduce the overall model size. The transition between the shell and solid regions is achieved using tied contact for the thermal analysis and shell-to-solid coupling for the stress analysis. The mesh is shown in Figure 1. The weld bead is shown in red.
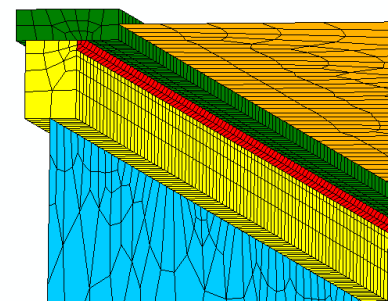

Figure 1: Mesh of the structure being welded

Thermal Analysis Procedure

The thermal analysis is performed to calculate the heat transfer resulting from the thermal load of the moving torch. Three user subroutines are activated for the thermal analysis, as described below:

DFLUX: Used to define the welding torch heat input as a concentrated flux. The heat source travels along the weld line to simulate the torch movement.

GAPCON: Used to activate the conduction of heat between the deposited weld material and the parent materials once the torch has passed a given location.

FILM: Similar to **GAPCON** in functionality, but used to activate film coefficients to simulate convective ambient cooling once the torch passes a given location.

The simulation is run as a fully transient heat transfer analysis.

Structural Analysis Procedure

The structural analysis uses the thermal analysis (temperature) results as the loading. The objective of the structural analysis is to determine the stresses and strains induced in the weld region during the cooling transient. Boundary conditions are applied to restrain the system against rigid body motion.

Results and Conclusion

Typical thermal results are shown in Figure 2, which displays a contour of nodal temperature as the torch travels along the weld line.

The weld is initialized at 1800°C, but no heat is allowed to transfer from the weld until the torch passes. As the torch passes a given point on the weld line, the flux input is initiated, resulting in a localized increase in temperature to over 2000°C.

In addition, as the torch passes the same weld line point, conductive and convective heat transfer is activated, causing a rapid drop in temperature as thermal energy is transferred to the surrounding structure and environment.

Figure 3 shows the 35 s transient temperature profile at three points close to the start of the torch path. The peak temperature is reached when the torch is activated. A sharp temperature drop is observed after the torch passes.

The stress response of the structure is driven by the high thermal gradients. Figure 4 and Figure 5, respectively, show plots of the von Mises stress and the effective plastic strain approximately 35 s into the process.

In conclusion, Abaqus/Standard provides a set of general, flexible modeling tools that allow for the prediction of residual stresses and final shapes in welded components.
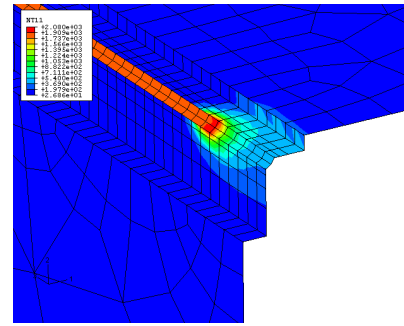

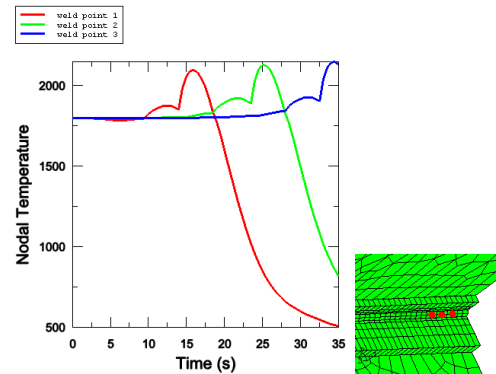*Figure 2: Nodal temperatures in a typical region of the weld as the torch traverses the weld line*

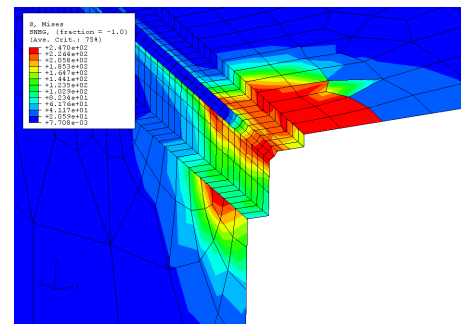*Figure 3: Transient temperature profile at three points in the torch path*

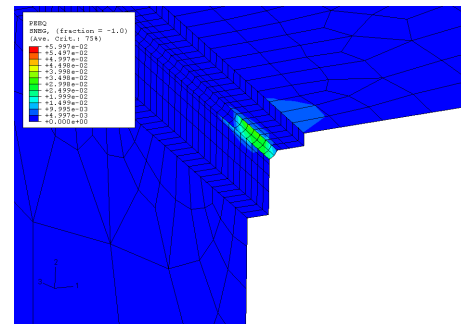*Figure 4: Von Mises stress in the weld region at t=35 s.*

Figure 5: Effective plastic strain in the weld region at t=35 s.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.11 documentation references:

- Abaqus Analysis User's Manual
  - "Sequentially coupled thermal-stress analysis," Section 6.5.3
- Abaqus User Subroutines Reference Manual
  - "**DFLUX**," Section 1.1.3
  - "**FILM**," Section 1.1.6
  - "**GAPCON**," Section 1.1.10

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 Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com