

Modelling and Analysis of Welding Processes in Abaqus using the Virtual Fabrication Technology (VFT) Analysis Software developed by Battelle and Caterpillar Inc

Matt I. Dawson

Rolls-Royce plc, PO Box 2000, Derby, DE21 7XX, UK

Abstract: *The ability to accurately model welding processes in order to predict residual stresses and distortions is becoming increasingly important in the engineering industry. Abaqus can be used to model the welding process but this has been found to be considerably time consuming and requires a large number of assumptions to be made. Virtual Fabrication Technology is an analysis software suite that is designed to allow Abaqus to accurately model complex welding procedures. It was developed by the Battelle Memorial Institute in conjunction with Caterpillar Incorporated in the USA. The software is centred on a Graphical User Interface that is used to assign welding paths, parameters and material properties to FE models that have been imported from Abaqus/CAE. A closed form analytical thermal solver (CTSP) and advanced user material subroutine (UMAT35) are also included in the package. The final structural analysis of the welded component is carried out using Abaqus/Standard. This paper presents the results of an extensive evaluation of the VFT software package conducted by Rolls-Royce Marine. This was carried out in order to ascertain the capabilities of this software and to determine how it improves on the weld modelling methods currently available using Abaqus stand-alone. This evaluation has shown that VFT significantly enhances the ability of Abaqus to model welding procedures while providing significant savings in both the user and computer time required. A large benefit is seen in the further integration of this software into the Abaqus/CAE package.*

Keywords: *Assembly Deformation, Design Optimization, Experimental Verification, Forming, Heat Transfer Optimization, Post processing, Residual Stress, Seam Welding, Thermal Stress, Third-Party Software, Visualization, Welding.*

1. Introduction

There is currently a large market for Finite Element (FE) software designed specifically for modelling welding procedures. This is mainly due to the increased interest in welding residual stress and distortion that has occurred over the past decade. Most of the new welding simulation software packages currently available are designed to act as stand alone packages. Many engineering companies though require a welding simulation solver that works and interacts with their current FEA software. This allows their current analysts to work with the new software

without having to undergo extensive re-training. Also, for consistency, it is desirable to use the same FE package for all of the various structural and thermal analyses that are performed.

This paper presents an evaluation of the ‘Virtual Fabrication Technology’ (VFT) welding analysis software developed by the Battelle Memorial Institute and Caterpillar Incorporated in the USA. It assesses the functionality and capability of this software and determines how it improves on the current abilities of Abaqus/CAE. It gives examples of the type of analyses that VFT is capable of performing and compares some of these analyses with previous FE methods. The high-temperature material modelling methods used by VFT are evaluated and the improvements to the Abaqus/Standard solver highlighted.

The VFT software has been designed to be capable of analysing many different types of welded joints and components by working in conjunction with Abaqus/CAE and Abaqus/Standard. Of particular interest to Rolls-Royce Marine is the ability of the software to model through thickness residual stress distributions in thick-section welded components. VFT has already seen use in the US Shipbuilding and Power Generation industries.

2. Background

2.1 Welding Residual Stress

Whenever a welding procedure is applied a metallic material is subjected to very large temperature gradients. Melting is usually achieved by the application of an electric arc or oxy-acetylene flame, resulting in a rapid local heating of the metal in the area of the weld. The large temperature gradients generated produce significant local plastic strain around the welded area. This results in the formation of large residual stress fields upon cooling in the local area of the weld and surrounding structure. As these stresses are not caused by the direct (and continued) application of any externally applied load, they are known as ‘residual stresses’ and are generally self-balancing within the structure.

The presence of residual stresses in a welded component can cause a number of problems. The component will be subject to local and global distortion caused by the effects of the stress/strain fields. The typical example of this is a groove weld on a flat plate, where contractions and plastic strains have caused the edges of the plate to bow upwards. Tensile residual stress fields will also encourage the initiation and propagation of defects in a material, which can be a major safety concern in safety critical components.

Experimental techniques can be used to determine residual stress levels at specific locations on engineering components. However, these techniques rarely provide a complete map of the residual stress and strain fields present in a welded component. Each measurement technique has its own capabilities and limitations. For example, the strain relief methods requiring drilling or cutting are either semi or fully destructive. Non-destructive measurement methods such as X-ray or neutron diffraction are typically laboratory based and applicable to only small welded specimens. The costs of through thickness stress measurements are also much greater than those incurred from performing a finite element welding simulation.

It is for these reasons that now, with the continued increase in computing power worldwide, many companies are starting to focus on the development of validated residual stress FE modelling methods.

2.2 Weld Modelling Software

Finite element packages have been used to model welding procedures for about twenty-five years. However it is only comparatively recently that specialised, user-friendly, welding simulation packages have come onto the market. Virtual Fabrication Technology (VFT) is a good example of such a modelling tool. Before the advent of these specialised programs, the user had to manually modify existing FE packages in order to set up an adequate welding simulation.

Welding specific software packages provide a number of advantages over this manual modelling method. The use of unique Graphical User Interfaces (GUI's), heat source mapping tools, automatic element activation and material modelling subroutines significantly increase the ease with which welds can be modelled. It should be noted though that, despite this automation, the basic welding analysis carried out remains relatively unchanged. It will still consist of a thermal analysis that is either coupled or un-coupled to a highly non-linear structural analysis.

The majority of welding analysis software favours the use of an un-coupled analysis procedure. In welding simulation this allows the thermal analysis model to be checked and validated before the time-consuming structural analysis is performed. Welding fusion and heat-affected zones can be checked against weld macrographs and, if necessary, the predicted temperature distribution can be compared to measurements gained from thermocouples. The data from the thermal model can then be used to conduct a structural analysis of the welded component.

Although this is theoretically a simple procedure, in practice the accurate modelling of the weld path and deposition can be complex and time consuming. Without specialist software it is not easy to successfully create a model that simulates the deposition of molten weld metal in a component without making several large compromises (or simplifications). As an example, the following section details the methods that are generally employed when using Abaqus/Standard and Abaqus/CAE to conduct a welding simulation.

2.3 Current Weld Modelling Methods using Abaqus

When modelling a welding procedure using Abaqus there are two distinct approaches that are available to users. The first of these is the construction of a 2D axisymmetric or shell FE model of the component in question. The second involves the construction of a full 3D solid model so that a 3D block-dumped or moving heat source analysis can be performed. A block-dumped analysis is one where the weld bead is deposited in several discrete steps. As a lot of literature on applying these methods is already available they will only be summarised in brief here.

The 2D axisymmetric method of modelling welds (Figure 1) is commonly used where a weld either has a large number of beads or is of the circular or girth type. The model represents a cross-section of the welded area with the entire weld removed as its initial state. The beads are added to the model one at a time at the material molten temperature. A DFLUX routine can also be used to provide a better simulation of the power of the welding torch. While the geometry for this method

can be set up in Abaqus/CAE the majority of the step data has to be entered manually via the input file.

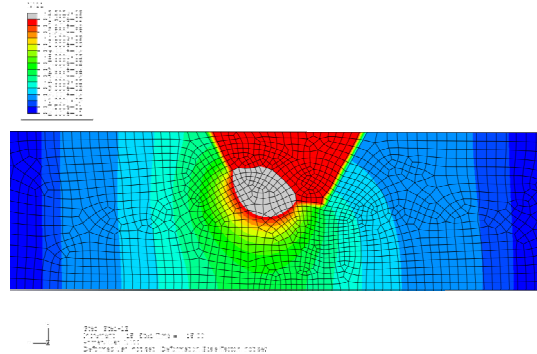


Figure 1. 2D Axis-symmetric model of an eight-pass butt weld

It is also possible to model the welding process in 3D using Abaqus/CAE and Abaqus/Standard. A 3D model provides a more accurate prediction of residual stress (especially in the direction of torch travel) and also allows factors such as weld stop/start effects to be taken into account. A block dumping technique is generally used to model the welding process with each bead being deposited in several segments. It is possible to simulate the path of a moving heat source but this requires the user to write a subroutine (or similar) that sets up a DFLUX routine for the particular model. Again, most of this work has to be done by manual editing of the input file.

2.4 Limitations of the Current Abaqus Methods

The main problem with the methods described above is the amount of manual manipulation of the input files required. Many of the commands associated with the weld modelling procedure are not supported by the Abaqus/CAE interface. This results in weld models that cannot be easily modified or changed without a large time investment by the user.

The 2D and 3D block dumping methods also contain several large analytical simplifications. In reality the weld metal is deposited in a continuous process, not in discrete steps. This discretisation can result in a residual stress field that has harmonic variations in its magnitude corresponding to the location of each deposited lump. Also, the block addition of large amounts of hot metal to a cold model does tend to cause convergence issues with the Abaqus thermal solver and has resulted in analyses failing to complete.

Problems are also encountered in relation to the heat source modelling employed. In both the 2D and 3D block dumping method no allowance is made for the shape of the heat source, or the fact that a ramped and varying heat flux input will occur as the torch passes a given location on the weld path. A 3D moving heat source model does resolve most of these issues but requires extensive editing of the input file as well as a modified user subroutine unique to each analysis. In experiments that were carried out run times were found to increase significantly when using this method.

3. Virtual Fabrication Technology

3.1 Introduction and Description

Virtual Fabrication Technology (VFT) is a welding simulation software package that has been developed by Battelle and Caterpillar in the USA (Brust, 2006). Its primary function is to simplify the modelling of residual stresses and distortions in welded structures by use of a user friendly GUI and advanced thermal and material modelling techniques.

Development started on the VFT software package in 1995 as a joint partnership between Caterpillar and Battelle (Yang, 2003). The software has since been implemented across Caterpillar and has been adopted by several other US companies, many in the marine and shipbuilding industries. In the UK, TWI and Rolls-Royce are the main users of this software (Bagshaw, 2007). Battelle have been running a rolling update/improvement program for VFT since 2001 and they intend to continue this into the future (Brust, 2006).

VFT is designed to run in either a Unix or Windows based PC environment. It requires at least version 6.5 of Abaqus/Standard and Abaqus/CAE to be available on the same machine for the full functionality of the software to be gained. The software package itself does not require very high system specifications to run, but run times will be reduced with higher spec machines.

The Graphical User Interface (GUI) is a Windows based interactive program that uses the standard Windows layout and functionality. The remainder of the programs contained in the VFT package are run via a command prompt, in a similar fashion to Abaqus/Standard.

3.2 Components - Graphical User Interface

The VFT GUI is used to define weld passes, material properties and welding parameters through use of its interactive 3D interface (Figure 2). Models are imported into the GUI from Abaqus input files and can then be manipulated and rotated in a 3D environment similar in its presentation to Abaqus/CAE.

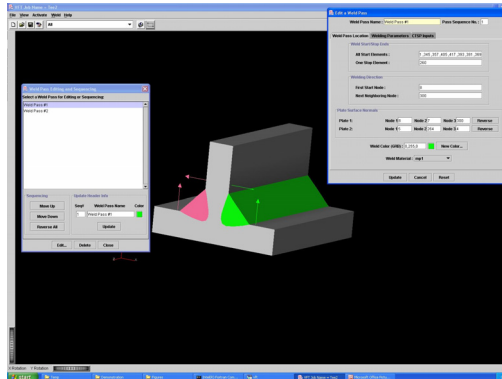


Figure 2. The VFT Graphical User Interface.

The GUI can be used to define welding passes on a model in a number of different ways. These vary from a simple text input method to a 'point and pick' selection method using the 3D graphical representation of the welding model. Welding parameters can be entered into a database and then assigned to as many weld passes as necessary. Material properties are assigned in a similar fashion.

A variety of display options allow the user to edit how the model is viewed and what nodes and elements are subject to labelling. The orientation of the model is controlled by a series of interactive buttons and drop down menus. Individual weld passes and model sections can be removed from the viewpoint for clarity of presentation.

The GUI saves files in a unique "file.vft" file format. This allows 3D representations of a welding model to be saved and loaded back into the GUI at a later date for further editing. The input files for both the thermal and structural analysis runs are generated automatically by the GUI. The user has the ability to change the naming procedure used for these files but they are all exported to the directory in which the .vft file has been saved.

3.3 Components – CTSP Solver

The Comprehensive Thermal Solution Procedure (CTSP) program is a closed form analytical thermal solver developed and distributed by Battelle (Brust, 2006). The input files for it are automatically generated by the VFT GUI software but are in a text format that can be edited by the user if required. CTSP is executed through a command prompt window. It outputs progress to this window in the style of Abaqus/Standard.

CTSP creates several results files detailing nodal temperatures at different time steps. These results files are used as an input for the Abaqus/Standard structural analysis.

CTSP is an extremely rapid thermal solver. This is primarily due to it using an analytical rather than a numerical solving routine (Brust, 2006). Also, the ability of the user to set a 'cut off' temperature below which nodal temperatures will not be calculated stops the software spending hours calculating temperatures in the far field region that are of no consequence to the analysis of the weld.

CTSP is capable of providing a thermal solution for the vast majority of known welding joints and configurations. It is also possible to select if a 'moving arc' heat source, or a 'block dumped' deposition sequence is to be used. These (and other) parameters are all set using the VFT GUI.

There are two main CTSP programs packaged with VFT: CTSP-Solid and CTSP-Shell (Brust, 2006). These are used to analyse 3D solid and shell models respectively. Several other CTSP routines are contained in the software package for the analysis of new welding methods including Laser and EB welding.

3.4 Components - CTSP-Temp

This program is an add-on to the CTSP solver routine. It allows an Abaqus odb file to be generated directly from the CTSP output thermal data. This allows thermal results to be viewed in Abaqus/Viewer before the structural analysis is initiated.

The program takes the form of an executable file that is run from a command prompt window. As with the CTSP program, data is output to this command window to update the user on the progress of the generation of the odb file.

3.5 Components - UMAT35-Weld

This software is an Abaqus/Standard user subroutine developed by Dr Pingsha Dong in conjunction with Battelle and Caterpillar Inc. (Dong, 1997). It is an extensive high-temperature material modelling UMAT that modifies the Abaqus/Standard solver routine to take into account factors such as annealing, phase change and different material hardening models.

The UMAT35 routine is designed to be used as a ‘black box’ analysis tool. Its code does not need modification by the user at any point. All information relating to a specific welding model (annealing temperature etc) is output from the VFT GUI in the form of a Fortran object file. This object file is then combined with the UMAT35 routine by use of a small executable file ‘ucombine.bat’ that is packaged with the software. Thermo-mechanical material data is contained in a separate user generated text file.

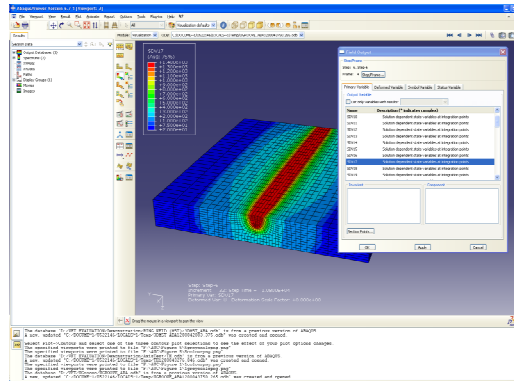


Figure 3. Post-processing using the Solution Dependent Variables

Three different types of annealing can be modelled using the UMAT routine. These are: no annealing, a single annealing point and progressive annealing. The single annealing point option requires a temperature to be specified by the user at which complete strain hardening history annihilation will occur. Progressive annealing requires two temperatures to be specified by the user. The lower of these signifies the temperature at which material hardening will start to decrease. The upper signifies the strain hardening history annihilation point (Dong, 1997).

The UMAT35 routine brings together the thermal, model and material data during the Abaqus structural analysis run. As a result of this it does not output any progress data or error messages. These are all handled at a higher level by the Abaqus/Standard software.

The UMAT35 routine automatically outputs a number of ‘solution dependent variables’ into the Abaqus odb file. These are designed to aid in post processing analysis and presentation work. They include such variables as maximum temperature reached, extent of fusion and heat affected

zones, areas where yielding has occurred and detail on the phases of the welded materials (Brust 2006, Dong, 1997).

3.6 Use of the Software

VFT uses Abaqus/CAE as a pre and post-processor and Abaqus/Standard as a structural analysis tool (Figure 4). The VFT software itself is used for the weld definition, thermal analysis and high temperature material modelling. The basic procedure for constructing and running a welding simulation model is outlined below.

The initial model will be constructed using Abaqus/CAE. This model will be meshed and element sets assigned that detail the region to be welded. The model will then be written to file as an input file. This file can then be imported into the VFT GUI where the 3D interface is used to define the weld passes, welding parameters, material properties and heat source information.

The GUI can then export the input files required to run the CTSP thermal solver routine. This solver produces a series of text files detailing the results of the thermal analysis in a node/temperature/time format. At this point CTSP-Solid-temp can be used to write an Abaqus odb file of the thermal results if required.

The GUI is then used to generate the necessary input files required for the structural analysis run. The UMAT-35 subroutine is combined with the relevant data for the particular model during this process. The structural analysis is then run using Abaqus/Standard and the UMAT-35 subroutine. Results from this analysis are written to an Abaqus odb file. A series of welding specific solution dependant variable and field variables are output. Post-processing is then carried out using Abaqus/Viewer.

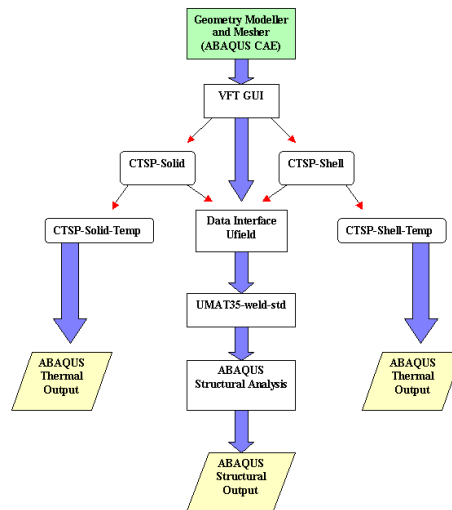


Figure 4. Basic process of modelling a weld using VFT

3.7 High-Temperature Material Modelling

Through the use of the UMAT-35 subroutine, VFT significantly expands on the high temperature material modelling abilities already present in Abaqus/Standard. The subroutine can model a variety of different material hardening models, progressive annealing, phase transformation and other high temperature material effects. The material models not only allow a more accurate welding model to be run, but it has been noted that the performance of the Abaqus/Standard solver routine is improved.

The way that material property data is handled by the UMAT35 subroutine is somewhat different to the way that material data is usually input and used in an Abaqus/Standard analysis. The material property data is contained in a user generated text file that is separate to the main Abaqus input file. To allow the data to be successfully read by the subroutine, the formatting of the property file is also different from the usual Abaqus/Standard format. The amount and complexity of material property data that needs to be generated by the user varies depending on which material hardening model is being used.

Isotropic hardening is the baseline material model used, but bi-linear and multi-linear kinematic hardening can also be modelled. Elastic-plastic creep effects can be applied if required and a complex phase transformation model can also be run. The phase transformation model requires the user to input a large amount of data relating to continuous cooling transformation curves. Also the physical and mechanical properties of each of the different phases are required at various temperatures. Although this procedure was found to be time consuming, once the property file has been written it can be re-used for any analysis by simply indicating where the material text file is using the VFT GUI.

It should be noted that it is not currently possible to model phase transformation in Abaqus. Internal research has shown that the inclusion of phase transformation in models can significantly alter the distribution and sign of calculated residual stresses in ferritic steel welds. The high temperature material melting/re-melting and the type of progressive annealing modelled by the UMAT-35 solver can also not be modelled in Abaqus stand-alone.

In the VFT manual it is stated that the UMAT-35 subroutine increases the pace at which Abaqus/Standard can run a welding analysis as well as the chance of a successful outcome. In the course of the evaluation procedure this has been found to be the case. Using the Abaqus/Standard solver to model high temperature procedures has (in the past) been found to result in multiple time step cutbacks due to the large thermal strains present. These cut backs cause the analysis to take longer than expected and often result in the analysis terminating with an error. The UMAT-35 subroutine does seem to enhance the ability of the Abaqus/Standard solver to converge on a solution. This provides a significant time saving when complex welding processes are modelled.

3.8 Demonstration of Capabilities

In order to ensure that the VFT software package is capable of modelling the wide-variety of components that are manufactured at Rolls-Royce Naval Marine, several complex FE models were constructed. These were designed to evaluate the ability of VFT to cope with correctly defining and modelling welds on a variety of large and complex geometries. The models were each

designed to test a different aspect of the Abaqus/VFT modelling process. The results from these studies are outlined below.

In order to assess the ability of VFT to model circular weld paths a partial penetration 3D ring weld was modelled (Figure 5). This model consisted of a pipe welded to a plate by an eight-pass attachment weld. In the past these types of welds have almost always been analysed using axisymmetric models, which results in a significantly conservative residual stress prediction.

No problems were found with the ability of VFT to model circular welding paths in 3D. With the minimum of input into the GUI, VFT was able to successfully map the circular paths of all eight passes of the weld. There were no solver problems encountered while running Abaqus/Standard with UMAT-35 and the analysis did not take long to complete. Previously, where Abaqus stand-alone has been used to model 3D circular welds, the analysis has had to be abandoned.

A model depicting a large butt-weld between two thick plates was also run. On large pressure vessel components, welds between plates can consist of forty or more individual beads. For this reason it is vital that VFT can model these large welds without a significant slowdown occurring. In this case the weld modelled was a sixty-eight-pass butt weld.

The model was set up in the usual way using the VFT GUI. Some slowdown was noted in the latter stages of the thermal analysis but the overall run time was not substantially increased because of this. The structural analysis using Abaqus/Standard took over a day to complete, but did not suffer from the dropouts and cutbacks that would have occurred in an Abaqus stand-alone analysis.

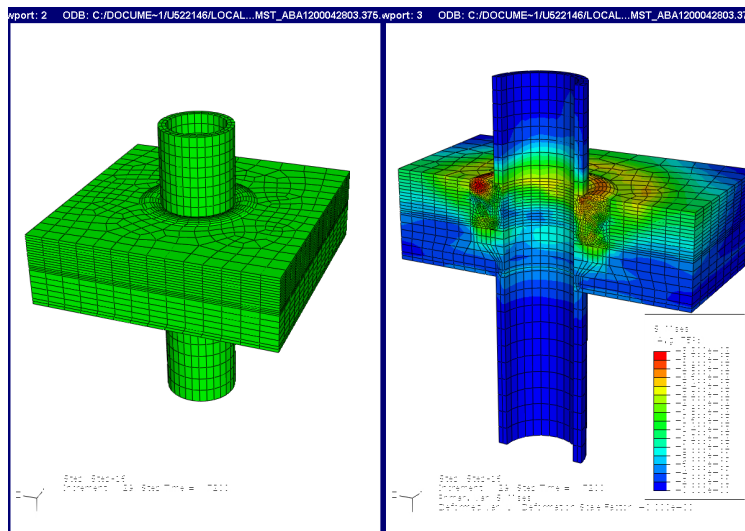


Figure 5. Eight-pass partial penetration ring weld model.

One of the main requirements for any software that relates to welding design is the ability to be able to perform multiple sensitivity studies in order to optimise a welding procedure. The CTSP thermal solver packaged with VFT has the ability to model multiple welders working

simultaneously on a large welded structure and the VFT GUI allows the easy re-ordering of the actions of these welders.

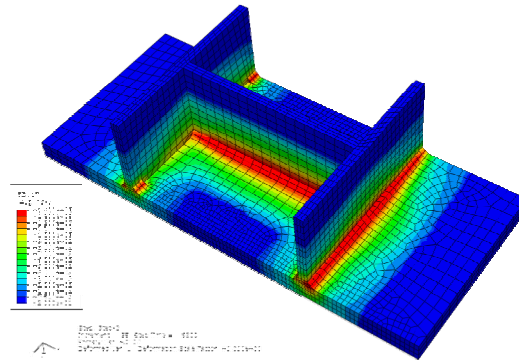


Figure 6. Bulkhead structure consisting of tack welds and fillet welds.

A model was constructed that consisted of a ten-pass sequence of single-pass welds on a bulkhead structure (Figure 6). Several different welding sequences were modelled, each involving two welders working simultaneously. The re-ordering of the welding sequence for each analysis run was found to be very easy to accomplish. The welding sequence can be swapped around using the VFT GUI without any further input of data required. This is in contrast to Abaqus stand-alone, where it is usually necessary to manually edit the input file to re-order a deposition sequence of a welding model.

4. Interaction between VFT and Abaqus

4.1 Overview

The VFT software suite is designed to work with Abaqus/Standard and Abaqus/CAE to produce high quality FE models of welding procedures. The ability for VFT to work successfully with these programs throughout the analysis process is an area that was subject to a large amount of scrutiny. This section details how VFT interacts with Abaqus/CAE and Abaqus/Standard and evaluates the software's ability to work with Abaqus in all stages of an analysis.

4.2 Model Generation and Import

FE models for use with VFT can be generated using Abaqus/CAE. All that is required from this initial model is that it contains a mesh of sufficient density and element sets that cover all the areas of the model that are to be welded. This model then needs to be written to an input file for import into the VFT GUI.

VFT is capable of importing models written in either Abaqus import file (.inp) format or a generic .msh format that is unique to VFT. The .msh format allows models to be imported from other FE generation programs with a small amount of input file editing by the user. No problems were

encountered in the course of the evaluation with regards to the import of the original model from Abaqus/CAE to the VFT GUI. The ability to create the initial geometry of the model in Abaqus/CAE greatly increases the complexity of weld model that can be input into the VFT GUI.

4.3 Structural Analysis

The weld definition process and thermal analysis are carried out by VFT alone. The results from the thermal analysis are written to a text file in a node/temperature/time format. This format is read by the UMAT-35 subroutine during the final structural analysis, which is carried out using the Abaqus/Standard solver as a base. The VFT GUI generates all the input files for the structural analysis automatically and hence no user editing of the files is required. Model specific subroutine data is also output from the GUI and this is combined with the UMAT-35 template to produce a model-specific subroutine.

No problems were encountered during the evaluation process relating to the operation of the UMAT-35 subroutine. It was found to bridge the gap between the VFT GUI and the Abaqus/Standard solver allowing the user to concentrate on the outputs gained from the analysis, rather than the analysis procedure itself.

4.4 Results Output and Post-Processing

The UMAT-35 subroutine produces a different set of outputs to those usually associated with an Abaqus/Standard run. These outputs are designed to provide as much useful information as possible on the welding procedure modelled. They also contain data on the annealing and phase transformation that may be occurring in the material during welding. These modified outputs are known as solution dependant variables and are summarized in detail in the VFT user manual (Brust, 2006).

The welding specific outputs created by the UMAT-35 routine can be used in a variety of ways to aid post-processing. Thermal outputs such as the maximum temperature reached by each node in the model are useful if it is necessary to determine the depth of fusion of a weld or the maximum temperature in a certain area. Outputs that display the volume proportion of each material phase present in a material allow the user to determine where phase transformation has taken place and what the final phase make up of the component is.

5. Evaluation of what VFT adds to Abaqus CAE

5.1 Overview

In the course of the evaluation work carried out using VFT, it has become clear that the software expands significantly on the current limited abilities of Abaqus/CAE and Abaqus/Standard to model high-temperature fabrication procedures. VFT allows the easy modelling of 3D welding procedures to be achieved through the use of a GUI. This is something that is not currently possible using Abaqus stand-alone. This section highlights the improvements made to the Abaqus package by the VFT software suite and identifies areas where Abaqus/CAE and Abaqus/Standard would benefit from development.

5.2 Evaluation of Improvements to Abaqus/CAE

Abaqus/CAE is one of the world leaders in 3D FE modelling and analysis of engineering components. However, it has been shown to be lacking in its abilities to create and run realistic 3D models of welding procedures. With the current level of interest in residual stress and its analysis, quick and accurate ways of modelling welding processes are required. Abaqus/CAE does not currently meet with this requirement.

The VFT software package fills this ‘hole’ in Abaqus/CAE. The GUI that is the centrepiece of the software allows users to deposit welds onto their Abaqus/CAE generated models and run analysis using 3D moving heat sources. The ability to construct and run such models completely through a GUI is a significant improvement on the input file editing that is currently required when using Abaqus stand-alone to perform a similar task.

With the VFT GUI is possible to change the welding parameters, process and bead sequence through the use of drop down menus. This allows multiple sensitivity studies to be set up and run in a short time with relatively little user effort. This is another area where Abaqus/CAE could see improvement. Once a 3D welding analysis has been set up in Abaqus/CAE it is difficult to alter the bead sequence or other welding parameters without a large number of adjustments to the model and its input files. This is not ideal.

5.3 Evaluation of Improvements to Abaqus/Standard

The VFT software package enhances the Abaqus/Standard solver to enable it to deal with the high-temperature material modelling that is required for welding simulations to be performed. This results in more stable analyses and shorter run times (Table 1). It also modifies the Abaqus solving routine to enable it to take account of any phase transformation that occurs in a material during welding. These additions allow Abaqus/Standard to run welding analysis that produce results that are potentially much more accurate than would otherwise be possible.

Table 1. Comparison of run times for some of the test models.

Model	8-Pass Ring Weld	10-Bead Bulkhead Weld	Edge Beam Weld
Abaqus Thermal Run Time	25 hrs	7 hrs 18 min	38 min
CTSP Thermal Run Time	2 hrs 54 min	16 min	3 min
Abaqus Structural Run Time	52 hrs + (fell over)	16 hrs 30 min	4 hrs 8 min
Abaqus + Umat35 Structural run time	15 hrs 51 min	3 hrs 12 min	1 hr 48 min
Total Difference in Run Times	59 hrs 15 min +	20 hrs 20 mins	2 hrs 55 mins

The welding-specific solution dependent variables that are output by the UMAT-35 modified Abaqus/Standard solver allows a much more in depth analysis of a welding procedure to be conducted. They also allow the user to easily iterate a welding analysis by displaying outputs such as the level of fusion or the size of the heat-affected-zone. Abaqus/Standard would benefit from having such outputs included in the stand-alone software.

5.4 Future Potential for Further Integration

This evaluation has highlighted the current gaps in the ability of Abaqus/Standard and Abaqus CAE to model 3D welding procedures. VFT has been shown to fill most of these gaps by providing a GUI through which welding procedures can be modelled and a subroutine that significantly improves the high-temperature analysis capability of Abaqus/Standard. Welding residual stress is an issue that is unlikely to go away in the near future. Both Abaqus/CAE and VFT should develop to keep pace with the increasing interest in this subject.

The further development of both pieces of software to allow even greater integration between the two can only be beneficial. VFT is a powerful analysis tool but manages (like Abaqus/CAE) to keep its GUI interface simple and user-friendly. Abaqus/CAE has seen a large amount of development in recent years, but modelling of high-temperature fabrication methods is an area that still needs a lot of development.

The current abilities of VFT provide an excellent model for future developments in Abaqus/CAE that would meet with the requirements of the residual stress modelling community.

6. Conclusions

The VFT software package is made up of several separate programs. These have been found to work well together and all necessary file generation and output is done automatically. This allows users with only a general knowledge of FE software to adequately use the VFT programs.

The VFT GUI is a modern, easy to use interface that is capable of operating on either Windows based or Unix based platforms. It allows the user to define welding parameters and material properties without having to do any manual editing of the input files. This is a significant improvement on the methods available using Abaqus stand-alone.

The CTSP heat source-modelling tool included in the VFT package is capable of solving welding thermal models to a high degree of accuracy in a very short period of time. It is capable of modelling a wide variety of welds with the only real problems being encountered in certain types of girth weld. It significantly reduces the time required to set up and execute a welding thermal analysis.

The VFT software is capable of modelling several different material-hardening models. It is also capable of modelling the phase change process. Once a user has entered material data it can be used for any VFT analysis, significantly saving on time and allowing the user to build up a large material database. The material modelling used by VFT has been demonstrated to be significantly more advanced than that employed by Abaqus.

The UMAT user subroutine supplied with VFT integrates the thermal, material and geometric data. It allows the material models and CTSP data to be used in an Abaqus environment and has been shown to require no user editing, whatever the welding case. It models progressive annealing and material melting / re-melting and outputs a large amount of results data that is not available using Abaqus stand-alone.

The VFT software was designed to be used as an add-on for the Abaqus software package and has been shown to interact well with it. Data import and export to the Abaqus platform is smooth and without error. The ability to export the CTSP thermal output decks into an Abaqus format before a structural run is performed is another useful tool. The solution dependent variables generated by the UMAT subroutine for viewing in Abaqus allow any welded component to be extensively analysed.

The VFT software package improves substantially on the abilities of Abaqus to model welded components. The material and thermal modelling used by VFT is significantly more advanced than that contained in Abaqus. VFT is capable of modelling welded components that would not be possible using the Abaqus software alone. VFT significantly reduces the time required to perform any type of welding analysis. Its interface system ensures that users no longer have to be FE experts to model complex 3D welding processes.

The further integration of VFT and Abaqus/CAE would result in a program capable of modelling the vast majority of welding procedures in a quick, accurate and user-friendly manner. The current abilities of VFT provide an excellent model for future developments in Abaqus/CAE that would meet with the requirements of the residual stress modelling community.

7. References

1. Brust, F. W., "VFT: Virtual Fabrication and Weld Modelling Software, Version 2.00," User Manual, Battelle Memorial Institute, Columbus, Ohio, 2006.
2. Brust, F. W., "CTSP-SOLID: A Comprehensive Thermal Solution Procedure for Solid FEM Models," User Manual, Battelle Memorial Institute, Columbus, Ohio, 2006.
3. Brust, F. W., "UMAT-WELD Version 3.5: A Welding Specific User Material Routine Interfaced with Abaqus", User Manual, Battelle Memorial Institute, Columbus, Ohio, 2005.
4. Bagshaw, N., and J. Mawella, "Prediction and Control of Distortion During Welding in Ship Panels", TWI, UK, 2007
5. Yang, Y. P., Brust, F. W., and Z. Cao, "Virtual Fabrication Technology Weld Modelling Tool and its Applications in Distortion Predictions", ASME PVP Conference Paper, July 2003.
6. Dong, P., Brust, F. W., and J. Zhang, "A Constitutive Model for Welding Process Simulation Using Finite Element Methods", Battelle Memorial Institute, Columbus, Ohio, 1997.