A 3-D composite shell element model for residual stress analysis of multi-pass welds

Zhang J., Dong P., Brust F.W.
Battelle, USA

ABSTRACT
Within the framework of shell and plate theory, a composite shell element model is developed for modeling multi-pass welds in complex welded structures. A unified thermoplasticity constitutive law is also incorporated with special consideration of near-melting material behavior during welding. The composite shell model is particularly suited for capturing three-dimensional residual stress characteristics of multi-pass welds in pressure vessel and piping components. Furthermore, a combined analysis procedure using both global 3-D shell and local 2-D/axisymmetric models is also introduced for an improved prediction of local residual stress distributions in piping components. Its application in analyzing a repair weld in a girth-welded pipe is discussed in detail.

1.0 INTRODUCTION
Two dimensional plane strain/axisymmetric element models have been dominantly used in the open literature for residual stress analysis of welded structures. This is largely due to the fact that the 2-D/axisymmetric models can be used to simplify analysis procedure and reduce computational intensity. In doing so, however, 3-D weld residual stress features are ignored. In some cases, the inherent assumptions associated with 2-D/axisymmetric models are not appropriate. A typical example is repair welding. In a repair weld, the repair length is usually short with respect to its original weld length. The surrounding welded structure provides deformation constraints that significantly affect the formation of residual stresses due to repair. 2-D cross sectional models cannot be used to simulate these 3-D constraint effects directly. In such situations, a combined analysis procedure using both 3-D and 2-D models is preferred. Using this combination procedure, the global structural constraint effects can be directly modeled in the 3-D analysis. The constraint information obtained from the 3-D analysis can then be fed into a 2-D cross sectional model for detailed local analysis. In addition, other important 3-D residual stress issues can be addressed, such as the effects of repair weld length on the residual stress redistribution in the entire welded structure.

A full 3-D solid element model for welding analysis of a complex structure is impractical. Instead, a 3-D shell/plate element model is more preferable because of its advantages of computational efficiency and ease in performing pre/post processing. However, a major obstacle exists in applying traditional shell/plate models to welding
simulation, i.e., its limitation in modeling multiple pass welds in thick-shell structures. As such, multi-pass welds have been ignored in shell model analyses published in the open literature.

In this study, a composite shell modeling technique is proposed to simulate the effects of multi-pass welds using a shell/plate element model. The proposed shell element model is based on an idealization that a multiple pass weld can be treated as a laminated shell composed of sequentially-deposited layers. The temperature fields associated with the deposition of each weld layer (pass) were generated by introducing a novel weld heat flow solution procedure. These temperature solutions were then mapped onto the composite shell model for residual stress analysis. A special constitutive model and its numerical implementation were also developed for residual stress analysis using the composite shell model. The effects of sequential weld metal deposition were simulated using a consistent layer activation/deactivation scheme. To demonstrate its effectiveness, the proposed shell modeling technique was used to analyze a pipe repair weld. The analysis results provide insight on the development and characteristics of 3-D residual stress distributions in similar multi-pass pipe repair welds.

2.0 COMPOSITE SHELL ELEMENT MODEL FOR MULTI-PASS WELDS

2.1 Idealization of Multiple Pass Welds
In a multiple pass weld, the weld groove is filled by a sequential deposition of weld beads. In residual stress analysis, the deposited weld beads are usually grouped into a number of lumped passes. The sequence of lumped passes follows the deposition sequence of weld beads. In a 2-D cross sectional model, these lumped passes can be easily modeled using element groups. The process of sequential metal deposition can be simulated using an element activation/deactivation scheme [1] or a more robust procedure [2]. In a shell/plate element model, however, element discretization through the weld cross section (shell thickness-direction) is not possible. This inherent limitation makes it very difficult to simulate the effects of multiple pass welds in a shell/plate model.

In the following, we present a concept of using composite shell element model to simulate multi-pass welds in thick-shell structures. Using this concept, a multiple pass weld is idealized as a laminated composite shell of multiple layers as illustrated in Fig. 1. It is assumed that the weld is made by a number of weld layers that are deposited sequentially. Each weld layer can be regarded as a lumped weld pass. The weld groove geometry is modeled by a proper assignment of weld material to the corresponding layers of shell elements within the weld area. In doing so, a refined finite element mesh is required in the weld region. The resulting weld groove model is of a step-wised geometry as shown in Fig.1.

2.2 Composite Shell Element Formulation
The finite element formulation for structural analysis of isotropic shells can be found in many textbooks [3, 4, 5]. The basic assumption in the shell element formulation is that a material line originally perpendicular to the shell’s reference surface (mid-surface) remains straight throughout the deformation. For a laminated composite shell element, the deformation of the material points in each lamina follows this assumption, as shown in Fig. 2. Thus, the displacement field in a laminated composite shell can be expressed as
\[ u = \sum_{i} N^{(i)} u^{(i)} + \sum_{i} N^{(i)} \xi \frac{t}{2} \mu^{(i)} \alpha^{(i)} \]  

(1)

where \( N^{(i)} \) are shape functions of node \( i \) and are functions of \( \xi \) and \( \eta \) only; \( \mu^{(i)} \) are matrices of direction cosines of the thickness-direction vector at node \( i \); \( t \) is the total thickness of the laminated shell element, \( t = \sum t_k \); \( u^{(i)} \), and \( \alpha^{(i)} \) are nodal translation and rotation vectors, respectively. The resulting stiffness matrix of a laminated shell element can be expressed as

\[ K = \int \int \left( \sum_{k} \left( \frac{\partial}{\partial \xi} B_k^T E_k^{\varepsilon p} B_k \cdot \det J \cdot \frac{\partial}{\partial \xi} \right) \right) d\xi d\eta \]  

(2)

where \( \xi_{k-1} \) and \( \xi_k \) are local \( \xi \) coordinates at the two surfaces of laminar \( k \); \( B_k \) are strain-displacement relation matrices and are functions of shape function derivatives, local coordinate \( \xi \), and thickness-direction cosines \( \mu \); \( E_k^{\varepsilon p} \) are the material Jacobian matrices, \( \frac{\partial \Delta \sigma}{\partial \Delta \varepsilon} \), which relate stress incremental to strain incremental at integration points in lamina \( k \); \( J \) is the element Jacobian matrix due to traditional coordinate transformation. The material Jacobian matrix \( E_k^{\varepsilon p} \) is supplied from the constitutive model in the evaluation of the element stiffness matrix. For an elasto-plastic material, \( E_k^{\varepsilon p} \) is not constant but depends on stresses, strains, and loading conditions (loading or unloading). In such situations, an incremental approach, such as Newton’s iteration method, must be used in the solution procedure. The stress vectors are also supplied from the constitutive model for equilibrium iterations.

2.3 Constitutive Formulation

In the incremental thermoplasticity theory, the total strain increment is decomposed into the following additive form:

\[ \Delta \varepsilon^{Tot} = \Delta \varepsilon^e + \Delta \varepsilon^p + \Delta \varepsilon^\theta + \Delta \varepsilon^a \]  

(3)

where \( \Delta \varepsilon^{Tot} \), \( \Delta \varepsilon^e \), \( \Delta \varepsilon^p \), \( \Delta \varepsilon^\theta \), and \( \Delta \varepsilon^a \) are the total, elastic, plastic and thermal strain incremental tensors, respectively; \( \Delta \varepsilon^a \) is an “annealing” strain increment introduced to model near-melting material behavior during welding [6]. The introduced “annealing” strain is based on the postulation that the material tends to forget its deformation history at near-melting temperatures. Consequently, the accumulated elastic and plastic strains are released (annealed) at these high temperatures. The introduction of this “annealing” strain into the constitutive formulation is important for welding simulation. For instance, when a material point reaches the melting temperature, a virgin material state can be restored with the introduction of this “annealing” strain. Or else, the material plastic straining effects would be over-predicted, particularly for multi-pass welds.

Based on the strain increment decomposition in Eq. (3), the stress-strain relationship can be expressed as follows:

\[ \Delta \sigma = E \cdot (\Delta \varepsilon^{Tot} - \Delta \varepsilon^p - \Delta \varepsilon^\theta - \Delta \varepsilon^a) \]  

(4)

where \( \Delta \varepsilon^a \) is the plastic part of the annealing strain increment and is a function of accumulated plastic strain, temperature, and temperature increment; \( E \) is the temperature
dependent elastic stiffness tensor. The plastic strain increment is evaluated by assuming a Von Mises yielding criterion with the associated flow rule.

For isotropic hardening, the material Jacobian matrix can be expressed as follows [1]:

\[ E^{p} = c \mathcal{J} + (K - \frac{1}{2} c) I \otimes I - h S \otimes S \]  \hspace{1cm} (5)

where \( c \) and \( h \) are scalar quantities determined by equivalent plastic strain increment, yield stress, shear modulus, and strain hardening coefficient; \( K \) is the elastic bulk modulus; \( \mathcal{J} \) and \( I \) are the fourth and second order unit tensors, respectively; \( S \) is the deviatoric stress tensor; and the symbol ‘\( \otimes \)’ represents a dyadic product of two tensors.

### 3.0 NUMERICAL IMPLEMENTATION

The proposed composite shell element model can be implemented in some of the advanced finite element codes, e.g., ABAQUS [1]. However, due to the limitation of the built-in modeling capabilities in these codes, a further development of codes and procedures is required for both thermal and mechanical models. These codes and procedures are developed to interface with the general-purpose codes to implement the proposed shell element model.

#### 3.1 Thermal Model

In a multi-pass weld of thick-shell structure, highly nonlinear temperature gradients occur at the shell cross section during the welding of each pass. These nonlinear temperature gradients play a very important role on the residual stress characteristics at the weld cross section. In the current ABAQUS heat transfer analysis using shell/plate elements, these nonlinear temperature gradients cannot be accurately captured. This is because a non-uniform welding heat source distribution through the shell thickness cannot be accurately modeled. As a result, the temperature distributions are smeared out at the shell cross section.

To capture realistic temperature gradients associated with the welding of each weld layer, an analytic-based approach was used in the implementation of the thermal model. The approach is based upon solutions by Rosenthal [7] and Kasuya [8] (both steady-state and transient solutions) with modifications to model specific welding processes. Reflecting heat sources were used to account for structural boundary effects. The Kasuya's thermal solutions directly account for convective and radiation heat losses. Contributions by Rykalin [9] for start/stop effects of the weld torch were also incorporated.

This analytic thermal model permits accurate high-speed temperature predictions for the multi-pass welds. The resulting solutions are three dimensional and independent of the finite element mesh. To feed the temperature solutions into the shell element model for stress analysis, a mapping procedure was used. During mapping, the shell sectional integration points were treated as independent nodal points with coordinates calculated from nodal coordinates, shell normal direction, and shell thickness. This mapping process was automatically incorporated into the thermal analysis procedure. The output is a file containing temperature history associated with each weld layer at each sectional integration point.
3.2 Mechanical Model

For the mechanical model, ABAQUS built-in solution procedures cannot directly model the process of sequential weld layer deposition. However, ABAQUS provides an interface through a user material subroutine UMAT that allows the user to control the material behavior at each integration point. A special UMAT subroutine was then developed to implement the thermoplastic constitutive model outlined in Section 2.3. The mathematical structure of the developed UMAT subroutine is shown in Fig. 3. When the UMAT subroutine was used, all ABAQUS built-in constitutive models were no longer available. Therefore, a complete thermoplastic model was built in the UMAT to supply the material Jacobian matrix and stress tensors.

As shown in Fig. 3, the process of sequential weld layer deposition was simulated in the UMAT subroutine using a consistent layer activation/deactivation scheme. For a weld layer that had not been deposited yet, a very small material stiffness was assigned so that it would not affect any deformation mechanism at the shell section. This process is called the layer deactivation process. The deactivated layers can be regarded as virtual layers that were not present. After the weld layer was deposited, it would be reactivated and actual temperature-dependent material properties were assigned. Through such a consistent layer activation/deactivation process, the effects of sequential metal deposition in a multi-pass weld were appropriately modeled in a shell/plate model. In a repair weld, this layer activation/deactivation process can also be used to simulate the process of material removal before repair.

4.0 AN EXAMPLE: REPAIR WELD IN A PIPE GIRTH WELD

In this section, a multi-pass pipe repair weld was analyzed by using the proposed composite shell element model. The finite element mesh of the repair weld is shown in Fig. 4 and the composite shell element definition is shown in Fig. 5. As shown in these figures, the repair weld was 120 mm in length and 70 percent in depth of the wall thickness. A refined finite element mesh was used in the repair area and a symmetric boundary condition was applied along the weld centerline. Three weld layers were assumed at the weld cross section. A step-wised V-shape weld groove was approximately modeled.

As shown in Fig. 5, the analysis procedure for the repair weld was composed of the following four steps:

- Simulate the original girth weld (Fig. 5a). The original girth weld was simulated along the entire 360° circumference. Three lumped passes were assumed. The start/stop position of the original weld was located at 90 degrees away from the center of the repair weld.
- Remove the original weld material from the repair area (Fig. 5b). Two layers of original weld material were removed from the repair area. The corresponding removal depth was about 70 percent of the wall thickness.
- Deposit the first pass repair weld (Fig. 5c).
- Deposit the second pass repair weld (Fig. 5d).

The global distribution of the resulting axial residual stresses at the outer surface of the pipe after repair is shown in Fig. 6. A large tensile axial stress zone occurred at the outer surface near the repair area. At the two ends of repair area, compressive axial stresses occurred due to the axial shrinkage of the repair weld. The stress field at the start/stop
position of the original weld is similar. In the rest of surface away from the repair area and start/stop positions, the residual stress distributions are close to axisymmetric.

Figure 7 shows the axial residual stress distribution along the longitudinal cross section at the center of the repair area. Compared to the original weld (Fig. 7a), the axial residual stresses were increased after repair (Fig. 7b). This is particularly obvious at the outer surface where the high tensile stress zone became much wider after repair. The axial and radial displacements along the cross section were plotted in Fig. 8. These displacements can be feed into a 2-D cross sectional model as deformation constraints for detailed local stress analysis.

5.0 CONCLUSIONS

In this study, a 3-D composite shell element modeling technique is developed for modeling multi-pass welds in complex structures. The proposed modeling technique is based on an idealization that a multiple pass weld can be treated as a laminated shell composed of sequentially-deposited layers. The procedure for numerical implementation of the proposed modeling technique is also presented. The temperature fields associated with the deposition of each weld layer (pass) are generated by introducing a novel weld heat flow solution procedure. The process of sequential metal deposition is simulated using a consistent layer activation/deactivation scheme. A unified thermoplasticity constitutive law is also incorporated with special consideration of near-melting material behavior during welding. The proposed composite shell model is particularly suited for capturing 3-D residual stress characteristics of multi-pass welds in pressure vessel and piping components. As an example, a pipe repair weld was analyzed using the proposed modeling technique. The analysis results provide insight on the development and characteristics of 3-D residual stress distributions in the multi-pass pipe repair welds.

REFERENCES

Fig. 1. Idealization of Multi-pass Weld Using Laminated Composite Shell Model

Fig. 2. Laminated Composite Shell Element
Fig. 3. Mathematical Structure of UMAT for Multi-Pass Welds in Composite Shell Model

Fig. 4. 3-D Composite Shell Element Model For a Pipe Repair Weld
Fig. 5. Composite Shell Element Definition

Fig. 6. Axial Residual Stress Distribution at Outer Surface - After Repair
Fig. 7 Axial Residual Stresses along Longitudinal Cross Section of Repair Area

Fig. 8 Displacement along Longitudinal Cross Section of Repair Area