Welding simulation of ship structures using coupled shell and solid volume finite elements

Dongjin Kim
Lehigh University

Follow this and additional works at: https://preserve.lehigh.edu/etd

Part of the Mechanical Engineering Commons

Recommended Citation
https://preserve.lehigh.edu/etd/816

This Thesis is brought to you for free and open access by Lehigh Preserve. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Lehigh Preserve. For more information, please contact preserve@lehigh.edu.
Kim, Dongjin

Welding

Simulation of Ship Structures Using Coupled Shell and Solid Volume Finite Elements

January 2004
Welding Simulation of Ship Structures Using Coupled Shell and Solid Volume Finite Elements

by

Dongjin Kim

A Thesis
Presented to the Graduate and Research Committee of Lehigh University in Candidacy for the Degree of Master of Science in

Department of Mechanical Engineering and Mechanics

Lehigh University
December, 2003
This thesis is accepted and approved in partial fulfillment of the requirements for the
Master of Science

Date
12/04/03

Dr. Herman Nied, Thesis Advisor

Dr. Herman Nied, Chairperson of Department
# Table of Contents

<table>
<thead>
<tr>
<th>Chapter 1 - Introduction</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1 Introduction to Welding Distortion</td>
<td>2</td>
</tr>
<tr>
<td>1.2 Introduction to Welding Heat Sources</td>
<td>4</td>
</tr>
<tr>
<td>1.3 Introduction to Simulation of Welding</td>
<td>5</td>
</tr>
<tr>
<td>1.4 Application of Research (Introduction to Research concerning ship hull construction)</td>
<td>9</td>
</tr>
<tr>
<td>1.5 The objectives of present research</td>
<td>11</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter 2 - Comparison between the coupled model with shell elements and the 3D volume model of a fillet welded T-beam</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1 Properties of material (AL-6XN)</td>
<td>12</td>
</tr>
<tr>
<td>2.1.1 Thermal properties</td>
<td>12</td>
</tr>
<tr>
<td>2.1.2 Mechanical properties</td>
<td>14</td>
</tr>
<tr>
<td>2.2 Model generation</td>
<td>16</td>
</tr>
<tr>
<td>2.3 Thermal Model (thermal source modeling)</td>
<td>19</td>
</tr>
<tr>
<td>2.4 Mechanical Model</td>
<td>21</td>
</tr>
<tr>
<td>2.5 Thermal Results</td>
<td>23</td>
</tr>
<tr>
<td>2.6 Mechanical Results</td>
<td>29</td>
</tr>
<tr>
<td>2.7 Conclusion about Comparison of both models</td>
<td>43</td>
</tr>
</tbody>
</table>
Chapter 3 - Comparison of coupled elements models with different boundary conditions

3.1 Properties of material
3.2 Model generation
3.3 Thermal Model
3.4 Mechanical Model
3.5 Thermal Results
3.6 Mechanical Results

Chapter 4 - Welding simulation of a practical problem (the hull of ship)

4.1 Model generation
4.2 Thermal and Mechanical Model
4.3 Thermal Results
4.4 Mechanical Results

Chapter 5 - Conclusions

5.1 Future work
5.2 Conclusions

Appendix

HEAT.DAT
MECH1.DAT

Reference

Vita
Abstract

The objective of this research is to develop an efficient approach to determine the global deformations in ship structures resulting from thermal effects from the welding process by using coupled shell and solid finite elements. The finite element method has emerged as one of the most attractive approaches for computing residual stresses in welded joints, but its application to practical analysis and design problems has been hampered by computational difficulties that occur due to the enormous computational size of any practical problem. These difficulties arose primarily in situations with three-dimensional (3D) modeling of a welding process. Although two-dimensional (2D) modeling has been used widely in residual stress problems, current studies have shown that 2D analysis cannot render accurate residual stresses in many specific welding problems. Therefore, it is most effective to use shell elements in conjunction with solid elements. In the zone close to the heat affected zone, 3D modeling is repeated. However, use of shell elements away from this region will decrease significantly the number of elements in the welding model, reducing the computational size for the overall 3D model. This study investigates the temperature, distortion and residual stress in a fillet welded T-joint, comparing those computed by the coupled elements of both volume and shell elements, with those computed by volume elements only. In addition, the displacements in welded T-joints with different constraints were compared. The practical problem of a welded box beam used in ship hull design was simulated by using a coupled model of shell and solid elements.
Chapter 1 - Introduction

1.1 Introduction to welding distortion

Residual deformations and stresses will be generated in a structure as a consequence of the local plastic deformations introduced by the local temperature history associated with welding, i.e., rapid heating and subsequent cooling.

K. Masubuchi discussed the various types of welding-induced distortion and residual stresses. This research resulted in a number of empirical relations and focused entirely on welding distortion that remains after the completion of the weld and ignoring the intermediate states. [1]

In a general welding problem, residual stresses are produced by plastic strains due to tremendous thermal gradients, by material dilation during solid phase transformations, and by plastic deformations caused by plastic strains and solid phase transformations. [2] Near the weld pool, the temperature change due to welding is extremely rapid and the temperature distribution uneven. In the region of the weld, the molten metal supports no load and no strength of the solid, but high-temperature metal around the weld is drastically reduced. As the temperature far from the weld is relatively low, the expansion of metal near the weld is constrained and forced into high compression. Regions far from the weld are forced into tension to balance the compressive stresses close to the weld. When the part cools, the material near the weld contracts and results are in high tension, yielding occurs, while the regions far from the weld balance with compression.

Distortion due to welding can be divided with three categories. The first is transverse shrinkage, or shrinkage perpendicular to the weld line. With this type of
distortion, the part often contracts uniformly along the weld. (See Figure 1) The second is longitudinal shrinkage parallel to the weld line, and the final category is angular distortion about the weld line. With thin-walled structures, buckling is also an important problem. [1]

Figure 1 - Distortions in Welding

Until recently, most researches of weld distortion relations were empirical because the analytical solution of welding distortion was too difficult to be practical. While empirical relations based on experiments are useful for estimating distortion in parts similar to those used for deriving the relations, these solutions are available only for simple geometries. [3] All experimental methods have at least two disadvantages. First, their application usually requires special equipment and personnel that are not usually
associated with welding research. Second, residual stresses can most often be measured at discrete locations on a weld, usually close to the weld surface. A complete picture of the residual stress distribution in a general weldment is practically impossible to obtain by experimental techniques. [4]

1.2 Introduction to welding heat sources

Rosenthal had developed a solution for conduction from a moving heat source in the late 1930s. This has been the most popular analytical method for calculating the thermal history of welds. [5] However, as these models assume a point source and therefore infinite temperature at the source, the model breaks down close to the weld pool. One additional limitation of the Rosenthal solution, when applied to direct metal deposition, is that it does not include any mass addition to the weld pool. [6]

To overcome most of these limitations, Pavelic et al [7] suggested a heat source modeled with a Gaussian distribution of flux deposited on the surface of the workpiece in 1969. With this model, the concentration of the heat source can be varied by changing a parameter called the concentration coefficient. Friedmen, Krutz, and Segerlind [8-9] developed a variation of Pavelic’s model that is expressed in coordinates that move with the heat source. While these models are a significant improvement over Rosenthal’s model, it has been suggested that heat should be distributed to the molten zone to reflect more accurately the digging action of arc. These models do not account for the rapid transfer of heat throughout the fusion zone. To better represent high power density sources, a hemispherical Gaussian distribution was developed. Unfortunately, this model was still ill suited to deal with deep penetration welds that are not spherically symmetric.
To account for this problem, Goldak, Chakravarti, and Bibby [10] proposed a nonaxisymmetric three-dimensional heat source model. This model accommodates shallow welds, deep welds, symmetrical welds, and asymmetrical welds, all of which lead to more accurate models of the welding process. Thermal models of welding processes have also improved, taking into account parameters such as weld torch width, non-linearities due to variation of thermo-mechanical properties of material with temperature, radiation heat transfer from the weld pool, temperature-dependent convective heat transfer coefficients, and more. [2]

For the simulation of the arc welding process, a double ellipsoidal geometry of the heat source is used in this research. This approach is numerically more stable and more accurate than a point or line source, especially in the temperature range above 600°C.

1.3 Introduction to simulation of welding

Interest in developing adequate analytical models of welding processes dates from the late 1930s and 1940s. [11-12] To be sure, from the perspective of continuum mechanics, the welding process can be viewed as transient boundary value problem. The constitutive equations in this problem take into account the physics of heat transfer and the mechanics of thermal dilatation, as well as the processes of change in material microstructure and phase transformation. The boundary conditions model the welding heat input, the surface heat losses, the mechanical restraints, and, most importantly, the contact between the welded parts and the filler metal deposited. At a minimum, a temperature-dependent elastic-plastic material model should be incorporated. Practical solution of such a complex boundary value problem became possible only in the 1960s,
with the computer implementation of powerful numerical techniques, among which the finite element method emerged as the most powerful. [13]

The computational demands of fully 3D welding models are so prohibitive that all FE investigations of residual stresses in welds before the late 1980s were performed on simplified two-dimensional (2D) models. In 2D models, only a plane perpendicular to the direction of the weld is considered. The behavior out of plane can be taken as plane stress (assuming zero out-of-plane stresses), plane strain (assuming zero out-of-plane strains), generalized plane strain (assuming constant strain normal to the model plane), or axisymmetric. [5] Argyris et. al. [14] computed the thermo-mechanical response using 2D models in a staggered solution strategy to combine and integrate the thermal and mechanical computational steps. Rybicki et. al. [15] performed thermo-elasto-plastic analysis on a 2D axisymmetric finite element model for a two-pass girth-butt welded pipe problem, and verified the numerical results with the experimentally obtained temperature history and residual stress distributions. Papaxoglu and Masubuchi [16] solved the multipass GMAW process problem by performing uncoupled 2D heat transfer and stress-strain analyses, incorporating the phase transformation strains.

Since investigators tried to avoid modeling in 3D, the computed residual stresses were verified by comparison with experimental measurements. Certain discrepancies between computed and experimentally measured residual stresses were reported. This led to the belief that 2D models in certain situations are inadequate to quantify the residual stresses accurately in a welded joint. [4] 2D models, as mentioned above, have been particularly useful with their high efficiency and accuracy in determining the solution in
the analysis plane and reduced computational requirements. However, for welding practices where tack welding or fixturing allow out-of-plane movement 2D analyses may not be accurate. This seems to be particularly for distortion predictions. Furthermore, longitudinal heat transfer, instability aspects and end effects (i.e. due to initiation and termination of the heat source) cannot be realized in two dimensional formulations. [17]

3D modeling of welds was first attempted by Tekriwal and Mazumder [18-19] and, independently, by Karlsson andJosefson. [20] Their analyses confirmed the 3D nature of the temperature and stress fields developed during welding, but at the same time demonstrated the restrictions of the 3D calculations. The investigators were limited to relatively coarse FE meshes to accommodate the analysis in the computer facilities available to them. [4]

Most of the currently performed welding simulations, both 2D and 3D, are based on small deformation theory and are limited to simpler structures and weld geometries or focus only on the heat affected zone, ignoring the surrounding structure. [17]

Brown and Song [21] show that the interaction between the weld zone and the structure can have a dramatic effect on the accumulated distortion in many cases, the contribution of the structure dominates the state of distortion and stress, a state that is much different from the one predicted by a simulation of the weld zone alone.

As a full three-dimensional model is computationally expensive and unnecessary in many temperature and stress calculations, Daniewicz [22] developed a hybrid (experimental-numerical) approach that the weld joints are represented by “weld elements” to simulate the shrinkage caused by welding, which is determined
experimentally. This approach does not deliver the desired accuracy due to the difficulty in measuring weld shrinkage.

Michaleris et al developed a two-step numerical analysis technique for predicting welding-induced distortion and assessing the structural integrity of large and complex structures, that combines two-dimensional welding simulations with three-dimensional structural analysis in a decoupled approach. [3-23] First, a two-dimensional welding simulation is performed to determine the residual stress distribution. The model limited to a portion of the structure that represented the mechanical restraints that were used. Then a three-dimensional structural (elastic) analysis performed using the stress distribution of the welding simulations as loading to determine if the structure would buckle and the corresponding mode and/or magnitude of deformation. The advantage of a decoupled approach is computational simplicity and efficiency. Complex 3D welding simulations were not performed. It should be pointed out that Michaleris' approach has the difficulty of applying the accurate weld load, obtained from 2D model. The difficulty is due to the different mesh size, the limited region applied weld load to 3D model and the assumption that residual stress distributions are generated by imposing a strain as load. Although this method delivers reasonable results by using limited computer resources, a critical buckling load can be only predicted using decoupled 2D welding simulation and 3D eigenvalue buckling analyses. The effects of temperatures and distortions per time step cannot be calculated over predicted and the temperatures around the weld pool, since the two-dimensional model neglects conduction in the weld direction.
1.4 Application of Research (Introduction to Research concerning ship hull construction)

Many ships of current construction are of the conventional hull type, i.e., basically a single skin of steel plating stiffened orthogonally by stiffeners and transverse members. The double hull is a relatively new development in the evolution of ship structural systems. The fundamental difference from the conventional hull is that its twin skins of steel plate are separated from each other by longitudinal girders that span between transverse bulkheads along the length of the ship. The double hull offers some advantages over conventional hulls, such as improved combat and collision resistance, fewer areas of discontinuities and complex welded details, possibilities for automated fabrication techniques, and simplified distribution systems.

Recent advances in steel making have resulted in the development of new steels with improved material properties such as high yield strength, good weldability, good ductility and high corrosion resistance. Application of AL-6XN non-magnetic superaustenitic stainless steel in double hull ship structure could be essential for the development of future double hull construction because of the characteristics that this alloy begins to deform inelastically under relatively low stress level, but have more work hardening capability and higher ultimate strength than many plain carbon steels. [24]

Most solutions concerned with ship hull fabrication are numerical, i.e., based upon the finite element method. Because the determination of the ultimate limit state of the overall hull girder is a complex problem involving large deflection, and elasto-plastic behavior of the hull components. Much research has been carried out in developing a simplified model due to the expense and time consuming process of full modeling. Lu
and Pang investigated the ultimate strength of a ship hull under axial compression induced by vertical longitudinal bending without considering of strain hardening. [25] Thereafter, Lu developed an analytical capability to predict the load-deformation relationships and the ultimate strength of double hull cells under axial compression, which incorporated strain hardening due to the highly ductile nature of the AL-6XN. [24]

For the investigation the effects of welding in ship construction, Dydo et al. [26] researched buckling distortion that is caused by compressive stresses between stiffeners. 2D thermal finite element models of the weld cross section in conjunction with a 2D elastic-plastic mechanical finite element model were used to predict the longitudinal stress induced along the weld. The equivalent axial load was then applied to the 3D structural model and the Critical Buckling load was predicted.

In order to calculate the deformations per time step by local temperature history, Ramasy [27] investigated predictive methods of assessing the effects of the welding process using the finite element method to be used in welding simulation for ship structures. This study provided information to help decide on mesh densities which are feasible and sufficiently accurate.

Murugan [28] investigated the temperature and residual stress distributions per time step in back step welding process to reduce the residual stresses in welded joints of ship construction with 3D welding modeling of T-beam. However, these 3D welding simulations were limited to the simple structure of T-beam and weld geometries and focus primarily on the heat affected zone.
1.5 The objectives of present research

The prediction of residual stresses can be done on local models while the computation of global deformations requires the modeling of the whole structure. The mesh sizes involved with this kind of simulation are very important due the movement of the heat source and the very high temperature gradient near the weldment. This may lead to unreasonable CPU time. Due to this reason, the accurate analysis of global deformation is currently lacking. In fact, comparisons between the coupled model with shell and volume elements and the 3D model with only volume elements applied to the same weld problem are practically nonexistent.

This study couples in the same analysis shell and volume elements. This method enables one to simulate more efficiently the welding of thin structures as the solid volume elements can be limited to an area close to the heat affected zone, with the rest of the structure modeled using shell elements. Non-linear thermal and mechanical behavior is available in shells. The computation is optimized in the sense that the additional degrees of freedom associated with volume elements are largely eliminated. Compatibility elements must be defined to transfer rotations between solid and shell elements. [29]

The purpose of this research is to develop a comprehensive 3D finite element model and compute global deformation of the ship structure.
Chapter 2 - Comparison between the coupled model with shell elements and the 3D volume model of a fillet welded T-beam

2.1 Properties of material (AL-6XN)

AL6XN stainless steel (45Fe-25.7Ni-21Cr-6.3Mo) is one of the leading materials being considered for Navy hull fabrication. Therefore, this material was used in the following computations to compare results between shell and volume elements.

2.1.1 Thermal properties

Variation of thermal conductivity and volumetric specific heat with temperature were considered in the thermal model. [24, 30]

Models including magnetohydrodynamic effects, thermo-solutal buoyancy effects, and Marangoni or surface tension effects offer new insight into the formation of the melt pool in welding. [31, 32] By artificially increasing the thermal conductivity above the melting temperature, one can achieve a reasonable approximation for the effects of convective mixing without much increase in complexity or solution times. To compensate for weld pool convective heat transfer of AL-6XN material, a high conductivity value, 160 W/m-K, was used in the weld pool. [33] Heat of fusion is the energy required to change a solid at its melting temperature to liquid at the same temperature. The release or absorption of latent heat of fusion was simulated by an artificial increase in the value of specific heat over the melting temperature range. The latent heat was taken as $2.1 \times 10^9$ J/m$^3$; the melting range was 1320-1400°C.

Thermal property data for AL6XN stainless steel used in the analysis are given in Figure 2, 3.
Figure 2 – Adjusted Thermal Conductivity of AL6XN

Figure 3 – Adjusted Specific Heat of AL6XN
2.1.2 Mechanical properties

Temperature dependent values of properties such as modulus of elasticity, Poisson's ratio, and yield strength, coefficient of thermal expansion (or thermal strain) were provided to the model. The mechanical properties used in the study are given in Figure 4 to 7.

When a material is deformed repeatedly, its mechanical properties may continue to change. This behavior can be accounted for by using adapted strain hardening models and isotropic strain hardening model was used in the analysis. Figure 7 obtained from SYSWELD database approximately represents the behavior of AL6XN stainless steel.

![Modulus of elasticity of AL6XN](image)

*Figure 4 – Modulus of elasticity of AL6XN*
Figure 5 – Yield Strength of AL6XN

Figure 6 – Thermal Strain of AL6XN
In a coupled shell-solid analysis using SYSWELD, there are many things that are different from standard 3D analysis. Mechanical properties of shell elements should give the thermal expansion coefficient instead of thermal strain in volume elements. For the plastic part of the stress strain curve, volume elements give the values of plastic strain and the difference between the stress at that strain and the yield stress was provided to the model, while shell elements give the values of plastic strain and stress at that strain. [34]

2.2 Model generation

A T-beam made of two plates of thickness 9 mm with a fillet weld between them was taken up for the investigation. The width of the non-butting member of the T-beam was 160mm and butting member was 122.5 mm.
Of the two fillet welds of T-beam on either side of the butting member, first a one sided fillet weld was considered. The fillet width was 8 mm. The length of the T-beam was taken to be 128 mm. The size of the T-beam was chosen to be small to reduce the computational time required for the analysis. The sketch of the T-beam taken for investigation is shown in Figure 8.

For full volume model of T-beam, 8 noded linear hexahedron volumetric elements (H8 elements) in SYSWELD were used to construct the geometry. 4 noded linear quadrilateral elements (Q4 elements) were used on the surface to apply the convective and radiative boundary conditions. The fillet weld bead that arises due to deposition of metal was divided into finer meshes using P6 elements. The exposed surface of the weld bead was divided using Q4 elements to apply the boundary conditions. The total number of nodes in the model was 8606, and the number of elements in the mesh was 11704.

For shell-volume coupled model, 4 noded linear quadrilateral elements (Q4 elements) were used to apply shell elements with the thickness of 9 mm. A convection coefficient is defined for the two sides of the shell elements. In mechanical analysis, the number of degrees of freedom for shell elements (ux, uy, uz, rx, ry, rz) and for three-dimensional elements (ux, uy, uz) is different. In order to obtain compatibility between the movement of the plates modeled with 3D and 2D elements, special elements called...
“compatibility” or “transition elements” are added in the connection section area. A relation is therefore created on the nodes of the 3D section to ensure that global displacement remains perpendicular to the mean plane of the shell element to which it is connected. The three-dimensional mesh at the connection with the shell elements has to be composed of 2 layers corresponding to the half-thickness of the shell, to correctly apply relations of compatibility. [35] The total number of nodes in the model was 8246, and the number of elements in the mesh was 11158.

The meshed views of the solid T-beam and the mixed element model are shown in Figure 9, 10 respectively.

![Figure 9 - mesh of full-volume model](image)
2.3 Thermal Model (thermal source modeling)

The heat transfer model best suited for arc-welding applications is the Goldak, or double ellipsoid, source. (See Figure 11) Goldak's source corrects the Rosenthal model's point source assumption by distributing power through a volume of specified size and shape.

This size and shape is adjusted through a number of Gaussian parameters, each independently controlling the width, forward length, rearward length, and depth of
heating. By manipulating these parameters, the heat source can be changed to reflect a very wide variety of welding conditions.

The formulation of the Goldak model is shown in Equations 2.1 and 2.2. By using different parameters for the front and rear ellipsoids, it is possible to specify an asymmetric distribution of power. Here, \( f_f \) and \( f_r \) are the fractions of power sent to the front and rear ellipsoids, respectively. Parameters \( a, b, c_f \) and \( c_r \) determine the shapes of the ellipsoids as shown in Figure 11. Finally, \( Q \) is the total power that enters the part, given by \( Q = \eta VI \), where \( \eta \) is the arc efficiency, \( V \) is the voltage, and \( I \) is the current.

\[
q(x, y, z, t) = \frac{6\sqrt{3} f_f Q}{abc_f \pi \sqrt{\pi}} e^{-3x^2/a^2} e^{-3y^2/b^2} e^{-3[z + v(t-t_0)]^2/c_f^2} \tag{2.1}
\]

\[
q(x, y, z, t) = \frac{6\sqrt{3} f_r Q}{abc_r \pi \sqrt{\pi}} e^{-3x^2/a^2} e^{-3y^2/b^2} e^{-3[z + v(t-t_0)]^2/c_r^2} \tag{2.2}
\]
Goldak's heat source model was used in this study. The code has a provision to apply this heat source model to the welding problems. In the model, the heat source was made to traverse along the length of T-beam at a 45° inclination to deposit the fillet weld.

Heat transfer to the ambient takes place by convection and radiation. Both convection and radiation heat transfer were considered in the model. The convective heat transfer coefficient of 15 W/m²·K was used. The emissivity value used in the analysis was 0.5. To simulate material deposition during welding, the 'activation/deactivation' function of SYSWELD was employed. In the thermal analysis, the elements were activated a little in front of the heat source to avoid numerical problems. The thermal analysis was carried out up to 5010s, till the T-beam cooled down to room temperature, after welding.

The welding parameters are as follows: Voltage: 35 V, Current: 250 A, Arc efficiency: 75 %

The case considered was continuous forward welding over the 128 mm full length of T-beam. The weld speed was 5 mm/s and the duration of weld deposition was 25.6s. Cooling phase started at 25.6s and the analysis was completed after 5010s.

2.4 Mechanical Model

The transient temperature distribution file obtained from thermal analysis was given as input to the mechanical model. The boundary conditions or restraints applied to the T-beam during mechanical analysis are shown in Figure 12.
The points A, B and C of Figure 12 were used to estimate the progress in proportion to displacement with weldment. The displacement between point A, a start point of shell elements in the coupled model, and weldment is 29mm, point B is 73.5mm and point C is 118mm. The direction of arrows means the direction fixed.

As in thermal analysis, elements representing filler metal were activated in mechanical analysis whenever required. In the mechanical analysis, the elements were activated slightly behind the centre of the heat source. The stress analysis was carried out up to 5010s as in the case of thermal analysis. At 5000s, the restraints of the T-beam were reset to an unrestrained condition, resulting in “spring back” and redistribution of stresses and deflection in the T-beam.
2.5 Thermal Results

Thermal results for both models are shown in Figure 13 to Figure 18. The temperature and stress contours were obtained using the post processing module of SYSWELD.
Figure 14 - Volume Model at 20 seconds

Figure 15 - Volume Model at 250 seconds
Figure 16- Coupled Model at 10 seconds

Figure 17- Coupled Model at 20 seconds
The results of temperature at point A, B and C of Figure 12 were compared in order to estimate the progress in proportion to displacement with weldment. (Figures 19, 20, 21)

In Figure 19, the peak point of temperature in 3D volume model is 133.5°C at 130 seconds and the peak point in coupled model is 130.3°C at 140 seconds. The peak temperature of both models at A point is almost same, while the temperature of coupled model decreases slowly compare to the temperature of volume model. This is due to the difference of heat transfer between shell and volume elements.

In Figure 20, the peak temperature in volume model is 51.0°C at 516 seconds and the peak point in coupled model is 56.3°C at 652 seconds. The rate of difference is 9.4%
and the delayed time of peak point is 136 seconds. The temperature at point B of coupled model decreases slowly compared to the temperature of volume model at point A.

The tendency of temperature at point C is similar to that at point B (Figure 21). Although the absolute values of temperature are small, the difference of temperature between both models increases a little and the delayed time of peak point is also longer than point B. The difference of temperature between volume model and coupled model at peak point is 9.2°C and that is large as compared to 5.2°C of point B.

![Figure 19- Temperature at Point A](image-url)
Figure 20- Temperature at Point B

Figure 21- Temperature at Point C
The reasons for the differences in temperature between the volume model and coupled shell/solid model are

i) the difference in heat transfer between the shell and volume elements at the connecting line (When shell elements are connected with volume element, the middle node of both volume elements is connected with the node of shell element.)

ii) the difference in heat loss through heat convection and radiation (In the volume model, the heat loss can occur through edges as well as both of the front and rear parts of plate, while the heat loss in the shell model occurs at only through the front and rear faces of the plate.)

iii) the difference element in the meshes of shell and volume elements (size, calculating method, averaging method of result, ... etc.).

2.6 Mechanical Result

The thermal results in the previous section were used to create the mechanical models. The results for distortion are of more interest than the results of stresses at shell elements, because the purpose of this research was to develop a comprehensive 3D finite element model in order to compute global deformation of assembled ship structures. The resulting displacements for both models are shown in Figures 22 to Figure 25. The contours in these figures represent the displacements in the x-direction.
Figure 22- Volume Model at 16 seconds (deformed shape x 10)

Figure 23- Volume Model at 5010 seconds (deformed shape x 10)
Figure 24- Coupled Model at 16 seconds (deformed shape x 10)

Figure 25- Coupled Model at 5010 seconds (deformed shape x 10)
The results of distortion in the x-direction at point A, B, C were compared in order to estimate transient distortion relative to displacement from weld fusion zone. (See Figures 26, 27, 28, 29, 30, 31) The values of distortion of both volume model and coupled model at each point per time step show a good agreement. The difference of deflection at point A after being cooled is 0.06 mm (the distortion of volume model is 1.16mm and that of coupled model is 1.10mm), that at point B is 0.05 mm (the distortion of volume model is 2.60mm and that of coupled model is 2.55mm) and that at point C is 0.04 mm (the distortion of volume model is 4.04mm and that of coupled model is 4.00mm). There is no difference of distortion relative to displacement from weld fusion zone.

Figure 26- Deflection ($u_x$) of Point A (0 s to 5010 s)
Figure 27 - Deflection (ux) of Point A (0 s to 70 s)

Figure 28 - Deflection (ux) of Point B (0 s to 5010 s)
Figure 29- Deflection ($u_x$) of Point B (0 s to 70 s)

Figure 30- Deflection ($u_x$) of Point C (0 s to 5010 s)
In the mechanical models of welding presented, longitudinal stresses (y-direction) show the largest stress contours compared with those of other directions. The results of longitudinal stresses for both models are compared and are shown in Figure 32 to Figure 35. At the case of the coupled model, the stress contours of shell and solid elements are displayed separately because the stress results of shell elements can not be showed with the solid elements in SYSWELD. In addition, the values of stress in the connected nodes are the added values of both results from solid and shell elements.
Figure 32- Volume Model at 16 seconds

Figure 33- Volume Model at 5010 seconds
Figure 34- Coupled Model at 16 seconds
Figure 35- Coupled Model at 5010 seconds
The stresses in the volume elements obtained from the coupled model are almost same with those obtained from the volume model entirely composed of solid volume elements. The results of longitudinal stresses at points A, B, C were compared in order to estimate the stress evolution in relation to displacement from the welded zone. (See Figures 36, 37, 38, 39, 40, 41) The absolute values of longitudinal stresses at shell elements are relatively small, when compared to equivalent stresses in the volume elements of welded part. However, the values of stress for both the volume model and coupled shell model at each point per time step show a little difference though the tendencies of these stresses are similar.

---

**Figure 36- Longitudinal stress of Point A (0 s to 5010 s)**
Figure 37- Longitudinal stress of Point A (0 s to 100 s)

Figure 38- Longitudinal stress of Point B (0 s to 5010 s)
Figure 39- Longitudinal stress of Point B (0 s to 100 s)

Figure 40- Longitudinal stress of Point C (0 s to 5010 s)
The difference of residual stress at point A after cooling is 36.6Mpa (the residual stress of volume model is -145.2Mpa and that of coupled model is -108.6Mpa), that at point B is 3.0Mpa (the residual stress of volume model is -1.2Mpa and that of coupled model is 1.8Mpa) and that at point C is 7Mpa (the residual stress of volume model is 18.9Mpa and that of coupled model is 25.9Mpa). Although the values of the nodes in the connecting region show the difference of 36.6Mpa, this difference is small, when compared to the stresses in the volume elements of welded part. (See Figure 41)

In Figure 41, the longitudinal stresses after cooling according to displacement from the weld fusion zone are presented. The residual stresses of both models show the similar results.
Figure 41- Longitudinal stresses in relation to displacement from welded zone

The most different value within the results of residual stresses shows in the connecting region between the shell and solid elements.

2.7 Conclusion about Comparison of both models

The results of the coupled model with shell and volume elements and the 3D model with only volume elements applied to the practical welding problem of T-beam were compared in this chapter.

The Followings are the major observations of the present comparison:
i) The difference in temperature between both models slightly increases and the delayed time to reach the peak temperature is also taken longer in proportion to displacement from weld fusion zone. The temperature for shell elements cools down slowly. This difference, however, can be ignored because the absolute values of temperature at shell elements are very small.

ii) The values of deformation at coupled model are almost same with those at volume model. Therefore, this coupled model of shell and volume elements can be considered as an effective method to estimate the global distortion of a large structure such as ship construction.

iii) The values of residual stress for both the volume model and coupled shell model show a little difference though the tendencies of these stresses are similar. Especially, the temperature at connecting region should be considered carefully. This difference also can be ignored due to the fact that the absolute values of stresses at shell elements are relatively small.
Chapter 3 - Comparison of the coupled elements models with different boundary conditions

3.1 Properties of material

AL6XN stainless steel is used in this chapter and the same thermal and mechanical properties of chapter 2 are applied.

3.2 Model generation

The vertical height of the butting member of the T-beam is 78mm. All dimensions of T-beam are same with what was used in chapter 2 except the vertical height of the butting member. Both sides fillet welding was considered.

The total number of nodes in the model was 8201, and the number of elements in the mesh was 11116. The meshed views of the T-beam are shown in Figure 42.

![Figure 42 - mesh of model](image-url)
3.3 Thermal source modeling

The double ellipsoid source that is the same source with chapter 2 was applied and all of the parameters in this heat source were same but both sides fillet weld.

After the first weld deposition time (25.6s), a cooling time of 60s was allowed before the second fillet weld deposition was started on the other side of the vertical plate. Thus, the two welds art not deposited simultaneously. The thermal analysis was carried out up to 5010s.

3.4 Mechanical Model

The boundary conditions applied to the T-beam during mechanical analysis are shown in Figures 43, 44, 45 separately. The BC I is the same BC in previous chapter, and the z-direction displacement is fixed. In the BC II, the x-direction displacement of both edges in butting plate additionally. The BC III constrained the two end points of the middle line at non-butting member in order to make the displacement of four edges in this plate free. The direction of arrows means the direction fixed.

The points A, B and C in Figure 44 and lines DD', EE', FF' in Figure 45 were used to compare the results for each boundary condition. The displacement between point A and weld fusion zone is 11.5mm, point B is 29mm, and point C is 73.5mm and the lines are in the middle part of T-beam.
Figure 43 – Boundary Condition (BC) I of Model

Figure 44 – BC II of Model
At 5000s, the restraints of the T-beam were removed using the 'release' option of the SYSWELD code.

3.5 Thermal Results

Thermal results for the three models are identical, and are shown in Figure 46, Figure 47 and Figure 48.
Figure 46 – Contour of temperature at 16 seconds

Figure 47 – Contour of temperature at 95 seconds
The results of temperature at point A, B and C of Figure 43 per time step were displayed in order to estimate the temperature evolution in relation to displacement with welded zone. (See Figure 49)

In Figure 48, the temperatures of point A, the point closest to the weld fusion zone, shows the highest values and two peak points influenced by both sides welding having started at different times. The temperatures of points B and C hardly show the influence of both sides being welded with a time delay between depositions of the separate fillet welds, but the values of temperature are higher than those of one sided weld in previous chapter.
3.6 Mechanical Results

The thermal results were used to create mechanical models with different constraints. The results of displacement for the three models are shown in Figure 50 to Figure 56 and the contours of result represent the normal displacement, the magnitude of the displacement vector.
Figure 50- Normal displacement of BC I at 16 seconds (deformed shape x 10)

Figure 51- Normal displacement of BC I at 101 seconds
Figure 52- Normal displacement of BC I at 5010 seconds

Figure 53- Normal displacement of BC II at 99.5 seconds
Figure 54- Normal displacement of BC II at 5010 seconds

Figure 55- Normal displacement of BC III at 99.5 seconds
In order to compare the distortion of T-beam, the angular change and longitudinal bending distortion was tabulated.

The results of displacement after being cooled are shown in Table 3-1. The angular change $\alpha$, $\beta$, $\gamma$ and longitudinal bending distortion $d$ in Figure 57 are the degrees and displacement of line DD', EE', FF' in Figure 44.
The final results of distortions in the cross-section after removing the restraints on the T-beam are slightly different depending on the boundary conditions, while the
longitudinal bending distortions have almost same values. Especially, the final result of the model with BC II is different as compared to BC I and III. (See Figures 58, 59, 60, 61, 62) The interesting part of the model with BC II is that the displacement shows a large change of values at the instant that restraint is released. This is because the BC I, III basically allowed the free-movement of vertical plate while the constraint of BC II was fixing the distortion of model during welding and cooling. When the constraint was removed, the suppressed distortion occurred abruptly.

After the first welding and 60 seconds cooling, the angle changes of BC I, II are very different with those of BC III due to the fact that both of BC I and II constraint the four corners of non-butting member and BC III does the two end points of the middle line at non-butting member. The angle changes of three models, however, are converged to similar values when the other side welding is finished. Although the difference of distortion reduced after welding of both side, the results were varied greatly by boundary condition.
Figure 58 - $\alpha$ Angle change (0 s to 5010 s)

Figure 59 - $\alpha$ Angle change (0 s to 200 s)
Figure 60 - β Angle change (0 s to 5010 s)

Figure 61 - β Angle change (0 s to 200 s)
Figure 62- $\gamma$ Angle change (0 s to 5010 s)

Figure 63- $\gamma$ Angle change (0 s to 200 s)
The results of longitudinal stresses that have the largest stress values at three models are shown in Figure 64 to Figure 66.

Figure 64- Longitudinal stresses of BC I at 5010 seconds
CONTOURS
Sigma 22
Time 5010
Comput.Ref G1

Min = -231.5
Max = 715.8

-300
-233.333
-166.667
-100
-33.3333
33.3333
100
166.667
233.333
300

Figure 65- Longitudinal stresses of BC II at 5010 seconds
Figure 66 - Longitudinal stresses of BC III at 5010 seconds

The results of longitudinal stresses at point A, B and C of Figure 43 per time step were displayed at Figures 67, 68, 69.
Figure 67- Longitudinal stress of Point A (0 s to 5010 s)

Figure 68- Longitudinal stress of Point B (0 s to 5010 s)
The results of longitudinal stresses with different constraints are similar, though the results of BC II show a bit of difference due to the restraints of the vertical plate. Therefore, it could be analogized that the most influenced factor of stresses at welding simulation is the contours of temperatures per time steps and the boundary conditions applied in this chapter did not affect so much.
Chapter 4 - Welding simulation of a practical problem (the hull of ship)

4.1 Model generation

A single cell section of a double hull welded box beam was simulated. This cell with a cross section shape of a cellular box is fabricated using AL-6XN steel plates. All dimensions of the cell were based on a prototype double hull section. The dimensions are shown in Figure 70.

Figure 70- Cross section of box cell
The length of box cell used in the finite element models is 3 feet 2 and 3/4 inches. It is believed that this length is sufficient to obtain satisfactory results that could estimate a states of distortion for longer beams and still allow reasonable computation times and storage.

One-quarter of this cell was meshed due to the assumption of symmetry conditions. This assumption implies that the welding proceeds on all four edges simultaneously. The total number of nodes in the model was 10231, and the number of elements in the mesh was 15868. The meshed views of this model are shown in Figure 69.

4.2 Thermal and Mechanical modeling

The applied heat source is a double ellipsoid source and both sides on the vertical plate arc fillet welded. In the model, the heat source was made to traverse along the length of T-beam at a 45° inclination to deposit the fillet weld metal. Some of the welding parameters are given in the Table 4-1.

<table>
<thead>
<tr>
<th>Convective heat transfer coefficient</th>
<th>15 W/m²-K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Emissivity</td>
<td>0.5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Welding parameters</th>
<th>Voltage</th>
<th>35 V</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Current</td>
<td>250 A</td>
</tr>
<tr>
<td></td>
<td>Arc efficiency</td>
<td>75 %</td>
</tr>
</tbody>
</table>

Table 4-1 – Distortion after removing the restraints

After the first weld deposition to 197s, the model was cooled for 5000s and the second weld deposition of the other side was started. At 5000s, the restraints of the T-
beam were reset to an unrestrained condition. The thermal analysis was carried out up to 5010s. The boundary conditions applied to this cell box during the mechanical analysis are shown in Figure 71.

Figure 71 – Boundary Condition of Cell box (1/4 Model)

4.3 Thermal Results

Thermal results for this model are shown in Figure 72, Figure 73 and Figure 74.
Figure 72 – Contour of temperature at 50.1 seconds

Figure 73 – Contour of temperature at 150 seconds
4.4 Mechanical Results

The results of displacement for the symmetric cell are shown in Figure 75 to Figure 77 and the contours of result represent the magnitude of the displacement vector.
Figure 75 – Normal displacement at 30 seconds

Figure 76 – Normal displacement at 102.6 seconds
The results of longitudinal stresses that have the largest stress values for this model are shown in Figure 78 to Figure 80.

In Figure 80, high tensile residual stresses are produced in areas near the weld and the longitudinal stresses in areas away from the weld are compressive.
Figure 78 – Longitudinal stress at 30 seconds
Figure 79 – Longitudinal stress at 102.6 seconds
In Figure 81, the residual stresses after cooling according to displacement from the weld fusion zone are presented.
The boundary condition that was used in this chapter may not precisely replicate the state of stress in the “actual” problem due to the fact of that one-quarter symmetry of a cell box is assumed. However, it is expected that this will provide a reasonable estimate of the final distortion. In addition, the model presented here are too small to adequately simulate the complete behavior of an entire hull structure. These results, however, show that a practical problem in welded ship construction can be simulated by using the coupled model of shell and volume elements.
Chapter 5 - Conclusions and Future Work

5.1 Future Work

The greatest limitations in modeling a large structure are computation time and storage requirements. The advantage of plate and shell elements is that the elements can be fairly coarse and still deliver reliable results. Therefore, to use solid elements in regions of high thermal gradients and plates and shells elsewhere offers a significant reduction in solution times and storage requirements without a painful loss of accuracy.

In the simulations considered here, only simple models such as T-beam and single cellular box beam of the double hull were considered. Eventually, an entire hull ship, with more complicated geometry, should be simulated using this technique. (See Figure 82)

Figure 82 - Design of double hull in ship

77
When a three-dimensional mesh is generated to accurately simulate welding, the entire region that will be welded must be very refined. Therefore, any model that has a large welding region presents enormous computational problems in terms of CPU time and storage.

5.2 Conclusions

The temperature, distortion and residual stresses of the coupled model with shell and volume elements and the 3D model with only volume elements in a welded T-joint were investigated in this study. The all results for a coupled shell/solid volume model showed the same tendencies and similar values obtained from an equivalent solid volume model. Therefore, a coupled model can be considered as an effective method to estimate the global distortion of a large structure, such as ship double hull structures.

Finally, models of both sides welded T-joints with different constraints were simulated and compared. According to the boundary conditions, the results of distortions varied greatly and the stress concentration occurred at the place that was fixed. It is important to recognize that boundary condition constraints play a critical role in determining distortion for welded structures.
Appendix

Listed here are the main input files for a single cell section of a double hull welded box beam simulation discussed the chapter 4. The second mechanical input file for solving the released constraints is not presented because this file is almost same with the first mechanical file except the constraints part.

HEAT.DAT (Thermal model data)

NAME
SEARCH DATA 101

COMPATIBILITY PENALTY 200*6
MIXING SOLID SHELL
SOLID ELEMENT GROUP $PART$
SHELL ELEMENT GROUP $SHELL$
RETURN

DEFINITION
T-JOINT CONTINUOUS WELDING
OPTION THERMAL METALLURGY SPATIAL
RESTART GEOMETRY
MATERIAL PROPERTIES
ELEMENTS GROUP $PART$ / C=-10001 KX=-10002 KY=-10002 --
KZ=-10002 RHO=1 MATE=1
ELEMENTS GROUP $BEAD1$ / C=-10001 KX=-10002 KY=-10002 --
KZ=-10002 RHO=1 STATE=-5 MATE=1
ELEMENTS GROUP $SHELL$ / C=-10001 KX=-10002 KY=-10002 --
KZ=-10002 RHO=1 MATE=1 H=7.9375
CONSTRAINTS
ELEMENTS GROUP $SKINPART$ / KT=1 VARIABLE=10
ELEMENTS GROUP $SHELLS$ / KT=1 LOWER UPPER VARIABLE=10
LOAD
1 WELDING/ NOTHING
ELEMENTS GROUP $PART$ / QR=1 VARIABLE=-100
ELEMENTS GROUP $BEAD1$ / QR=1 VARIABLE=-100
ELEMENTS GROUP $SHELLS$ / TT=20 LOWER UPPER
ELEMENTS GROUP $SKINPART$ / TT=20
TABLE
10001 / 1 20 0.004030 500 0.004836 1200 0.005239 1293 0.00529
1300 0.0054 1310 0.0060 1325 0.02525 1330 0.02600 1333 0.02625
1388 0.02625 1392 0.02600 1395 0.02525 1400 0.0066 1405 0.0058
1420 0.0053599 1600 0.0053599
10002 / 1 20 0.0137 100 0.0137 500 0.0250 1283 0.0382 1306 0.042
1327 0.05048 1397 0.152 1412 0.156 1445 0.1600 1570 0.1600
1700 0.1600

79
10 / FORTRAN
function f(t)
c radiation losses : f = sig * e * (t + to) (t**2 + to**2)
e = 0.5
sig = 5.67*8
to = 20.
t0 = 20. + 273.15
t1 = t + 273.15
a = t1 * t1
b = to * to
c = a + b
d = t1 + to
d = d * c
d = d * e
d = d * sig
c convective losses = 15 W/m2
f = d + 15.
g=1.0*6
f=f*g
return
END

; Heat Source Definition
100 / FORTRAN
FUNCTION F(X)
DIMENSION X(5)
xa = X[1];
ya = X[2];
z a = X[3];
time = X[4];
c initial position of heat source in the new frame
xc = 7.14375;
yc = 1.0;
z c = 11.1125;
c Translation
xa = xa - xc
ya = ya - yc
za = za - zc
c rotation matrix
a1 = 0.7071;
a2 = 0.0;
a3 = -0.7071;
b1 = 0.0;
b2 = 1.0;
b3 = 0.0;
c1 = 0.7071;
c2 = 0.0;
c3 = 0.7071;
c rotation
aa = a1 * xa
bb = a2 * ya
cc = a3 * za
xx = aa bb cc + +
aa = b1 * xa
bb = b2 * ya
cc = b3 * za
yy = aa bb cc + +
aa = c1 * xa
bb = c2 * ya
cc = c3 * za
zz = aa bb cc + +
c
Q = uu*I
uu = 35.0;
ii = 250.0;
q = uu*ii;
q = q*6.0;
q = q*1.7320508; // q = q*sqrt(3)
q = q/3.1415927;
q = q/1.7724539; // q/sqrt(pi)
c
EFFICIENCY
e = 0.75;
q = q*e;
c
WELDING VELOCITY mm/s
wv = -5.0
c
PARAMETERS OF WELDING POOL
a = 4.5; // Half width of weld pool (from T-Joint welded piece)
b = 4.9; // Depth of weld pool (estimation using AWS Doc.A3.0-94)
Yf = a * 0.75; // Length of weld pool in front of center (Goldak’s paper)
Yr = a * 1.5; // Length of weld pool behind center (Goldak’s paper)
c
Fitting power base to mesh
Qc = 1.0; // Energy for a density = 1.
Qe = 1./Qc; // density of energy
c
Proportion of heat in front and rear (Non symmetric Gaussian distribution)
c
Note: Qf + Qr = 2
Qf = 0.6; // Fraction of energy in front of HS (Goldak’s paper)
Qr = 1.4; // Fraction of energy in the rear of HS (Goldak’s paper)
c
POSITION OF HEAT SOURCE CENTER

tim1 = time-tim1
center = wv*tim2;
center = wv*time;
if (yy .GT. center) Qg = Qr;
if (yy .LE. center) Qg = Qf;
if (yy .LE. center) cc = Yf;
if (yy .GT. center) cc = Yr;
Qg = Qg*Qe;
Qg = Qg/a;
Qg = Qg/b;
Qg = Qg/cc;
c
CALCULATION HEAT SOURCE BY GOLDAK'S FORMULA
rx = xx;
rx = rx*rx;
rx = -rx;
rx = rx*3.0;
s = a*a;
rx = rx/s;
\begin{verbatim}
rx = \exp(rx);
ry = center - yy;
ry = ry * ry;
ry = -ry;
ry = ry * 3.0;
e = cc * cc;
ry = ry / e;
ry = \exp(ry);
rz = zz;
rz = rz * rz;
rz = -rz;
rz = rz * 3.0;
dc = b * b;
rz = rz / dc;
rz = \exp(rz);
coef = ry * rx;
coef = coef * rz;
f = coef * Qg;
f = f * q;
RETURN
END

5 / FORTRAN
FUNCTION F(X)
DIMENSION X(4)
XX X(1);
YY X(2);
ZZ X(3);
TT X(4);
VY = -5.0;
C OUTPUT PARAMETERS
C F=1 ELEMENT ACTIVATION
C F=-1 ELEMENT DEACTIVATION
C F=0 NO EFFECT
F=1
VYT = VY * TT
VYT = VYT - 0.01
IF (YY.LT.VYT) f = -1
RETURN
END

RETURN
SAVE DATA 102

SEARCH DATA 102
RENUMBER ITERATION 50
RETURN

SAVE DATA 102

SEARCH DATA 102
TRANSIENT NON-LINEAR EXTRACT 0
BEHAVIOUR METALLURGY 2 FILE META.DAT
ALGORITHM BFGS IMPLICIT 1 ITERATION 250
PRECISION ABSOLUTE NORM 0 FORCE 1*10 DISPLACEMENT 1
\end{verbatim}
METHOD ITERATIVE NONSYMMETRICAL

INITIAL CONDITIONS

NODES / TT 20
ELEMENTS GROUP $PART$ / P 1 0
ELEMENTS GROUP $SHELLS$ / P 1 0
ELEMENTS GROUP $BEAD1$ / P 1 0 IS -1
ELEMENTS GROUP $BEAD2$ / P 1 0 IS -1

TIME INITIAL 0.0
  0.25 STEP 0.125 / STORE 1

SEARCH DATA 102
ASSIGN 19 TRAN102.TIT
TRANSIENT NON-LINEAR EXTRACT 0
BEHAVIOUR METALLURGY 2 FILE META.DAT
ALGORITHM BFGS IMPLICIT 1 ITERATION 200
PRECISION ABSOLUTE NORM 0 FORCE 1*10 DISPLACEMENT 1
METHOD ITERATIVE NONSYMMETRICAL
INITIAL CONDITION RESTART CARD LAST
TIME INITIAL RESTART
  1 STEP 0.2 / STORE 1
  6 STEP 0.25 / STORE 1
  201 STEP 0.3 / STORE 1
  210 STEP 1 / STORE 1
  260 STEP 2 / STORE 1
  500 STEP 5 / STORE 1
  800 STEP 20 / STORE 1
  5000 STEP 100 / STORE 1
  5004 STEP 0.5 / STORE 1
  5010 STEP 1 / STORE 1
RETURN

SAVE DATA 102
DEASSIGN 19
MECH.DAT (Mechanical model data)

SEARCH DATA 102
DEFINITION
  T - WELD JOINT
OPTION SHELL SPATIAL MULTI THERMOELASTICITY
RESTART GEOMETRY
MATERIAL PROPERTIES
  ELEMENTS GROUP $PART$ / E=-10001 YIELD=-10004 LX=-10003 LY=-10003 --
  LZ=-10003 MODEL=3 NU=-10002 SLOPE=-10008 PHAS=2 TF=1400
  ELEMENTS GROUP $BEAD1$ / STATE=-4 E=-10001 YIELD=-10004 LX=-10003 --
  LY=-10003 LZ=-10003 MODEL=3 NU=-10002 SLOPE=-10008 PHAS=2 TF=1400
  ELEMENTS GROUP $BEAD2$ / STATE=-6 E=-10001 YIELD=-10004 LX=-10003 --
  LY=-10003 LZ=-10003 MODEL=3 NU=-10002 SLOPE=-10008 PHAS=2 TF=1400

  ELEMENTS GROUP $SHELL$ / H=7.9375 INTE=902 TYPE=4 E=-10001 YIELD=-
  10004 --
  LX=-10021 LY=-10021 LZ=-10021 MODEL=3 NU=-10002 SLOPE=-10023 PHAS=2
  TF=1400
  ELEMENTS GROUP $SKINPART$ / TYPE=5
  ELEMENTS GROUP $SKINBEAD1$ / TYPE=5
  ELEMENTS GROUP $SKINBEAD2$ / TYPE=5
  ELEMENTS GROUP $ELEM_TRAN_SH$ / E=200000000 SHAPE=1 TYPE=9

CONSTRAINTS
  NODES 27101 / UX UY
  NODES 27100 / UX
  NODES 27102 / UX
  NODES 27103 / UX
  NODES 27104 / UX
  NODES 27105 / UX
  NODES 27106 / UX
  NODES 27107 / UX
  NODES 27108 / UX
  NODES 27109 / UX
  NODES 27110 / UX
  NODES 27111 / UX
  NODES 27112 / UX

  NODES 27205 / UZ
  NODES 27206 / UZ
  NODES 27207 / UZ
  NODES 27208 / UZ
  NODES 27209 / UZ
  NODES 27210 / UZ
  NODES 27211 / UZ
  NODES 27212 / UZ
  NODES 27213 / UZ
  NODES 27214 / UZ
  NODES 27215 / UZ
  NODES 27216 / UZ
  NODES 27217 / UZ

LOAD
  1 WELDING/ NOTHING
### TABLE

| 10001 | 1 24 195000 93 189000 204 180000 316 171000 427 161000 -- | 538 152000 982 90000 1093 72000 1200 45000 1260 41000 1300 -- | 20000 1320 10000 1350 50 |

| 10002 | 1 0 0.29 900 0.3 |

| 10003 | -10006 -10006 |

| 10006 | 1 20 0.000000 100 0.001224 300 0.004396 400 0.006080 500 -- |

| 10021 | -10022 -10022 |

| 10022 | 1 20 0.000153 100 0.000153 200 0.000153 300 0.000157 400 |

| 10004 | -10007 -10007 |

| 10007 | 1 21 365 93 325 149 290 204 270 260 255 316 235 371 230 427 |

| 230 -- | 482 220 538 215 982 70 1093 39 1200 31 1260 28 1300 20 1320 10 1350 1 |

| 10008 | -10009 -10009 |

| 10009 | 7 20 10010 500 10011 1450 10012 |

| 10010 | 1 0 0 0.01 23 0.02 60 0.04 102 0.06 136 0.08 168 0.10 198 0.15 270 |

| 10011 | 1 0 0 0.01 23 0.02 42 0.04 70 0.06 92 0.08 109 0.10 126 0.15 156 |

| 10012 | 1 0 0 0.01 0.5 0.02 0.5 0.04 0.5 0.06 0.5 0.08 0.5 0.10 0.5 0.15 0.5 |

| 10023 | -10024 -10024 |

| 10024 | 7 20 10025 500 10026 |

| 10025 | 1 0 365 0.01 388 0.02 425 0.04 467 0.06 501 0.08 533 0.10 563 0.15 635 |

| 10026 | 1 0 218 0.01 241 0.02 260 0.04 288 0.06 310 0.08 327 0.10 344 0.15 374 |

| 4 / FORTRAN |

```fortran
FUNCTION F(X)
DIMENSION X(4)
XX = X(1);
YY = X(2);
ZZ = X(3);
TT = X(4);
VY = -5.0;
C OUTPUT PARAMETERS
C F=1 ELEMENT ACTIVATION
C F=-1 ELEMENT DEACTIVATION
C F=0 NO EFFECT
F=1
VYT = VY * TT
VYT = VYT + 0.1
IF (YY .LT. VYT) f=-1
```

85
SAVE DATA 106

SEARCH DATA 102
SEARCH TRAN 102
TEMPERATURE METALLURGY TRANSIENT SHELL CARD 0 TO 800 STEP 1

SEARCH DATA 106
TRANSIENT NON-LINEAR STATIC EXTRACT 0
BEHAVIOUR PLASTIC METALLURGY 2
ALGORITHM BFGS IMPLICIT 1 ITERATION 200
PRECISION ABSOLUTE NORM 0 FORCE 1 DISPLACEMENT 1*-3
METHOD NONSYMMETRICAL TEST 1 ITERATIVE
INITIAL CONDITION
   ELEMENTS GROUP $BEAD1$ / IS -1
   ELEMENTS GROUP $BEAD2$ / IS -1
TIME INITIAL 0
   0.25 STEP 0.125 / STORE 2
RETURN
SAVE DATA TRAN 106

SEARCH DATA 106
ASSIGN 19 TRAN106.TIT
TRANSIENT NON-LINEAR STATIC EXTRACT 0
BEHAVIOUR PLASTIC METALLURGY 2
ALGORITHM BFGS IMPLICIT 1 ITERATION 300
PRECISION ABSOLUTE NORM 0 FORCE 1 DISPLACEMENT 1*-3
METHOD NONSYMMETRICAL TEST 1 ITERATIVE
INITIAL CONDITION RESTART CARD LAST
TIME INITIAL RESTART
   1 STEP 0.2 / STORE 3
   6 STEP 0.25 / STORE 4
   210 STEP 1 / STORE 10
   260 STEP 2 / STORE 10
   500 STEP 5 / STORE 6
   800 STEP 20 / STORE 3
   5000 STEP 100 / STORE 3
RETURN
SAVE DATA 106
DEASSIGN 19
References

VITA

Dongjin Kim was born in Pusan, South Korea on 26th of March 1971. He is the second son of Chunchu Kim. He graduated from high school in 1989 and started his undergraduate education in mechanical engineering at Pusan National University. He completed his undergraduate studies in 1996 and started to work for the company, POSCO (Pohang Iron and Steel Co., Ltd.). After 6 years work for his company, he came to Lehigh University in 2002. He expects to get his Master’s degree in December 2003.
END OF TITLE