STUDY ON THE EFFECT OF RESIDUAL STRESS ON THE EXTERNALLY LOADED WELDED STRUCTURE

Rajesh S.R*, Bang Han Sur**, Joo Sung Min*, Kim In Sik***

* Ph D Student, Department of Ocean Engineering and Aerospace Engineering, Chosun University, Korea
** Professor, Department of Ocean Engineering and Aerospace Engineering, Chosun University, Korea
*** Master Student, Department of Ocean Engineering and Aerospace Engineering, Chosun University, Korea

ABSTRACT In the field of welding the behavior of a welded structure under consideration may be predicted via heat transfer and residual stress analysis. In order to facilitate the industrial applications of welding, numerical modeling of heat transfer and residual stress in weldment has been carried out applying Finite Element Method (FEM) and the analysis with the external load including this residual stress due to welding has been done. The present work includes the specialized finite element codes for the calculation of nonlinear heat transfer details and residual stress redistributed along with the external load in the welded structures. A basic interface, which allows models, built in commercial preprocessing package access to the data necessary to build standard input decks for these specialized FEM codes, which are not supported by commercial package. The results from the FEM codes are imported back into commercial package for visualization. In addition the residual stress values are exported to commercial package (such as ANSYS, PATRAN etc.) for further analysis with the external loads, which make the FEM codes fully applicable to the industrial purpose.

Nomenclature

q	: Heat flux
$\lambda_x, \lambda_y, \lambda_z$	: Thermal conductivity along the x,y,z
c	: Specific heat
ρ	: Density
Q	: rate of temperature change
x, y, z	: Cartesian coordinates
ε_T	: Total strain
$\varepsilon^e, \varepsilon^p, \varepsilon^t$	: Elastic, Plastic and Thermal strain
S^e, V^e	: Surface and volume of element
T	: Temperature
t	: time in sec
[B]	: Strain displacement matrix
[D]	: Material property matrix
{F}	: Force vector
[K]	: Stiffness matrix
$[D^e]$	: Elastic material property matrix
[N]	: Shape function
{φ}	: Vector of nodal temperature
α	: coefficient of linear expansion

1. Introduction

There are many practical engineering problems that require the analysis of heat transfer and residual stress and many research works are being carried out in this area of study. In this present study, the analysis of heat transfer and residual stress related to welded structures and its numerical simulation using FEM is carried out.

A finite element codes for the heat transfer and residual stress analysis is developed.

In many cases a significant percentage of the time spent on FEM analysis is devoted to pre-and post processing stage. This is the case for numerous FEM codes used in specialized researches. In the present work an interface is built using the high level language PCL (PATRAN Command Language) that can be compiled directly from PATRAN desktop so that PATRAN can be used as pre- and post processor for the developed codes.

In addition the residual stress calculated by the FEM codes has been applied as the initial loading case in the welded structure and analysis carried out in commercial package with other external loading.

2. Non-linear Heat- transfer analysis

The phenomenon of heat supplied by a

welded arc along x direction can be expressed by Fourier law as [1]

$$q = -\lambda_x \frac{dT}{dx} \tag{1}$$

So for 3D- case the rate of heat transfer is

$$\frac{\partial}{\partial x}\left(\lambda_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\lambda_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(\lambda_z \frac{\partial T}{\partial z}\right) \tag{2}$$

The thermal analysis was conducted using temperature dependent thermal material properties. From conservation of energy the governing equation of heat conduction in weldment is obtained as (considering the medium to be isotropic)

$$\rho c \frac{\partial T}{\partial t} = \lambda\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + Q \tag{3}$$

Applying the error distribution principal known as Galerkin method, and setting the n-integral form to zero and then applying Green-Gauss theorem the elemental formulation can be finally written as [2]

$$[k]\{\phi\} + [c]\left\{\frac{\partial \phi}{\partial t}\right\} = \{f\} \tag{4}$$

where Conductance matrix

$$[k]: -\int_{V^e} \lambda \left(\frac{\partial [N]^T}{\partial x}\frac{\partial [N]}{\partial x} + \frac{\partial [N]^T}{\partial y}\frac{\partial [N]}{\partial y} + \frac{\partial [N]^T}{\partial z}\frac{\partial [N]}{\partial z}\right) dV \tag{5}$$

Heat capacity matrix

$$[c]: -\int_{V^e} \rho c [N]^T [N]\, dV \tag{6}$$

Load vector

$$\{f\}: -\int_{S^e} q [N]^T dS + \int_{V^e} Q [N]^T dV \tag{7}$$

Using this formulation the finite element code is developed for the heat transfer analysis. The developed code consist of 28 subroutines and according to the task performed, subroutines can be grouped accordingly as shown in the block diagram of Fig 1.

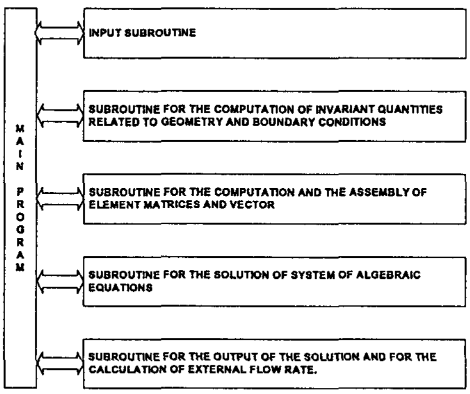

Fig 1. Structure of the Heat transfer analysis program

3. Residual stress analysis

The solid is assumed to be isotropic medium for welding residual stress analysis and mechanical properties of material are dependent on temperature change in both elastic and plastic region. Plastic-flow theory is adopted in the plastic region [3]. Total strain is given by

$$\varepsilon_T = \varepsilon^e + \varepsilon^p + \varepsilon^t \tag{8}$$

By principle of virtual work nodal force increment is

$$\{dF\} = [K]\{dU\} - \{dL\} \tag{9}$$

Where

$$[K] = \int_v [B]^T [D][B] dV \tag{10}$$

$$\{dL\} = \int_v [B]^T [C]\, dT\, dV \tag{11}$$

The total strain increment is

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} + \{d\varepsilon^t\} \tag{12}$$

and the total stress increment is

$$\{d\sigma\} = [D^e]\{d\varepsilon\} - [D^e]\left\{\frac{\partial f}{\partial \sigma}\right\}\lambda - [D^e]\{\alpha\}dT + \frac{d[D^e]}{dT}[D^e]^{-1}\{\sigma\}dT \tag{13}$$

Using this formulation the finite element code is developed for the residual stress analysis.

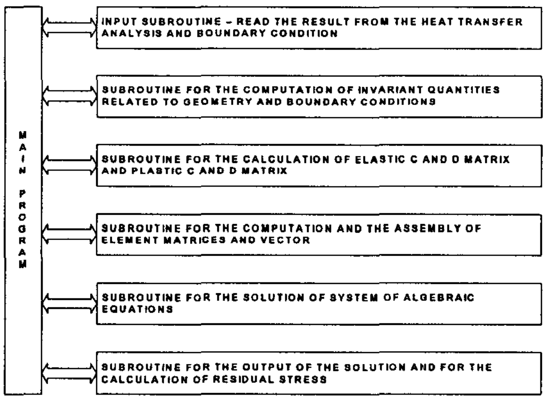

Fig 2. Structure of the Residual stress analysis program

The developed code consist of 47 subroutines and according to the task performed, subroutines can be grouped accordingly as shown in the block diagram of Fig 2.

4. Data transferring details

PATRAN Command Language (PCL) that can be compiled directly from the PATRAN desktop was used for making a basic interface between the in-house solver and commercial package [4]. Three PCL programs were developed. First is WELD.pcl, used to transfer coordinate and connectivity details of the model developed in PATRAN to in-house solver. Second is BC.pcl, used to transfer the details of the surface elements to in-house solver and the last one is MODEL.pcl, used to export the result (residual stresses) from the in-house solver to commercial package for further analysis with the external load. The block diagram for this data transferring between the in-house solver and commercial packages is shown in Fig 3.

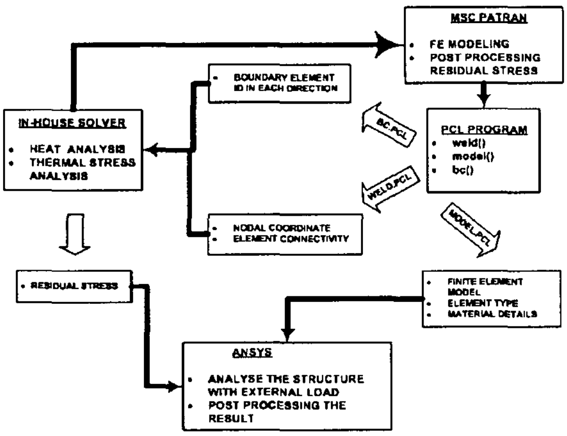

Fig 3. Typical Architecture of the data transferring

5. Result and discussions

A 3D beam model is developed in PATRAN with graded meshing near the weld zone as shown in Fig 4.

The nodal coordinate, the element connectivity and surface element details are transferred to in-house solver and nonlinear heat transfer and residual stress analysis are carried out and the results from the in-house solver is

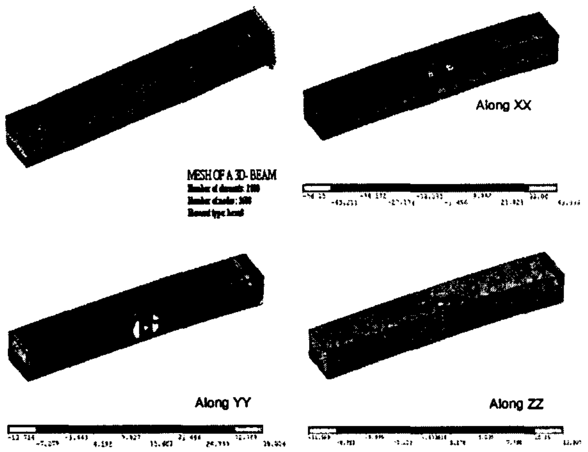

Fig 4. Residual stress contour along X Y and Z direction

transferred back to PATRAN. Then by exporting the model with the residual stress details to commercial package, analysis has been carried out with the application of external load on the model. The total stress contours along the X, Y, and Z direction are as shown in Fig 4. The residual stress, the stress due to external load and total stress values along the longitudinal direction are as shown in Fig 5.

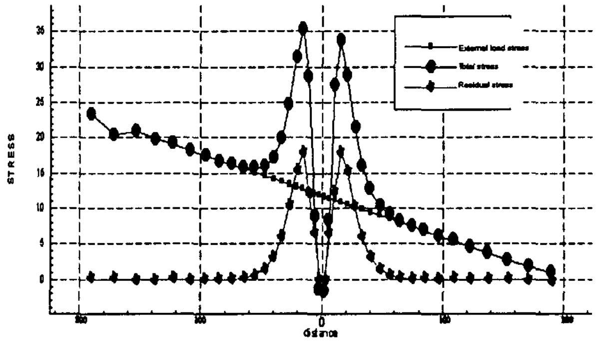

Fig 5. Stress values along longitudinal direction

6. Concluding remarks

Specialized Finite element codes of heat-transfer analysis and residual stress analysis are presented and developed. Related the in-house solver to MSC PATRAN by PCL for pre and post processing. The accuracy of the finite element models is evaluated based on experimental results and the results of the analytical solution. Transferring the residual stress values calculated from the in-house solver to commercial package and further analysis with external loads that address realistic issues in industrial based welding application has been made applicable.

References

1. Lewis R.W., Morgan K., Thomas H.R., and Seetharamu K.N., "The Finite Element Method in heat transfer analysis", John Wiley & Sons, New York, (1996).
2. Comini G., Giudice S.D., and Nonino C., "Finite Element Analysis in Heat Transfer" Taylor & Francis, Italy, (1994).
3. Ueda J., Wang J., Murakawa H., Juan M.G., "Three dimensional Numerical stimulation of various Thermo- Mechanical Processes By FEM" (Report I), Trans. of JWRI, Vol.21, No.2. p.111-117, (1992).
4. MSC Institute of technology, "PAT 304 Customization Course Notes", The MacNeal-Schwendler Corporation, Costa Mesa, CA pp.3-3, (1994).