INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

Bell & Howell Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
800-521-0600

UMI[®]

Computational Mechanics of Welding Complex Structures

by

Mihaela Ancuta Mocanita

B.E. Mechanical, University of Galati, Romania

A thesis submitted to

the Faculty of Graduate Studies and Research

in partial fulfillment of the requirements for the degree of

Master of Engineering

Department of Mechanical and Aerospace Engineering

Ottawa - Carleton Institute for Mechanical and Aerospace Engineering

Carleton University

Ottawa, Canada

September, 2000

© Copyright, 2000

Mihaela Ancuta Mocanita

National Library
of Canada

Acquisitions and
Bibliographic Services

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque nationale
du Canada

Acquisitions et
services bibliographiques

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file Votre référence

Our file Notre référence

The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

L'auteur conserve la propriété du droit d'auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

0-612-57733-3

Canada

The undersigned recommend to the Faculty of Graduate Studies
and Research acceptance of the thesis

Computational Mechanics of Welding Complex Structures

submitted by

Mihaela Ancuta Mocanita

in partial fulfillment of the requirements for

the degree of

Master of Engineering

A handwritten signature in black ink, appearing to read "John Golda", is written over a horizontal line.

Thesis Supervisor

A handwritten signature in black ink, appearing to read "Robert Bell", is written over a horizontal line.

Chair, Department of Mechanical and Aerospace Engineering

Carleton University

Abstract

Computational Weld Mechanics (CWM) methods and principles are employed to simulate the thermal and mechanical response of large welded structures with multiple pass welds. Special computational algorithms were developed to deal with the length scales that range from a fraction of a mm near the weld pool to the overall dimensions of the welded structure. Composite meshing and adaptive meshing were used in order to reduce the cost of computing inherent for a large structure analysis.

The transient equations for heat transfer, the stress and strain evolution and microstructure changes are all solved in a three dimensional(3D) domain. A prescribed temperature field model for the weld pool was used. Temperature dependent material properties are considered in the analysis, as well as filler metal addition. Different constitutive models are used to model the stress and strain fields for weld temperatures varying from room temperature to liquidus temperature.

A methodology to assist engineers in CWM analysis of large scale welded structures with complex geometry and multi-pass welds is presented.

The methods developed in this thesis are used to simulate welding on a pressurized natural gas pipeline and to determine if CWM can provide useful estimates of the risk of burn-through. Computed results agree well with the experimentally measured values.

To my daughter,

Mădălina

Acknowledgments

I would like to thank my thesis supervisor Dr. J.A. Goldak for the energy and time he put in teaching his students. For the good you brought in our lives. thank you, Prof. Goldak. The previous work of the team leaded by Dr. Goldak made possible the results of this thesis.

I would also like to thank the teachers at the Mechanical and Aerospace Department, Carleton University, for all the things I have learned here.

The members of the research team, Andrea Zypchen, Dan Downey, Jianguo Zhou, Victor Aldea, Vikram Kulkarni, Bin Wang, I thank them all for the enjoyable working atmosphere, for the way we shared the happiness and the pain of the research work. Special thanks to Victor and Vicky, who help me write and debug the Tcl scripts.

The financial support of Carleton University, MRCO, NSERC is gratefully acknowledged.

Contents

Acceptance Sheet	ii
Abstract	iii
Dedication	iv
Acknowledgements	v
Table of Contents	vi
List of Figures	xiii
List of Tables	xiv
1 Introduction	1
1.1 General view of the welding phenomena	1
1.2 Computer Aided Design for the analysis of the welds	5
1.3 Thesis organization	7
2 Energy and Stress Solver - FEM Theory and Implementation	10
2.1 Energy solver	11

2.1.1	Law of heat conduction	11
2.1.2	Law of heat transfer by convection	13
2.1.3	Law of heat transfer by radiation	14
2.1.4	The differential equation of heat conduction	15
2.1.5	Initial and boundary conditions	17
2.2	Finite element implementation of a heat transfer problem	20
2.2.1	Domain vs discretization	21
2.2.2	Galerkin's method for space discretization	22
2.2.3	Element equations	24
2.2.4	Global equations	26
2.2.5	Solution in time	27
2.3	Stress solver	30
2.3.1	Constitutive models in welding	32
2.4	Solve on a composite mesh	33
3	Geometric modeling and meshing issues	37
3.1	Geometric modeling of multi pass filler metal	38
3.1.1	The geometry of the joint	40
3.1.2	Define the weld path	40
3.1.3	Define the weld procedure	42
3.1.4	Frenet trihedron (tangent, normal, binormal to a curve)	49

3.1.5	The connection between the weld procedure and weld path . .	51
3.2	Geometric modeling of the weld pool	57
3.3	Composite mesh, adaptive mesh	63
4	Thermal stress analysis of large welded structures	65
4.1	Introduction.	65
4.2	Domain, initial and boundary conditions	67
4.3	Computational Results	69
4.3.1	CWM on large structures	69
4.3.2	Predicting burn-through when welding on pressurized natural gas pipelines	80
4.3.3	Solving on a composite mesh using implicit function represen- tation of the weldpool	103
5	Conclusions	108
	Bibliography	111

List of Figures

3.1	The internal and external boundary of pass 1.	44
3.2	The internal and external boundary of pass 2.	44
3.3	The internal and external boundary of pass 3.	45
3.4	Computed weld procedure for a fillet weld.	47
3.5	Computed weld procedure for a butt weld. For some of the passes, the local coordinate system of each ellipse is rotated relative to the weld procedure coordinate system.	48
3.6	Tangent to a curve.	50
3.7	Trihedron of a curve.	51
3.8	Filler metal passes for a fillet weld.	55
3.9	Filler metal passes for a butt weld. The weld procedure is swept along an arbitrary path in space.	56
3.10	The reference weld pool mesh.	58
3.11	The physical weld pool A and one of the chain bricks.	60

3.12	The physical weldpool B.	61
3.13	The weld pool mesh WP1 and the complete welded structure mesh. .	64
4.1	An example of a weld procedure is shown.	73
4.2	An example of a curvilinear coordinate system for a weld Tee-joint. .	75
4.3	The meshes created around each weld joint of the welded structure. .	77
4.4	The straight weld mesh.	77
4.5	The mesh for the tee joint.	77
4.6	The mesh for the circular joint.	77
4.7	The thermal field as a result of the FEM analysis performed on the girth weld.	79
4.8	The first pass thermal analysis for the tee joint.	79
4.9	The second pass thermal analysis for the tee joint.	79
4.10	Thermal conductivity vs. temperature.	87
4.11	Specific heat vs. temperature.	88
4.12	Young's modulus vs. temperature.	88
4.13	Yield strength vs. temperature.	88
4.14	Poisson's ratio vs. temperature.	88
4.15	Viscosity vs. temperature.	89
4.16	Temperature field distribution shown on the entire test pipe.	91

4.17 Microstructure analysis results for weld E: figure showing 0.9 and 0.1 fraction austenite isosurfaces.	92
4.18 Microstructure analysis results for weld E: figure showing 0.8 and 0.2 fraction ferrite isosurfaces.	92
4.19 Microstructure analysis results for weld E: figure showing 0.085 and 0.015 fraction pearlite isosurfaces.	92
4.20 Microstructure analysis results for weld E: figure showing 0.9 and 0.1 fraction bainite isosurfaces.	92
4.21 Microstructure analysis results for weld E: computed phase fractions along a line parallel to the welding direction.	93
4.22 FEM analysis of weld A.	95
4.23 Experimental results of weld A.	95
4.24 FEM analysis of weld B.	95
4.25 Experimental results of weld B.	95
4.26 FEM analysis of weld D.	96
4.27 Experimental results of weld D.	96
4.28 FEM analysis of weld E.	96
4.29 Experimental results of weld E.	96
4.30 FEM analysis of weld G.	97
4.31 Experimental results of weld G.	97

4.32 FEM analysis of weld H.	97
4.33 Experimental results of weld H.	97
4.34 FEM analysis of weld I.	98
4.35 Experimental results of weld I.	98
4.36 FEM analysis of weld J.	98
4.37 Experimental results of weld J.	98
4.38 FEM analysis of weld K.	99
4.39 Experimental results of weld K.	99
4.40 FEM analysis of weld L.	99
4.41 Experimental results of weld L.	99
4.42 Temperature profile on the inside wall surface for the “safe” welds. . .	102
4.43 Temperature profile on the inside wall surface for the “marginal” welds.	102
4.44 The mesh shown for time step 5 of the analysis.	105
4.45 The mesh shown for time step 10 of the analysis.	105
4.46 The mesh shown for time step 20 of the analysis.	105
4.47 The mesh shown for time step 30 of the analysis.	105
4.48 The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $10 \text{ W/m}^2 \text{ K}$ and a welding speed of 0.7 cm/s.	107
4.49 The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $5000 \text{ W/m}^2 \text{ K}$ and a welding speed of 0.7 cm/s.	107

4.50	The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $10 \text{ W/m}^2 \text{ K}$ and a welding speed of 7 cm/s.	107
4.51	The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $5000 \text{ W/m}^2 \text{ K}$ and a welding speed of 7 cm/s.	107

List of Tables

4.1	The positions of the test welds on the pressure vessel used for the burn through experiments.	82
4.2	The cross-section dimensions of the test welds.	84
4.3	The actual welding parameters for burn-through potential experiments.	85
4.4	Maximum temperatures on the internal pipe wall.	101

Chapter 1

Introduction

1.1 General view of the welding phenomena

A weld can be simply described as a region of bonding between adjacent parts. Ideally the mechanical, metallurgical and chemical properties of the weld should be identical or superior to those of the parts being joined. The primary requirement of a successful bonding operation is to bring the metal surfaces into intimate contact, that is, the atoms of the two surfaces have to be brought together within atomic distances. A welding process, therefore, has to remove the oxide layers, and bring the surfaces into close contact, for a bond to be made. Depending on the welding process, the oxide film is removed in several ways: mechanically (by compressing the joint surfaces as in cold-pressure welding, by rubbing the surfaces together as in friction welding)

or thermally (in fusion welds, the oxide film floats off the molten pool and can be disrupted by the welding arc action).

In any welding process that is being used, generation of heat is involved. In the solid-state bonding processes, the heat reduces the yield strength and allows easier plastic flow. In the fusion processes, the adjacent surfaces are melted together and linked in the solidification process. The direct addition or indirect evolution of heat is one of the keys of the welding processes, but it is also the source of many of the welding engineer's problems.

Why welding?

There are many reasons for using welding design and construction. Some of the advantages and disadvantages of welded structures are presented below, in comparison with the riveted structures.

Advantages of welded structures

High joint efficiency A properly welded joint is stronger than the material joined.

This is called over matching. The fused joints create a compact and rigid structure that cannot be obtained using riveted joints. Welded joints are better for fatigue loads, impact loads and severe vibration.

Water and air tightness Welding is an ideal technology to assemble structures which require water and air tightness.

Weight saving The weight of a hull structure (and therefore the static loading) can often be reduced by at least 10 to 20 per cent if welding is used.

No limit on thickness It is difficult to rivet plates that are more than 5 cm thick. In welded structures there is virtually no limit to the thickness.

Simple structural design In riveted structures, complex joints are required. Welding permits the architect and structural engineer to develop and use modern and economical design principles.

Reduction in fabrication time and cost A welded structure can be fabricated in a shorter period of time. Less time is required on detailing, layout and fabrication since fewer pieces are used. Punching, drilling, reaming or countersinking are eliminated. Automation is also easier to achieve in the case of welding.

Problems with welded structures

Some of the major difficulties with welded structures are:

Possibility of defects Porosity due to hydrogen, oxygen and nitrogen contamination, hot and cold cracking, are some of the possible defects of a weld.

Sensitive to materials The physical properties of the metals or alloys being joined influence the efficiency and applicability of the various welding processes. The thermal conductivity and the thermal expansion of the materials being welded have a direct effect on the distortion of the weldment. In the case of resistance welding, base metal resistivity, thermal conductivity and specific heat influence the power requirements. For the arc welding process, the arc initiation and stability are influenced by the ionization potentials of the metal being welded.

Residual stress and distortion Distortion or residual strain as a result of the welding process increases the cost of fabricating welded structures and may influence their performance or external appearance.

In order to design and fabricate a quality welded structure, it is essential to have an adequate design, a proper selection of materials, adequate equipment and proper welding procedures, skilled workmanship, and strict quality control.

Phenomena during welding

The modeling of welds is a complex process that involves such topics as:

- arc physics
- heat transfer and fluid flow
- surface tension effects

- solidification
- solid-state transformations
- development of mechanical properties for the fusion zone and the heat affected zone
- plastic deformation and the creation of residual stress and distortion

An ideal weld process simulation will have to consider each of these phenomena in relation one with the other. The multitude of influencing parameters and their non-linear, transient and temperature dependent effect have to be considered in order to produce an accurate computation.

1.2 Computer Aided Design for the analysis of the welds

Because of their capability to store, manage and process large amounts of data, computers are good engineering tools, which allow objectives to be reached in a cost and time-effective way. Modeling and simulation have become a well defined stage in the design process of any engineering application.

Today's increase in computer power and availability allows numerical methods of calculating temperatures, stresses and distortions during welding to be used to predict the weld performance and quality. Welding engineers can use such techniques to reduce the costs of the testing process to establish optimum welding conditions. The process of modeling the behavior of a welded structure has to consider that complex weld joints and weld types are to be modeled. Simple weld modeling does not satisfy industry requirements because:

- under welding conditions large structures behave differently (especially the distortion and residual stresses) than small test specimens.
- adaptive meshing refinement is required through the whole structure to capture the sharp gradients that occur.

Finite element techniques permit the complex geometry and temperature dependent material properties to be taken into account, limited only by the available computing resources. Diverse welding processes can be simulated by changing the heat input, preheat conditions and imposed cooling patterns.

The process of welding a structure follows these stages:

- Structural design - specifies the geometry and material for each of the parts to be welded, the location of each weld, the loads and service conditions that the joint must withstand.

- Weld procedure design - specifies the welding procedure for each pass in a weld joint: weld process type, consumables, current, voltage, speed, weld pool shape and size, preheat and post heat.
- Weld production design - specifies the sequence of weld joints in a structure, the start and end of each weld, the fixturing to be used.

The computation process has to be designed so that the information flow between these stages is continuous and complete.

1.3 Thesis organization

This thesis presents the fundamental principles of CWM and its application in practice. The approach in solving welding problems focuses on providing tools to assist the welding analyst in the design and production of the welded structures. Different meshing strategies and special solvers are used to solve the thermal, stress and microstructure fields evolving in a large structure as a result of welding.

Chapter 1 introduces some of the welding process characteristics.

Chapter 2 presents part of the theoretical aspects behind the phenomena as a result of applying the welding process. The temperature, displacement, strains

and stresses in a weld and a welded structure are computed solving the differential equations governing the heat transfer and the stress state in a Lagrangian frame. FEM is used as a numerical simulation method.

Chapter 3 develops various algorithms to model the geometry of the welded structure. The concepts of weld procedure, weld joint, weld path are introduced as a part of the industrial practice requirements for weld specification.

Chapter 4 presents the results of the thermal, stress and microstructure analysis for large welded structures. A CWM framework is developed in an object oriented environment to support the large amount of data input for the analysis. An accurate modeling of the real physics of the weld was obtained by coupling the thermal and stress field computations. Where possible, the computed results were compared with experimental results.

The software framework used for this research uses C++ objects and Tcl as a glue language. Tcl is a scripting language. C++ objects are instantiated and C++ methods are executed from the Tcl language. This thesis project used the MMO software (version 2000) developed in C++ by Dr. John Goldak, Dr. Ed Hughes, Andrea Zypchen, Jianguo Zhou, Daniel Downey and others. The Energy Solver class, the Stress Solver class and the Mesh class are used to obtain a solution of the thermal, stress and displacement fields. The algorithms presented in this thesis are developed and implemented by the author based on discussions with Prof. John Goldak.

I worked in a team of about 15 researchers developing software for designers of manufacturing processes and parts. Problems arose almost every day, ideas discussed, proposals made and criticized. My primary responsibility was the work presented in this thesis. However, many people contributed ideas and particularly software tools that made this thesis possible.

Chapter 2

Energy and Stress Solver - FEM

Theory and Implementation

The origin of welding residual stresses and welding distortion is the concentrated heat input characteristic of welding processes. The temperature fields during welding are extremely heterogeneous and transient. Temperatures can vary from $-40\text{ deg } C$ (when the components are welded in severe winter conditions) to $3000\text{ deg } C$ (the local maximum temperature near the weld pool surface in the case of steel). At temperatures above the melting point the base metal and filler metal fuse, and metallurgical processes evolve in the weld pool. Regions of the metal solidify, recrystallize and microstructural transformations take place during heating and cooling. The temperature field, therefore, drives the welding residual stresses not only directly through

the thermal strains, but also indirectly through the transformations strains which accompany the changes in state and microstructure.

The basic principles of transient heat conduction and the finite element formulation used in the present welding analysis are described in this chapter.

2.1 Energy solver

In studies of heat transfer three distinct modes of heat transfer are considered: *conduction, convection and radiation*. The temperature distribution in a medium is controlled by the combined effects of these three modes of heat transfer. In the welding processes the concentrated heat input generated by the welding source propagates rapidly through the workpiece components by conduction. Radiation and convection play an important role in heat losses in and near the welding arc and at the surface of the components.

2.1.1 Law of heat conduction

Conduction is the mode of heat transfer in which energy flows from a region of high temperature to a region of low temperature by the kinetic motion or direct impact of

molecules with no mass flow. In a solid that is a good electrical conductor, a large number of free electrons move about the lattice. Electrons are also carriers of heat and therefore materials that are good electric conductors are generally good heat conductors. Fourier's law of heat conduction, based on experimental observations, states that the heat flow density or flux q [J/m^2s] is proportional to the temperature gradient $T_{,n}$ [K/m] through the coefficient of thermal conductivity κ [J/msK] [1]:

$$q = -\kappa T_{,n} \quad (2.1)$$

In vector form, equation 2.1 can be written [2]:

$$\mathbf{q}(\mathbf{r}, t) = -\kappa \nabla T(\mathbf{r}, t) \quad (2.2)$$

where the temperature gradient is a vector normal to the isothermal surface. The heat flux vector $\mathbf{q}(\mathbf{r}, t)$ represents the heat flow per unit time, per unit area of the isothermal surface in the direction of decreasing temperature. The thermal conductivity of an isotropic material can be represented by a positive, scalar quantity times the 3×3 identity matrix. Since the heat flux vector points in the direction of decreasing temperature, the minus sign is included in the equation to reflect that the heat flows down the temperature gradient.

In a rectangular coordinate system, equation 2.2 becomes:

$$\mathbf{q}(x, y, z, t) = -\mathbf{i}\kappa T_{,x} - \mathbf{j}\kappa T_{,y} - \mathbf{k}\kappa T_{,z} \quad (2.3)$$

where $\mathbf{i}$, $\mathbf{j}$ and $\mathbf{k}$ are the unit direction vectors along the x , y and z directions.

Thermal conductivity is an important property, which controls the heat flow in the medium and it depends on the chemical composition, the microstructural state and the temperature of the material.

2.1.2 Law of heat transfer by convection

When fluid flows over a solid body or inside a channel while the temperatures of the fluid and the solid surface are different, the heat transfer between the fluid and the solid surface takes place as a consequence of the motion of fluid relative to the surface.

This mechanism of heat transfer from the solid is called *convection*.

As the temperature in the fluid is influenced by the fluid motion, heat transfer by convection is a complicated phenomena. In engineering applications, to simplify heat-transfer calculations the heat flux q_{conv} is considered proportional to the difference between the surface temperature T_w and the mean temperature T_f of the fluid flowing outside the thermal boundary layer [1]:

$$q_{conv} = h (T_w - T_f) \quad (2.4)$$

The coefficient of convective heat transfer h is the conductance of the thermal boundary layer and depends on the flow conditions on the surface, on the surface properties

and on the properties of the flowing medium.

2.1.3 Law of heat transfer by radiation

According to the Stefan-Boltzmann's law, the heat flux q_{rad} radiated per unit area and time from a heated body is proportional to the fourth power of the surface temperature T through the radiation coefficient ϵC_o [1]:

$$q_{rad} = \epsilon C_o T^4$$

The radiation coefficient $C_o = 5.67 * 10^{-14} [J/mm^2 K^4]$ applies to an “absolutely black body”. A “grey body” is characterized by the blackness degree $\epsilon < 1.0$. For the case of a relatively small body in extensive surroundings (encountered in welding technology), heat dissipation occurs by means of radiation according to the equation:

$$q_{rad} = \epsilon C_o (T^4 - T_o^4) \tag{2.5}$$

where T_o is the temperature radiating back to the body.

2.1.4 The differential equation of heat conduction

The conservation of energy can be written for a small control volume V in a domain as:

$$\begin{aligned} &\text{Rate of heat entering through the bounding surfaces of } V + \\ &\text{rate of energy generation in } V = \text{rate of storage of energy in } V \end{aligned} \quad (2.6)$$

The terms in equation 2.6 can be evaluated as:

$$\begin{aligned} &\text{Rate of heat entering by conduction through the bounding surfaces of } V = \\ &= - \int_A \mathbf{q} \cdot \mathbf{n} dA = - \int_V \nabla \cdot \mathbf{q} dv \end{aligned} \quad (2.7)$$

where A is the surface area of the volume element V , $\mathbf{n}$ is the outward normal unit vector to the surface element dA , $\mathbf{q}$ is the heat flux vector at dA . The minus sign ensures that the heat flow is from hot to cold, and the divergence theorem is used to convert the surface integral to a volume integral.

$$\text{Rate of energy generation in } V = \int_V Q(\mathbf{r}, t) dv \quad (2.8)$$

where $Q(\mathbf{r}, t)$ is the heat generation rate in the medium, specified as heat generation per unit time, per unit volume.

In the case of solids and liquids the specific heats under constant volume and pressure are equal, that is, $C_p = C_v$. Then, the rate of increase of internal energy is

reflected in the rate of storage in the volume V :

$$\text{Rate of energy storage in } V = \int_V \rho C_p T(\mathbf{r}, t)_{,t} dv \quad (2.9)$$

where the specific heat C_p and the density ρ do not vary with time. Substituting in equation 2.6:

$$\int_V \left[-\nabla \cdot \mathbf{q}(\mathbf{r}, t) + Q(\mathbf{r}, t) - \rho C_p T(\mathbf{r}, t)_{,t} \right] dv = 0 \quad (2.10)$$

Because the integral is true for any volume, it must be true in the neighbourhood of any point in the domain:

$$-\nabla \cdot \mathbf{q}(\mathbf{r}, t) + Q(\mathbf{r}, t) = \rho C_p T(\mathbf{r}, t)_{,t} dv \quad (2.11)$$

Substituting $\mathbf{q}(\mathbf{r}, t)$ from equation 2.2, we obtain the *differential equation of heat conduction* for a stationary, homogeneous, isotropic solid with heat generation within the body as:

$$\nabla \cdot [\kappa \nabla T(\mathbf{r}, t)] + Q(\mathbf{r}, t) = \rho C_p T(\mathbf{r}, t)_{,t} \quad (2.12)$$

This equation is valid for temperature or space dependent κ as well as temperature dependent C_p , in which case the equation is nonlinear. When the thermal conductivity is assumed to be constant (i.e., independent of position and temperature), equation 2.12 simplifies to:

$$\nabla^2 T(\mathbf{r}, t) + \frac{1}{\kappa} Q(\mathbf{r}, t) = \frac{1}{\alpha} T(\mathbf{r}, t)_{,t} \quad (2.13)$$

where

$$\alpha = \frac{\kappa}{\rho C_p}$$

is the *thermal diffusivity*. The physical significance of thermal diffusivity is associated with the speed of propagation of heat into the solid during changes of temperature with time. The higher the diffusivity, the faster the propagation of heat in the medium.

In a rectangular coordinate system (x, y, z) , at every point in the domain Ω , equation 2.12 becomes:

$$(\kappa_x T_{,x})_{,x} + (\kappa_y T_{,y})_{,y} + (\kappa_z T_{,z})_{,z} + Q = \rho C_p T_{,t} \quad (2.14)$$

For an isotropic medium the thermal conductivities in x, y, z directions are equal, i.e., $\kappa_x = \kappa_y = \kappa_z$.

2.1.5 Initial and boundary conditions

The energy equation, equation 2.12, has a unique solution for given initial and boundary conditions. The initial condition specifies the temperature distribution in the medium at the origin of the time coordinate ($t = 0$). In a rectangular system (x, y, z) , the initial condition of every point in the domain of the equation 2.14 $(x, y, z) \in \Omega$ can be written as:

$$T(x, y, z, 0) = T_0(x, y, z) \quad (2.15)$$

The partial differential equation 2.14 is well posed and a unique solution exists if at every point on the boundary $\Gamma = \Gamma_1 + \Gamma_2$ of the domain Ω either temperature or flux is specified.

The Neumann boundary conditions specify the flux at the boundary surfaces of the structure and can be derived by writing an energy balance equation at a point on the surface of a solid subject to convection, radiation and/or external flux:

$$q_n = q_{conv} + q_{rad} + q_{sup} \quad (2.16)$$

The flux q_{sup} represents energy supplied to the surface from an external source. The quantity q_{conv} represents the convection flux with a heat transfer coefficient h into an external ambient at temperature T_{amb} and is given by

$$q_{conv} = h(T - T_{amb}) \quad (2.17)$$

The flux q_{rad} represents the heat loss from the surface by radiation into an external surface at temperature T_{amb} and is given, as shown above, by

$$q_{rad} = \epsilon C_o (T^4 - T_{amb}^4) \quad (2.18)$$

The quantity q_n represents the conduction heat flux vector normal to the surface elements and is given by the relation:

$$q_n = \mathbf{q} \cdot \mathbf{n} = -\kappa \nabla T \cdot \mathbf{n} \quad (2.19)$$

Substituting equations 2.17 to 2.19 to equation 2.16, the boundary condition becomes:

$$-\kappa T_{,n} = h(T - T_{amb}) + \epsilon C_o (T^4 - T_{amb}^4) + q_{sup} \quad (2.20)$$

The Neumann boundary conditions constrain the gradients or derivatives of the unknowns specified on a part of the boundary Γ_2 . They are sometimes called *natural* boundary conditions. In a Cartesian coordinate system, equation 2.20 can be written as:

$$\begin{aligned} \kappa T_{,x} n_x + \kappa T_{,y} n_y + \kappa T_{,z} n_z = h(T - T_{amb}) + \\ \epsilon C_o (T^4 - T_{amb}^4) + q_{sup} \quad \text{on the boundary } \Gamma_2 \end{aligned} \quad (2.21)$$

where n_x, n_y, n_z are the direction cosines (i.e., cosine of angles) of the unit normal vector $\mathbf{n}$ with the x, y, z coordinate axes.

The Dirichlet boundary conditions are commonly specified in terms of known values of the unknowns on a part of the boundary Γ_1 . Because they are imposed values of the unknown variable, they are also called *essential* or *forced* boundary conditions. The prescribed temperature distribution at the boundary surface can be expressed as:

$$T = T_p(\mathbf{r}, t) \quad \text{on the boundary } \mathbf{r} \in \Gamma_1 \quad (2.22)$$

where the prescribed surface temperature $T_p(\mathbf{r}, t)$ is a function of position and time.

Or, writing equation 2.22 in a rectangular coordinate system (x, y, z) :

$$T = T_p(x, y, z, t) \quad \text{on the boundary } \Gamma_1 \quad (2.23)$$

2.2 Finite element implementation of a heat transfer problem

To obtain a solution to the partial differential equation of heat flow, numerical methods are used. Among them, the finite element method has developed into one of the most general analysis tool. The success of the finite element method is based largely on the basic finite element procedures used: the formulation of the problem in variational or weighted residual form, the finite element discretization of this formulation, and the effective solution of the resulting finite element equations. These basic steps are the same regardless of the problem that is being solved, providing a general framework and a natural approach to solving a large class of engineering problems.

The solution process for time-dependent problems usually involves two major steps: (1) discretization of space and then (2) propagation or marching in time at

various time levels using (often implicit) time stepping methods.

2.2.1 Domain vs discretization

In order to obtain a finite element solution to the heat equation, the first step is the *discretization* of the continuum body or structure that constitutes the domain on which the heat equation is to be solved into a finite number of subdomains called *elements*. The advantage of the discretization is that the analysis needs only to be applied to the individual elements of certain simple geometry rather than the entire solid of complex geometry. For each particular problem the number, size and shape of the elements are chosen so the desired computational performance (computing time and/or accuracy of the solution) is achieved. The elements are assumed to be interconnected at a discrete number of points called *nodes* situated on their boundaries.

The physical quantities to be determined in the element, i.e., temperature, can be obtained from the values at the nodes through certain *basis*, *shape* or *interpolation* functions as described by the following equation:

$$T(x, y, z, t) = \sum_{i=1}^{nodes} N_i(x, y, z) T_i(t) \quad (2.24)$$

where $N_i(x, y, z)$ are the basis functions dependent only on the type of element and its geometry. The time dependence is included in the nodal values of the temperature as $T_i(t)$.

The gradient of the temperature is approximated as:

$$\nabla T(x, y, z, t) = \sum_{i=1}^{nodes} T_i(t) \nabla N_i(x, y, z) \quad (2.25)$$

The Laplacian of the temperature is approximated as:

$$\nabla \cdot \nabla T(x, y, z, t) = \sum_{i=1}^{nodes} T_i(t) \nabla \cdot \nabla N_i(x, y, z) \quad (2.26)$$

The next step is to evaluate the nodal temperatures $T_i(t)$. One of the major alternatives serving this purpose is the method of weighted residuals, as presented in the following section.

2.2.2 Galerkin's method for space discretization

The method of weighted residuals is a projection method for obtaining approximate solutions to linear and nonlinear partial differential equations. Most of the finite element applications use this method to formulate the finite element equations.

The first step in applying the method is to assume a functional representation of the dependent field variable in some way so that it approximately satisfies the given differential equation and boundary condition. Substitution of this approximation into the original differential equation and boundary conditions results in some error called *residual*. The residual is then minimized by an orthogonal projection onto a finite dimensional function space.

Considering the heat transfer problem, when the approximate solution (2.24) is substituted into the energy equation we obtain:

$$\nabla \cdot \left(-k \sum_{i=1}^{nodes} T_i(t) \nabla N_i(x, y, z) \right) + Q = R(x, y, z) \neq 0 \quad (2.27)$$

where R is the residual or error given by the approximation operation. When equation 2.24 is the exact solution, the error in the FEM solution is zero. Mathematically the idea of making the residual orthogonal to the test function space can be expressed as:

$$\int_{\Omega} R(x, y, z) W_i(x, y, z) d\Omega = 0 \quad (2.28)$$

where $W_i(x, y, z)$ denotes the basis functions in the test or weighting function space.

Galerkin's method chooses the weighting functions to be a subset as the basis functions $N_i(x, y, z)$ of the trial function space that is used in the discretization of the domain. Considering the integral form of the heat equation, Galerkin's formulation becomes:

$$\int_{\Omega} (\kappa \nabla^2 T + Q - \rho C_p T_{,t}) N_i d\Omega = 0 \quad (2.29)$$

The basis functions N_i are a function of space and the coefficients T_i are a function of time and space.

Applying Green's identity to the first term of the volume integral, equation 2.29 takes the form:

$$\int_{\Omega} \nabla N_i \kappa \nabla T d\Omega - \int_{\Gamma} N_i \kappa \nabla T \cdot \mathbf{n} d\Gamma - \int_{\Omega} Q N_i d\Omega + \int_{\Omega} \rho C_p T_{,t} N_i d\Omega = 0 \quad (2.30)$$

Equation 2.30 is known as the *weak form* of the heat conduction equation. It admits discontinuous thermal conductivity coefficients κ and discontinuous first derivatives of temperature $T(x, y, z, t)$, which are not allowed by the differential form.

2.2.3 Element equations

Substituting for the element temperature $T(x, y, z, t)$ according to equation 2.24 and expressing the quantities in matrix form, equation 2.30 becomes:

$$\begin{aligned} \int_{\Omega} \kappa \nabla [N_i] \cdot \nabla [N_i]^T \cdot [T_i] d\Omega &- \int_{\Gamma} \kappa [N_i] \cdot \nabla [N_i]^T \cdot [T_i] \cdot \mathbf{n} d\Gamma - \\ &- \int_{\Omega} Q [N_i] d\Omega + \int_{\Omega} \rho C_p [T_i]_{,t} [N_i]^T [N_i] d\Omega = 0 \end{aligned} \quad (2.31)$$

Applying the boundary conditions conforming with equation 2.20, the above equation can be written:

$$\begin{aligned} \int_{\Omega} \kappa \nabla [N_i] \cdot \nabla [N_i]^T \cdot [T_i] d\Omega &- \int_{\Omega} Q [N_i] d\Omega + \int_{\Omega} \rho C_p [T_i]_{,t} [N_i]^T [N_i] d\Omega - \\ - \int_{\Gamma_2} [N_i] (h([N_i]^T [T_i] - T_{amb}) &+ \epsilon C_o([N_i]^T [T_i]^4 - T_{amb}^4) - q_{sup} [N_i]^T) d\Gamma = 0 \end{aligned} \quad (2.32)$$

where

$[N_i]$ is the vector of the basis functions and its transpose is $[N_i]^T$.

$[T_i]$ is a column vector of nodal temperatures.

$[B]$ is the matrix of the basis functions derivatives with respect to space that maps the nodal temperatures to the temperature gradient ∇T .

$[D]$ is the thermal conductivity matrix.

$[\dot{T}_i]$ is the column vector for the rate of change of nodal temperatures with respect to time.

$[Q_i]$ is the column vector of nodal heat sources.

With these notations, equation 2.32 becomes:

$$\begin{aligned} \int_{\Omega} [B]^T [D] [B] [T_i] d\Omega + \int_{\Gamma_2} h [N_i] [N_i]^T [T_i] d\Gamma + \int_{\Omega} \rho C_p [N_i] [N_i]^T [\dot{T}_i] d\Omega = \\ = \int_{\Omega} [Q_i] [N_i] d\Omega + \int_{\Gamma_2} h [N_i] [N_i]^T T_{amb} d\Gamma + \int_{\Gamma_2} q_{sup} [N_i] [N_i]^T d\Gamma \end{aligned} \quad (2.33)$$

Denoting the matrices as follows:

$$\int_{\Omega} \rho C_p [N_i] [N_i]^T d\Omega = [C], \quad \text{as the capacitance matrix}$$

$$\int_{\Omega} [B]^T [D] [B] d\Omega + \int_{\Gamma_2} h [N_i] [N_i]^T d\Gamma = [K], \quad \text{the stiffness matrix}$$

$$\int_{\Gamma_2} h [N_i] [N_i]^T T_{amb} d\Gamma + \int_{\Omega} [Q_i] [N_i] [N_i]^T + \int_{\Gamma_2} q_{sup} [N_i] [N_i]^T d\Gamma = [F_i], \quad \text{as}$$

the external force vector, the element equations in matrix form are obtained:

$$[C][\dot{T}_i] + [K][T_i] = [F_i] \quad (2.34)$$

2.2.4 Global equations

The matrices represented in equation 2.34 are evaluated at Gauss points at the element level and then assembled element-by-element over the entire domain to form the global equation. The global non-linear set of differential equations in matrix form will be:

$$[C_g][\dot{T}] + [K_g][T] = [F] \quad (2.35)$$

where

$$[C_g] = \sum_1^{nbElements} [C]; [K_g] = \sum_1^{nbElements} [K]; [F] = \sum_1^{nbElements} [F_i]$$

The set of equations given by equation 2.35 is non-linear since $[K]$, $[C]$ and $[F]$ depend on $[T]$.

2.2.5 Solution in time

For time dependent or evolution problems (such as the heat equation), the finite element method is used to formulate the solution in the physical space. To obtain the solution over a range of time, a finite difference scheme can be used. The various finite-difference schemes available for the finite difference representation of the time-dependent heat problem can be grouped into two categories: *explicit schemes* and *implicit schemes*. Each method has its own advantages and disadvantages.

For example, the equations derived from the explicit method are uncoupled and simple to solve, but their solution is stable under restrictive conditions. If the space step ds is chosen very small to improve the accuracy of the calculations, the computational problem becomes very expensive because the time step dt must be chosen very small to satisfy the stability conditions. On the other hand, the implicit method offers coupled equations that are more difficult to solve, but their solution is not limited by stability criteria.

In the *fully implicit* or *backward* scheme, the finite-difference representation of the heat equation (for the one dimensional case) is given as:

$$\frac{T_n^{i+1} - T_n^i}{\Delta t} = \alpha \frac{T_{n-1}^{i+1} + T_{n+1}^{i+1} - 2T_n^{i+1}}{\Delta x^2} \quad (2.36)$$

where T_n^i is the temperature of node n at the instant of time i and α is the thermal diffusivity as defined earlier. The spacial derivatives on the right-hand side are evalu-

ated at time step $i + 1$. The left hand term originates from writing the first derivative of temperature with respect to time as:

$$T_{.t} = \frac{T_n^{i+1} - T_n^i}{\Delta t} \quad (2.37)$$

Equation 2.36 provides $N+1$ coupled (simultaneous) algebraic equations for the $N+1$ unknown nodal temperatures T_n^{i+1} ($n = 0, 1, 2, \dots, N$). The scheme is unconditionally stable and the correct equilibrium solution is obtained, but it is only first order accurate.

Once the values of the boundary condition functions, the heat transfer coefficients, the thermal conductivity have been set, the solver executes the following steps for each time step:

Trial Temperature The energy equation in the form

$C_p h_{.t} + \nabla \cdot (-\kappa \nabla T) = 0$ is solved for $T(t_3)$. This is a trial solution for the temperature at time t_3 .

Enthalpy Change As the specific enthalpy $h(T)$ and the heat capacity $C_p(T)$ are related by:

$$h(T) = \int_{T_o}^{T_1} C_p(T) dT + \int_o^f f_i L_i df$$

where f_i is the fraction of the phase i and L_i is the latent heat of the phase i transformation, the change in specific enthalpy h due to the temperature change $T(t_3) - T(t_1)$ can be written as:

$\Delta h = C_p (T(t_3) - T(t_1))$. The value of specific enthalpy at the end of the time step is:

$$h(t_3) = h(t_1) + \Delta h.$$

Final Temperature At the end of the last time step the final temperature $T(t_3) = T(h(t_3))$ is found as a function of enthalpy from the temperature/enthalpy table of the alloy stored in the material library.

If greater accuracy was desired, one could iterate this time step. To do this, compute the residual:

$$h_{,t} + \nabla \cdot (-\kappa \nabla T) = residual.$$

However, such iterations usually are not done. The error arises because in general $C_p T_{,t} \neq h_{,t}$. In a sense, the error is self-correcting. If ΔT is too small, too little heat will flow at this point in this time step. If it is too large, too much heat will flow at this point in this time step. However, energy is still conserved and so the error tends not to accumulate.

2.3 Stress solver

The thermal stress analysis computes the stress, strain and displacement fields by solving the basic equations defining stress: conservation of momentum, conservation of mass, constitutive equations and compatibility strain-displacement relation:

$$\nabla \cdot \sigma + b = 0 \quad (2.38)$$

$$\frac{D\rho}{Dt} + \rho \nabla \cdot v = \rho_{,t} + \nabla \cdot (\rho v) = 0 \quad (2.39)$$

$$\sigma = D\epsilon \quad (2.40)$$

$$\epsilon = \frac{1}{2}(\nabla u + \nabla u^T) \quad (2.41)$$

where σ is the Cauchy stress, b is the sum of the body force, ρ is the density. D is the material property tensor, ϵ is the infinitesimal strain and u is the displacement.

Given a set of trial functions, N , and a set of weight functions, W , the weak formulation for the stress problem is:

$$\int_{\Omega} W(\nabla \cdot \sigma + b) d\Omega = 0 \quad (2.42)$$

Green's identity can be applied to the first term of equation 2.42 to yield:

$$\begin{aligned} \int_{\Omega} W(\nabla \cdot \sigma) d\Omega &= - \int_{\Omega} \nabla W \sigma d\Omega + \int_{\Gamma_2} W(\sigma \cdot n) d\Gamma \\ - \int_{\Omega} \nabla W^T \sigma d\Omega + \int_{\Omega} W^T b d\Omega + \int_{\Gamma_2} W(\sigma \cdot n) d\Gamma &= 0 \end{aligned} \quad (2.43)$$

The weak form (equation 2.43) , known as the *virtual work equation* or *virtual work principle*, is of fundamental importance in solid mechanics. This principle states that for any compatible, small virtual displacements imposed on a body in equilibrium, the internal virtual work is equal to the external virtual work:

$$\int_{\Omega} \epsilon^{*T} \sigma d\Omega = \int_{\Omega} \delta^{*T} b d\Omega + \int_{\Gamma} \delta^{*T} \tau d\Gamma \quad (2.44)$$

The internal virtual work is given on the left side of equation 2.44 and is equal to the actual stress σ going through the virtual strain ϵ^* that corresponds to the virtual displacement δ^* . The external work of the actual external forces (body forces b and surface tractions τ) going through the virtual displacements is given on the right side of equation 2.44. Equation 2.44 is an expression of equilibrium of the body. However, the equation also contains the compatibility and constitutive requirements. To obtain an approximate FEM solution, the trial functions are chosen so that:

$$\begin{aligned} \delta^* &= N_i(x, y, z) \delta_i^* \\ \delta^{*T} &= \delta_i^{*T} N_i^T(x, y, z) \end{aligned} \quad (2.45)$$

If the constitutive relation is given as:

$$\sigma = D\epsilon = DBu_i \quad (2.46)$$

where B is the strain to displacement mapping matrix, D is the constitutive matrix and u_i is the nodal displacement, equation 2.44 provides:

$$\delta_i^{*T} \left[\int_{\Omega} B^T D B u_i d\Omega + \int_{\Omega} N^T b d\Omega + \int_{\Gamma} N^T \tau d\Gamma \right] = 0 \quad (2.47)$$

Since the weighting functions are chosen arbitrarily, equation 2.47 is true for any values of W and therefore:

$$\int_{\Omega} B^T D B u_i d\Omega - \int_{\Omega} N^T b d\Omega - \int_{\Gamma} N^T \tau d\Gamma = 0 \quad (2.48)$$

where $B^T D B = K$ is the stiffness matrix. This reduces to the standard FEM formulation:

$$K u = b \quad (2.49)$$

where u is the nodal displacement vector and B is the force vector written as a sum of the boundary vector $f_b = \int_{\Gamma} N^T \tau d\Gamma$ and the load vector $f_l = \int_{\Omega} N^T b d\Omega$. If sufficient boundary conditions for displacements are prescribed so that rigid body modes are constrained, then the K matrix is not singular and a unique solution can be obtained.

2.3.1 Constitutive models in welding

Modeling the stress field in welds where the temperatures varies from about 2000 deg C to room temperature requires different constitutive models that capture the physics in each different temperature range.

For the temperature range of less than 0.5 of the melting point, the probability of thermally activated dislocation motion is small, the dislocations only move due to the applied stress. The strain rate is low, which points to the rate independent

plasticity constitutive equations. The elasticity tensor, yield strength and hardening modulus characterize the solid at these temperatures.

Between 0.5 and 0.8 of the melting point the dislocation motion becomes important and the rate dependent constitutive equations have to be used. The thermally activated flow of dislocations is the dominant deformation mechanism at low strain rates and high temperatures. This viscoplastic solid behavior is described by the elasticity tensor, viscosity and deformation resistance.

For temperatures above 0.8 of the melting point, a linear viscous model is the appropriate model for both the solid and the liquid, characterized by a viscosity dependent on the diffusion of dislocations in the solid and momentum diffusion in the liquid. The constitutive equations used for each of these models are described at length in [12].

2.4 Solve on a composite mesh

In large welded structures the temperature field varies dramatically from high temperatures changes around the weld pool to basically no change in temperature further than a certain thermal distance (depending on thermal diffusivity and welding speed) characteristic for each weld. For a pure thermal analysis, only a relatively small region near the weld needs to be analyzed and therefore, that part of the structure

where there is no change in temperature during welding can be excluded from the solved domain. In contrast, in a thermal-stress analysis the complete structure being welded is in quasi-static equilibrium and the stresses generated by the welding process travel all through the structure. If the whole welded assembly is not included in the stress analysis, it is difficult to define boundary conditions to account for the parts not included in the domain.

If the stress analysis is performed after (or coupled with) the thermal analysis, composite meshes can be used to accommodate the requirements for both thermal and stress analysis: a fine mesh can be used around the weld to capture the high temperature gradients and the changes in the stress field and a coarse mesh can be used far from the weld to represent the structure being welded and the stress state in the complete assembly. At the transition interface between the fine and the coarse mesh, the temperature or displacement field of the fine mesh has to be constrained to be C^0 continuous (the functions have to be continuous) with that of the coarse mesh. Depending on the position of the fine mesh nodes in the coarse mesh, there are several possibilities for generating the constraints. For the case of an 8 node brick, they are enumerated below:

1. If a fine mesh node coincides with a coarse node, the constraint is simply

$$T_f = T_c$$

for this DOF, where T_f and T_c are the values of the DOF coefficients of the fine

and coarse basis functions for that node.

2. If a fine mesh DOF is in the interior of an edge of a coarse element, then the fine mesh DOF is constrained to be a linear combination of the two coarse mesh DOFs on the ends of this edge:

$$T_f = a_1 T_{1c} + a_2 T_{2c}$$

where a_1 and a_2 are constants that interpolate the coarse mesh DOFs and T_{1c} and T_{2c} are the DOFs of this edge of the coarse mesh.

3. If a fine mesh DOF is not on the edge of a coarse element, but in the interior of the face of the coarse element, then it is a linear function of all DOFs on this face of this element, e.g., for a four node quad:

$$T_f = a_1 T_{1c} + a_2 T_{2c} + a_3 T_{3c} + a_4 T_{4c}$$

where the constants a_i are the values of the basis functions for this face evaluated at the fine mesh node.

4. If a fine mesh DOF is in the interior of the 8 node element, the constraint will be obtained similarly:

$$T_f = a_1 T_{1c} + a_2 T_{2c} + a_3 T_{3c} + a_4 T_{4c} + a_5 T_{5c} + a_6 T_{6c} + a_7 T_{7c} + a_8 T_{8c}$$

As it can be concluded from the constraints defined above, the nodes and element faces on either side of the interface between the fine and the coarse meshes do not need to match. The process of meshing large complex structures is made easier

by using composite meshes because each part can be meshed independently. It is obvious that a composite mesh reduces the cost of the analysis (for the same accuracy of the computation) compared with an analysis that uses the same size elements for the whole computational domain. However, the cost of evaluating the constraints increases the cost of the analysis. For each particular welded structure, depending on the analysis requirements, an optimum use of composite meshes can reduce greatly the cost of the analysis.

Chapter 3

Geometric modeling and meshing issues

In order to perform a thermal, stress and microstructural analysis of a weld, one of the first steps is to model the geometry of the joint, the filler metal added and the weld pool. A mesh that captures the geometry is a good start for obtaining accurate results. Depending on the complexity and the size of the welded structure, the meshing procedure has to be chosen in order to do an analysis with sufficient accuracy in acceptable time.

Since most of the thermal and mechanical gradients are concentrated within a local zone around the weld pool, the rest of the structure can be modeled more coarsely than the local zone. Far from the weld a coarse mesh provides adequate

accuracy and a fine mesh would be wasteful.

In particular for welds performed on large structures, the problem of accommodating different length scales through the structure requires special attention. To describe the heat-affected zone in adequate detail requires a resolution of fractions of a millimeter, while the weld structure could have dimensions of the order of tens of meters.

For long, simple, straight welds, a steady state solution can be applied to reduce the cost of the analysis. In the case of complicated joint geometries and weld paths, a quasi-stationary steady state may not exist. Therefore the computational cost can be greatly reduced by using graded, adaptive or dynamic meshes.

The algorithms presented in this section are intended to be a step toward modeling the exact geometry of all weld components: weld pool, filler metal added and weld joint. The input is the geometric data of a real weld (measured from experiment or designed by the welding engineer) in the form of a weld procedure and a weld path.

3.1 Geometric modeling of multi pass filler metal

An algorithm to generate filler metal geometry of a multi pass weld with emphasis on a pipe tee joint is presented below. Note that the algorithm described can be applied to any joint geometry (butt joint, lap joint, etc.). The required input is a 2D weld

procedure and a 3D description of the welding path. The immediate result is a set of points fully describing the filler metal geometry.

Elements can be created by defining the topology (i.e., element description in terms of its vertices) and consequently a mesh can be generated and used for the weld analysis. This type of meshing belongs to the “manual meshing” category ([7]), with the user responsible for defining:

- the number of elements to be created by specifying for each:
 - its geometrical nature,
 - the list of its vertices in a proper way,
 - the physical attributes associated with the element and its constituent items (faces, edges and vertices)
- the number of vertices and their coordinates.

The mesh created to model the filler metal geometry can be combined with the mesh of the welded structure by juxtaposition to obtain the complete mesh of the welded structure.

3.1.1 The geometry of the joint

The configuration of the parts to be welded determines the type of joint. Some examples of joint types can include butt joint, tee joint, lap joint, corner joint. The type of joint preparation (single or double beveled, single or double U, etc.) will depend on the welding process and the fabrication procedure.

To illustrate the proposed algorithm, a structure of two orthogonal pipes that are to be welded on their intersection path is considered. The axes of the pipes intersect at the point (x_{yc}, y_{xc}, z_{yc}) . One pipe (py) has its axis parallel to the y axis of the global coordinate system and has radius r , the other pipe (px) has its axis parallel to the x axis and has radius $R > r$. The weld preparation can be described as an ordered set of points, in the global coordinate system of the structure. These data give the relative position between the pieces that are to be welded and a complete description of the weld joint, a tee joint in this case.

3.1.2 Define the weld path

In general, the curve describing the welding path can be given (or approximated) as an ordered set of points. For the case of a pipe tee joint, the path that the welding arc will travel around the joint is defined by the curve of the intersection between the two pipes, which can be computed by writing the equation of the cylinder for the

larger diameter pipe:

$$(y - y_{xc})^2 + (z - z_{yc})^2 - R^2 = 0 \quad (3.1)$$

and the parameterized equation of the cylinder for the smaller diameter pipe:

$$x = r \cos(\theta) + x_{yc} \quad (3.2)$$

$$z = r \sin(\theta) + z_{yc} \quad (3.3)$$

Equations 3.1 to 3.3 allow the coordinates of the points on the curve of intersection to be determined, with any desired discretization given by the angle θ .

As a part of the weld joint description, a start and an end of the weld has also to be specified in the global coordinate system of the structure. An example is shown below for a straight weld parallel to global y axis:

```
startPointWeldPath!  0.069   0.0189  0.004 [m]
```

```
endPointWeldPath!    0.069   0.145   0.004 [m]
```

The data about the welding location and the welding path is given by the design specifications, usually in the form of engineering drawings describing the size, orientation and position of the welded structure parts.

3.1.3 Define the weld procedure

For each weld joint, a weld procedure is chosen. The weld pool for each pass has an associated ellipse, with the center (xOrigin, yOrigin) and semi-axis (xSemiAxisFillerMetal, ySemiAxisFillerMetal). The local coordinate system of the ellipses can be rotated relative to the coordinate system of the weld procedure with an angle (angleFillerMetalEllipse).

The ellipse defines the filler metal added by this pass to be the points inside this ellipse that are not inside any previous ellipse or in the weld joint polygons. The weld joint polygons will be defined later. The welds are executed in the order specified in the weld procedure.

The weld procedure specifies for each pass the data needed to determine the filler metal cross-section. The size and the position of each filler metal ellipse is usually determined in the weld design stage depending on the thickness of the parts being welded, the type of joint preparation, the welding process used, etc. An example is given below for a three passes weld:

```
weldType      "fillet"

numberOfPasses  3

xOrigin          0.001      0.005      0.015 [m]
yOrigin          0.007      0.008      0.01  [m]
angleFillerMetalEllipse  0.0      0.0      0.0  [degrees]
```

xSemiAxisFillerMetal	0.008	0.009	0.008 [m]
ySemiAxisFillerMetal	0.005	0.004	0.003 [m]

These parameters of the weld procedure fully determine the cross section of each filler metal pass. The filler metal added by each pass can be computed from the difference of the ellipse of that pass and the part of the weld procedure that has been filled in the previous weld passes.

In order to do this, a geometric description of the joint preparation is given as a set of convex polygons. Each convex polygon is described by a CCW ordered set of points. To obtain a good approximation of the cross section for the weld pass, both the initial boundary defined by the edges and the ellipses for each pass are refined.

For each pass, two boundaries are computed: the internal and the external boundary. The connected set of points that are on the current ellipse and not in or on an earlier weld pass or weld joint define the external boundary. The points on the internal boundary of this weld pass must be on the external boundary that existed before this pass. For a three pass weld, the internal and external boundary are shown in figures 3.1 to 3.3.

Internal Boundary
of pass 1

External Boundary
of pass 1

Figure 3.1: The internal and external boundary of pass 1.

Internal Boundary
of pass 2

External Boundary
of pass 2

Figure 3.2: The internal and external boundary of pass 2.

Internal Boundary
of pass 3

Figure 3.3: The internal and external boundary of pass 3.

Two checks have to be done in order to build these boundaries: first, which points on the current ellipse are inside the previous external boundary and second, which points on the previous external boundary are inside the ellipse.

To check if an ellipse point is inside the previous boundary, the scalar product is

used. To determine if the boundary points are inside the current ellipse, the equation of the ellipse is checked. Also the exact intersection of the ellipse and the previous boundary is computed.

Based on this information, the ellipse and boundary points are flagged. By switching from the boundary to the ellipse at every intersection point, the cross section of the filler metal pass is constructed.

At the end, the cross section of the current filler metal pass (the computed internal boundary) is defined as an CCW set of points. The coordinates of these points are given in the local coordinate system of the procedure. Also the external boundary of the weld metal is stored after this pass.

An illustration of the result of this algorithm is given below for the case of a three pass fillet weld (figure 3.4) and a butt weld executed with eleven passes (figure 3.5).

Figure 3.4: Computed weld procedure for a fillet weld.

Figure 3.5: Computed weld procedure for a butt weld. For some of the passes, the local coordinate system of each ellipse is rotated relative to the weld procedure coordinate system.

3.1.4 Frenet trihedron (tangent, normal, binormal to a curve)

To illustrate the connection between the weld procedure and the weld path, some concepts from the general theory of curves [4] are presented below.

The tangent to a curve C at a point P of C is defined as the limiting position of the straight line L through P and another point Q of C as Q approaches P along the curve (figure 3.6). If C is represented by a continuously differentiable vector function $\mathbf{r}(t)$, where t is any parameter, L has the direction of the vector

$$\frac{\mathbf{r}(t + \Delta t) - \mathbf{r}(t)}{\Delta t} \quad (3.4)$$

where P and Q correspond to t and $t + \Delta t$. Therefore, if the vector

$$\dot{\mathbf{r}} = \lim_{\Delta t \rightarrow 0} \frac{\mathbf{r}(t + \Delta t) - \mathbf{r}(t)}{\Delta t} \quad (3.5)$$

is not zero, it has the direction of the tangent to C at P and it points in the direction of increasing values of t . $\dot{\mathbf{r}}$ is called a *tangent vector* of C at P and the corresponding **unit tangent vector** is

$$\mathbf{u} = \frac{\dot{\mathbf{r}}}{|\dot{\mathbf{r}}|} \quad (3.6)$$

If in particular C is represented by a three times continuously differentiable vector function $\mathbf{r}(s)$, where s is the arc length, the unit tangent vector can be written as

$$\mathbf{u} = \mathbf{r}' = \frac{d\mathbf{r}}{ds} \quad (3.7)$$

Figure 3.6: Tangent to a curve.

The unit vector in the direction of $\mathbf{u}'(s)$ is called the **unit principal normal vector** of C and is defined as

$$\mathbf{p} = \frac{\mathbf{u}'(s)}{|\mathbf{u}'(s)|} = \frac{\mathbf{r}''(s)}{|\mathbf{r}''(s)|} \quad (3.8)$$

and by definition is perpendicular to $\mathbf{u}$. The vector

$$\mathbf{b} = \mathbf{u} \times \mathbf{p} \quad (3.9)$$

is called the **unit binormal vector** of C . It follows that $\mathbf{u}$, $\mathbf{p}$, $\mathbf{b}$ forms a right-handed triple of orthogonal unit vectors called the **trihedron** of C at the point under consideration. The three straight lines through that point in the directions of $\mathbf{u}$, $\mathbf{p}$, $\mathbf{b}$ are called the **tangent**, the **principal normal** and the **binormal** of C .

Figure 3.7 shows also the names of the three planes spanned by each pair of these vectors.

Figure 3.7: Trihedron of a curve.

3.1.5 The connection between the weld procedure and weld path

In each point that defines the weld path, a local coordinate system formed by the tangent, the principal normal and the binormal to the curve of intersection can be defined (as presented in the section above). The plane defined by the principal nor-

mal and the binormal will be the plane of the weld procedure and by definition, the trihedron is always perpendicular to the tangent at the weld path. Figure 4.39 shows the position of these three vectors for a tee joint weld, using the notations $r = \tau_s, s = \tau_p, t = \tau_b$.

For the case of the tee joint the tangent vector τ_s at the point (x_i, y_i, z_i) on the path can be approximated by the central difference equation:

$$\tau_s = \left(\frac{x_{i+1} - x_{i-1}}{ds}, \frac{y_{i+1} - y_{i-1}}{ds}, \frac{z_{i+1} - z_{i-1}}{ds} \right)$$

where

$$ds = \sqrt{(x_{i+1} - x_{i-1})^2 + (y_{i+1} - y_{i-1})^2 + (z_{i+1} - z_{i-1})^2}$$

The weld joint normal vector can be obtained by summing the vector perpendicular to py cylinder:

$$py = (x_i - x_{yc}, 0, z_i - z_{yc})$$

and the vector perpendicular to px cylinder:

$$px = (0, y_i - y_{xc}, z_i - z_{xc})$$

The vector sum $\tau_{p1} = py + px$ is a vector that makes equal angles with the tangent planes to px and py at (x_i, y_i, z_i) and can be made perpendicular to the vector τ_s using Gram-Schmidt rule:

$$\tau_{p2} = \tau_{p1} - (\tau_{p1} \cdot \tau_s) \tau_s$$

and then normalized:

$$\tau_p = \frac{\tau_{p2}}{|\tau_{p2}|}$$

The binormal vector is the result of the cross product $\tau_b = \tau_s \times \tau_p$. It completes the triad that defines a local coordinate system with origin at (x_i, y_i, z_i) and Cartesian axes τ_s, τ_p, τ_b .

Any global point (x, y, z) can be transformed from the global coordinate system of the pipes to a point (r, s, t) in the local coordinate system with the following matrix operation:

$$\begin{pmatrix} x \\ y \\ z \end{pmatrix} = \begin{pmatrix} \tau_{s1} & \tau_{p1} & \tau_{b1} \\ \tau_{s2} & \tau_{p2} & \tau_{b2} \\ \tau_{s3} & \tau_{p3} & \tau_{b3} \end{pmatrix} \begin{pmatrix} r \\ s \\ t \end{pmatrix} + \begin{pmatrix} x_i \\ y_i \\ z_i \end{pmatrix}$$

The inverse transformation is given by:

$$\begin{pmatrix} r \\ s \\ t \end{pmatrix} = \begin{pmatrix} \tau_{s1} & \tau_{p1} & \tau_{b1} \\ \tau_{s2} & \tau_{p2} & \tau_{b2} \\ \tau_{s3} & \tau_{p3} & \tau_{b3} \end{pmatrix}^T \begin{pmatrix} x - x_i \\ y - y_i \\ z - z_i \end{pmatrix}$$

To generate the weld, the weld procedure is swept along the weld path so that at any point along the path, the plane of the procedure is the (τ_p, τ_b) plane and the vector τ_s is tangent to the weld path at that point. At any point on the path, the global coordinates of the points in the weld procedure can be computed using the above transformations.

By specifying the geometry and the topology of the elements/nodes in the weld procedure, a mesh can be generated and used for the analysis. The filler metal elements are brought into the domain of the analysis when the welding source passes over them. The result of this process is shown in figures 3.8 (for the fillet weld) and 3.9 (for a butt weld). Figure 3.8 shows the capability to model properly the given weld geometry by specifying the start and the end of the weld along the weld path.

Figure 3.8: Filler metal passes for a fillet weld.

Figure 3.9: Filler metal passes for a butt weld. The weld procedure is swept along an arbitrary path in space.

3.2 Geometric modeling of the weld pool

If the weld pool geometry (its size, shape and position within the weld) are known or can be estimated from experimental measurements after the weld is performed, the weld pool can be removed from the domain of the energy equation by prescribing its temperature. This approach effectively decouples the arc physics and fluid mechanics of the weld pool from the solid mechanics of arc welds. It allows weld pool data to be input either from experiment or from a previous analysis that predicted the size and the shape of the weld pool. The results of the present thermal analysis proved this model to be very accurate in predicting the size and shape of the heat affected zone and the temperature field distribution. Mesh refinement around the weld pool was found to be essential to obtaining accurate values of the thermal and mechanical fields.

To describe the weld pool shape, a double ellipsoid heat source model proposed by Goldak et al. [17] is used. The size and the shape of the double ellipsoid can be fixed, by recognizing that the solid-liquid interface is well approximated by the melting point isotherm. The semi-axis of the ellipsoid are chosen so its size and shape is that of the weld pool measured experimentally.

Below, an algorithm to model the geometry of the weld pool is presented. Input data are again, the ellipsoid semi-axis and the origin of the weld pool in the global coordinate system of the analyzed welded structure. As it is using elements and

nodes, the algorithm can be applied to a welded structure that has been meshed. Special mesh facilities are employed, but the ideas developed can be applied using any other available tools.

A sphere is deformed to make it coincide with the geometry of the physical weld pool, i.e., the shape and position. Four configurations of the weld pool are defined:

the *reference weld pool* is a sphere embedded in a mesh block (see figure 3.10).

Figure 3.10: The reference weld pool mesh.

The 8 vertex nodes of the *reference weld pool* will define a brick called *weld pool reference brick A*. From every quad on the sphere surface, a chain of elements is built up from the sphere to the external boundary of the mesh. Figure 3.10 shows such a chain. Each chain is treated as a single 8 node brick and called *chain brick* . If the sphere has for example 100 quads formed by the nodes on its surface, 100 chain bricks will result. As a particular observation, any new element created along the algorithm has to be a legal element with respect to the used type of mesh, i.e., to have the same number of nodes and the same topology as the rest of the mesh.

the *replace by weld pool* is a set of elements in the structure being welded containing the actual weld pool and some finite buffer layer (the size of the set can be chosen so that it includes the heat affected zone). The 8 vertex nodes of the *replace by weld pool* define a brick identified as *weld pool reference brick B* such that the node order is consistent with the front, back, left, right, top, bottom of the *reference weld pool*.

the *physical weld pool A* is obtained by deforming the *reference weld pool* such that the physical weld pool A fits the “hole” left by removing the *replace by weld pool* set. It has the correct external geometry and position in the welded structure, but the shape and the position of the weld pool are not correct yet.

Figure 3.11: The physical weld pool A and one of the chain bricks.

the *physical weld pool B* is the final form of the module that will be used in the thermal analysis. It results from deforming the sphere in the *physical weld pool A* to accurately fit the physical weld pool size, shape and position in the welded structure (see figure 4.22).

Figure 3.12: The physical weldpool B.

Two coordinate systems are used: the local coordinate system of the element (r, s, t) with the center in the center of the element and the coordinate system of the mesh (x, y, z) . The main steps of the algorithm are enumerated below:

1. Deform the *reference weld pool* into the *physical weld pool A*.

Given the local coordinates (r, s, t) for each node in the *weld pool reference brick B*, which are the same as in the *weld pool reference brick A*, compute the (x, y, z) coordinates of each node in the *physical weld pool A*. This operation deforms the *reference weld pool* into the *physical weld pool A* so that it now fits the space left by removing the *replace by weld pool*.

2. Deform the *physical weld pool A* into the *physical weld pool B*.

For each node in the *chain brick*, their (r, s, t) coordinates in the *chain brick* are computed. Then the intersection between the melting temperature isosurface and the edges of the *chain brick* is computed in the (x, y, z) coordinate system. The corresponding nodes on the sphere are set to the new coordinates, so that they will be on the melting isosurface. Using their (r, s, t) coordinates in the *chain brick* which is now deformed, the new (x, y, z) coordinates of all interior nodes (lying between the melting isosurface and the external boundary of the mesh) are determined. In this way, the nodes on the boundary of the sphere are moved on the actual weld pool boundary, defined by the melting point isosurface and all other nodes are moved accordingly with their (r, s, t) position

in the deformed chain. The result of this operation is the *physical weld pool B*, which now describes the desired weld pool geometry and position in the welded structure.

3.3 Composite mesh, adaptive mesh

The weld pool module described above can be used as a local mesh for the weld pool. It can be refined independently if greater computational accuracy is required around the weld pool and it can be moved with the welding arc speed. At each time step, at the interface between the weld pool mesh and the rest of the structure a master-slave relationship has to be specified as described in section 2.4. An illustration of a composite mesh used for a weld analysis on a vessel is given in figure 3.13.

Figure 3.13: The weld pool mesh WP1 and the complete welded structure mesh.

Chapter 4

Thermal stress analysis of large welded structures

4.1 Introduction.

A detailed numerical solution of a complex welding problem requires powerful computers and solution algorithms. The model used should be three dimensional, at least in the weld area. The process to be modeled is transient, non-linear and temperature-dependent, with different field gradients depending on position and time because of the rapid heating and cooling in the welding process. The analysis demands a multitude of material characteristic values and their temperature and microstructure state dependence. Any simplification of the above welding process characteristics intro-

duces errors or, in extreme cases, gives misleading results.

The number of possible welding parameters combinations is large in practice. To be of practical interest in the development and improvement of products and in the manufacturing process, the numerical simulation code has to be designed so that it can be applied to a large range of welded structures geometry and welding process variants, i.e., heat input, preheat conditions, welding speed, etc.

The complexity of including all these factors and their interactions has made computational weld mechanics hard to use in analyzing real welded structures. Mainly short single pass welds have been analyzed.

In the design and fabrication process of a welded structure, first a structural engineer designs the structure only specifying the requirements of the welded joints. A welding engineer then designs the weld procedure for each joint to meet the requirements specified by the structural engineer. Next a production engineer specifies the order in which welds will be made and where each pass in each weld joint starts and stops.

The methodology presented in this chapter for simulating the thermal stress response of the welded structures to the applied welding conditions supports distributed design in that the structure to be welded, the weld joints, the weld procedure and the sequence of welding the weld joints can be specified independently by a structural, a welding and a production engineer. The information about the structure to be

welded and the welding parameters is organized so that it can be easily accessed and changed accordingly to desired parameters in different design stages. Depending on the thermal stress field computed with given input parameters, the engineer can try different designs until the best (desired) response is obtained. Until now this process has been based on experience and testing. If the computational model proves to be accurate, the expense of using CWM will be much less than experimental testing.

4.2 Domain, initial and boundary conditions

A major goal of CWM is to analyze the fabrication of complete welded structures with many multipass welds, that is to obtain the transient temperature, displacement, strain and stress at each point in space and time in a weld.

The preferred domain on which the analysis is performed is the entire structure. For the temperature and microstructure calculations, the domain can be reduced to a relatively small region near the weld, without losing the accuracy of the analysis. For the thermal stress analysis though, because it is difficult to choose realistic boundary conditions, it is difficult to perform the analysis accurately on a domain smaller than the whole structure.

In the thermal analysis of a weld, the temperature field results from the effect of the welding heat source in a dimensionally limited structure that had an initial temperature state. It may be room temperature, or a pre-heating temperature specific to the weld process or, in the case of multi-pass welding, the temperature field produced by the preceding passes. For the stress analysis, the initial state of the structure is often assumed to be stress and strain free.

On every face on the boundary of the domain, boundary conditions have to be specified. There can be no node with either more than one boundary condition or none. For the thermal analysis performed in the present chapter, a model with prescribed temperature distribution in the weld pool is used. The temperature of the boundary between the weld pool and the domain is set to the solidus temperature. The weld pool size, shape and position is taken as data (from experimental welds performed with the same welding parameters). The temperature in the weld pool is assumed to vary parabolically from the melting point at the boundary of the weld pool to a maximum temperature of the melting point plus 400 K at the weld pool centroid. For the rest of the domain boundary the exchange of heat is assumed to occur mainly by convection. Depending on the condition of the fluid in contact with the boundary of the structure, specific heat transfer coefficients have to be assigned.

For the thermal stress analysis, rigid body modes were constrained. For each particular welded structure, the boundary conditions are specified depending on the

external reactions (forces) acting upon the structure at the time of welding (e.g., internal pressure on the inside walls of a pipe).

4.3 Computational Results

4.3.1 CWM on large structures

An object oriented approach towards solving industrial welding design issues is presented in this chapter.

A design of a product may be based on a previous design or originated by just considering the loading specifications. Following a previous design that performed satisfactorily offers the advantage of a “safe” starting point, but it also limits the creative thinking needed to develop a totally new approach to solve a given problem. This is a strategy that could prevent more efficient designs. Starting from the loading requirements, the designer can analyze what is needed and devise configurations that best satisfy the need. This is where intelligent computer software is a real help to the design engineer. New ideas can be tried by simulating the mechanical response of the structure. The object oriented tools allow the designers to hide much of the complexity and diversity of the information needed to take into account all the aspects of the problem, and produce usable and useful results.

In the welding domain, the concern is to produce a welding procedure which will lead to a defect free weldment. The problem is to manage the large number of process variables, many of which are interdependent. Incorrect selection of certain welding parameters leads to the failure of the weldment and therefore the decision process requires experienced welding engineers who will usually choose a solution based on previous similar designs.

A proper implementation of computer integrated manufacturing techniques will allow interaction between all of the engineers involved in the welding design and maintenance process. The design and fabrication process for a welded structure generally involves certain steps. Given a certain structure to be executed using welding as an assembly technology, first the mechanical engineer decides on the size, orientation, position and the material of the parts being welded. He also specifies the functional requirements of the weld such as the tensile, bending, shear, buckling, fatigue, etc.

The welding engineer has to design a welding procedure that meets the functional specification of the weld, taking into account also the manufacturability and the cost of the weld. The mechanical properties of a weld are determined by the composition and microstructure of the heat affected zone and weld metal zone. Currently designers use primarily trial and error to find acceptable designs. Using CWM, the actual microstructure can be predicted from a knowledge of the composition and thermal history of the zone. Ideally, the welding engineer should be able to ask for

the value of any material property (e.g., hardness) of the weld to evaluate a weld design.

Once created by the welding engineer and approved by the mechanical engineer, the cost of the materials, equipment and manpower required to perform a weld are evaluated. The cost analysis may require a modification of the weld procedure to meet cost criteria. After the weld design has passed this step, the production engineer must supervise the execution of the weld as specified. His responsibilities include specifying the start and end point of the welds, as well as the sequence of executing the welds. A schedule of materials, equipment and welders must also be specified.

Even though these stages and the responsibilities at each decision level are clearly established in the industry practice, the tools used to improve the design in order to obtain a quality weld are mostly based on rules-of-thumb.

Weld components

The input variables and the requirements of a weld are specified at each design stage. The data defining the weld is organized in the form of the weld procedure and the weld joint. Conventional welding process planning will have the welding variables grouped into categories coming from the sequence of the design steps:

1. Weld structure and joint variables as a part of structural design:
 - material type
 - geometry of each of the parts
 - welding location
 - quality standards
2. Weld procedure variables as a part of welding design
3. Weld production variables as a part of production design:
 - sequence of welds
 - equipment variables
 - welder skill levels
 - consumables in stock
 - contract budgets

Weld procedure selection is one of the key tasks of the welding engineer. For every welding job there is one procedure which will complete the joint at the lowest

cost and best quality. A weld procedure is a 2D specification of the joint preparation including the type of joint (tee, butt, lap, etc.), included angle, root opening and root face. It also specifies the type and size of the electrode, the arc voltage, the arc speed, the type of current (AC, DC), polarity and amperage. An example of a weld procedure is shown in figure 4.1.

SHIELDED METAL-ARC (MANUAL)					
Position: Horizontal Weld Quality Level: Code Steel Weldability: Poor
Weld Size, L (in.)	3/8*	3/8	1/2	5/8	3/4
Plate Thickness (in.)	1/2	1/2	5/8	3/4	1
Pass	1	1 - 2	1 - 2	1 - 3	1 - 4
Electrode Class	E7028	E7028	E7028	E7028	E7028
Size	1/4	7/32	1/4	1/4	1/4
Current (amp) AC	390	335	390	390	390
Arc Speed (in./min)	7.5 - 8.5	11.5 - 12.5	9.0 - 10.0	9.0 - 10.0	8.0 - 9.0
Electrode Req'd (lb/ft)	0.483	0.483	0.819	1.28	1.82
Total Time (hr/ft of weld)	0.0250	0.0333	0.0422	0.633	0.940

Preheat may be necessary depending on plate material.

Figure 4.1: An example of a weld procedure is shown.

The various variables of the weld procedure influence the quality of the resultant weld. For example, the root opening has to be designed large enough to allow good fusion at the root. If it is too small, thinner electrodes have to be used to achieve root fusion which has the effect of slowing down the welding process. If it is designed too large, the weld quality will not be affected, but more weld metal is required which will increase the weld cost and tend to increase distortion. The welding engineer must realize these problems and consider them in the design of the welding procedure. Once a welding procedure has been designed, it is tested, validated and becomes part of the welding standards of each company.

The proposed methodology implements the weld procedure as an abstract class. Each instance of the weld procedure is created by assigning values to the weld procedure variables. The advantage of the object oriented approach is that, once a object or class is created and tested it can be used for a large number of problems. It need not to be changed for each particular application. It can be easily called and used by a designer incrementally changing an old design to create a new design. New methods of the class can be created as they are needed, without changing the structure of the class.

One weld procedure can be used for various weld joints. The weld joint includes a 3D description of the path on which the weld is being deposited which follows the separation plane between the parts being welded. As already shown in sections

3.1.4 and 3.1.5, at any point on the curve describing the welding path, a curvilinear coordinate system can be defined by the tangent, normal and binormal vectors. Figure 4.2 shows the tangent vector r , the normal vector s and the binormal vector t at a point on weld path of a tee weld joint. The plane determined by the vectors (s, t) , normal to the tangent vector, is the plane of the weld procedure.

Figure 4.2: An example of a curvilinear coordinate system for a weld Tee-joint.

Analysis of a welded structure

Given the design parameters as shown above, numerical techniques can be employed to help the welding engineer select the most efficient weld procedure. The present approach uses FEM as a numerical analysis tool.

A computational model is built by defining the geometry of the structure to be analyzed. A mesh is then obtained by approximating the structure with an assemblage of discrete elements. An example of a large structure weld mesh is shown in figure 4.3. A cylindrical pressure vessel with caps and a vertical pipe with a saddle has to be joined by welding in a “hot-tap” welding procedure used to repair a corroded section of a pipeline or to add a branch line. The overall dimensions of the structure are: 3.66 m length, 0.508 m width, 0.950 m height. The welds to be executed are 2 lap circular welds, 2 butt straight welds and 1 tee saddle weld.

A major meshing problem for the large structures is to adequately describe the essential geometric features of the welded structure and at the same time capture sufficient detail around the joint. Figures 4.3 to 4.6 shows the coarse FEM mesh of the structure combined with the finer meshes used for the weld joints. Each of the weld joint meshes was obtained using the parameters specified by the weld procedure of that joint: thickness of plates, edge preparation, filler metal passes size and position. The techniques used to obtain the joint mesh are described at length in chapter 3, Geometric Modeling and Meshing Issues.

Figure 4.3: The meshes created around each weld joint of the welded structure.

Figure 4.5: The mesh for the tee joint.

Figure 4.4: The straight weld mesh.

Figure 4.6: The mesh for the circular joint.

The domain of the CWM analysis contains the coarse mesh of the welded structure and, as each weld is being performed, the fine mesh of the weld joint is added to the analysis domain. The filler metal elements are added to the domain as the weld pool reaches their position in the weld.

The energy equation, the conservation of momentum and the constitutive equations are solved for the analysis domain using FEM algorithms as discussed in chapter 2. Figures 4.7 to 4.9 show the results of the transient thermal analysis of the girth and tee weld.

Figure 4.7: The thermal field as a result of the FEM analysis performed on the girth weld.

Figure 4.8: The first pass thermal analysis for the tee joint.

Figure 4.9: The second pass thermal analysis for the tee joint.

4.3.2 Predicting burn-through when welding on pressurized natural gas pipelines

In making new connections to a natural gas pipeline or when repairing the external damage of the pipeline, the companies wish to avoid the high costs of shutting down a line and depressurizing. Welding on a pressurized pipeline performed with parameters that insure the safety of the welding personnel will bring excellent benefits. The objective of this analysis is to determine if CWM can simulate the process of welding on a pressurized pipeline and provides useful estimates of the risk of burn-through.

The tests were run on a 3.66 m (12 ft.) long pipe, 508 mm (20-in) in diameter and 7.9 mm (0.312 in.) wall thickness API 5LX-65 ERW. The influence of a variety of welding parameters on two thicknesses of pipeline material was investigated. Under certain welding conditions, a pressurized thin wall pipe can perforate because of the high internal pressure and the induced thermal stress field. To simulate this phenomena, called burn-through, the thermal analysis was coupled with the stress analysis, i.e., in each time step the thermal analysis was done on the geometry deformed by the stress analysis.

Methodology

The process of computing the thermal stress field followed these steps:

1. Build an FEM mesh of the pressure vessel as specified in the PRC-185-9515.

The shape and size of the weld reinforcement were carefully modeled to reproduce the profile given in the PRC-185-9515 macrographs. Special meshes had to be created to accommodate the large range in length scales varying from a fraction of a mm near the weld to the entire pressure vessel length. Several levels of grading elements were used, to allow a coarse mesh in the far weld area. This reduces greatly computing costs without losing accuracy.

The mesh used for the pressure vessel, called PV1, had 3006 8-node brick elements. For each weld, a fine mesh with 3456 elements, called MP1, was created for the machined slot region. Outside of MP1, a transition mesh with 768 elements, called TM2, and a coarser transition mesh called TM1 with 140 elements were made. Inside MP1 an even finer mesh was created, called WP1, with 3840 elements. The size of WP1 mesh was chosen to capture the high gradients of the field variables. The mesh WP1 was moved with the arc during the welding process. At the interface between these meshes the temperature and displacement fields were constrained in order to insure solution continuity between the meshes (as presented in section 2.4).

2. For each repair weld, specify the position, current, voltage, arc efficiency, welding speed, weld metal cross-section geometry and weld pool semi-axes lengths.

At each test weld location a flat groove was machined to one of the two different

depths into the pipe (as shown in Table 4.1) and a single weld pass was deposited along the centerline of the machined area. Tables 4.1, 4.2 and 4.3 taken from PRC-185-9515 specify the data for each weld.

Table 4.1 gives a description of the test welds position on the vessel for the burn through potential experiments. The axial distance is measured from one end of the pipe. Figure 4.16 shows, through the thermal field visualization, the relative position of the test welds on the pressure vessel.

Weld Id	Axial distance (in)	Circumferential position	Remaining wall thickness (in)
A	20.0	270	0.125
B	29.0	0	0.125
D	47.0	270	0.125
E	56.0	0	0.125
G	74.0	270	0.156
H	83.0	0	0.156
I	92.0	90	0.156
J	101.0	270	0.156
K	110.0	0	0.156
L	119.0	90	0.156

Table 4.1: The positions of the test welds on the pressure vessel used for the burn through experiments.

The experimental measurements of the resultant weld reinforcement, depth of fusion and HAZ dimensions are shown in Table 4.2.

The welds were made with combinations of different electrode diameters, heat in-

puts, welding amperages and travel speeds. The actual welding parameters used in each weld run are shown in Table 4.3.

This data were organized in the input design file of the solver in the weld procedure section. An example for weld E is given bellow:

```
@weldProcedure5
processType$ "Submerged-Arc Single electrode"
numberOfPasses! 1
xOffset! 0.00 [m]
yOffset! 0.00 [m]
xSemiAxisFillerMetal! 0.00513 [m]
ySemiAxisFillerMetal! 0.00198 [m]
angleFillerMetal! 0.0
depthOfFusion! 0.00091 [m]
heatInput! 881889.7 [J/m]
powerInput! 1679.11 [W]
weldPoolType$ "Double ellipsoid"
#ellipsoid semiaxis: a1, a2, b1, b2, c1, c2
a1weldpool! 0.0055 [m]
a2weldpool! 0.0055 [m]
b1weldpool! 0.01 [m]
b2weldpool! 0.00091 [m]
c1weldpool! 0.005 [m]
c2weldpool! 0.011 [m]
angleWeldpool! 0.0
weldingCurrentPerPass! 80.0 [A]
voltagePerPass! 21.0 [V]
arcEfficiencyPerPass! 0.65
weldingSpeed! 0.001904 [m/s]
@ENDOFSECTION
#
#
@weldJoint5
startPointStraightpath! 0.0 0.25283 -0.4064
endPointStraightpath! 0.0 0.25283 -0.3048
weldProcedureID$ "weldProcedure5"
@ENDOFSECTION
```

Weld	Bead Width (in)	Bead Height (in)	Depth of Fusion (in)	HAZ Width at ID surface (in)	ID Depression (in)
A	0.234	0.053	0.028	0.256	0
B	0.256	0.070	0.025	0.301	0
D	0.306	0.072	0.028	0.265	0
E	0.404	0.078	0.036	0.376	0.025
E-BT	0.00	0.0	0.125	0.501	0.085
G	0.270	0.086	0.025	0.029	0
H	0.390	0.075	0.039	0.404	0
I	0.368	0.075	0.045	0.418	0
J	0.448	0.081	0.036	0.501	0.028
K	0.448	0.095	0.042	0.397	0.039
L	0.513	0.095	0.036	0.446	0.061

Table 4.2: The cross-section dimensions of the test welds.

3. For each weld compute the 3D transient temperature, microstructure, displacement, strain and stress.

The energy equation is solved using 8-node bricks with backward Euler time integration, in a Lagrangian formulation. Temperature dependent thermal conductivity κ [W/msK], specific enthalpy h [J/m^3], volumetric specific latent heat L [J/m^3], specific heat capacity c_p [J/kgK] and density ρ [kg/m^3] were used (see figures 4.10, 4.11).

The heating effect of the arc is described by a Dirichlet boundary condition on nodes in the weld pool. The weld pool size, shape and position is taken as data determined

Test ID	Heat Input (kJ/in)	Electrode Size (in)	Current (amps)	Voltage (volts)	Travel Speed (in/min)
A	10.3	5/64	50	21	6.1
B	15.0	5/64	50	21	4.2
D	21.7	5/64	50	21	2.9
E	22.4	3/32	80	21	4.5
G	20.3	5/64	50	21	3.1
H	20.2	3/32	80	21	5.0
I	21.7	1/8	110	21	6.4
J	26.5	3/32	80	21	3.8
K	27.2	1/8	110	21	5.1
L	33.0	1/8	110	21	4.2

Table 4.3: The actual welding parameters for burn-through potential experiments.

from experiment for each weld. The temperature in the weld pool is assumed to vary parabolically from the melting point at the boundary of the weld pool (1800 K) to a maximum temperature of 2200 K in the center of the weld pool. The size of the weld pool and of the weld reinforcement were estimated from macrographs contained in PRC-185-9515. The weld pool model used is a nonconforming weld pool mesh and therefore, the accuracy of reproducing the weld pool boundary depends on the coarseness of the WP1 mesh.

External boundaries of the structure which are assumed to be in still air, have a convection boundary condition with an ambient temperature of 300 K and convection coefficient of $10 \text{ W/m}^2 \text{ K}$. In the experiment, the welded pipe was filled with water and pressurized to 6.2 MPa. To determine the convection coefficient on the internal

surface of the pipe, several runs were done with a convection coefficient in the range 500 to 5000 $W/m^2 K$. For each weld, for a fixed weld pool length, the thermal flux from the weld pool was computed for this range of convection coefficients on the internal pipe surface. The net input power for the weld, ηVA (arc efficiency η , voltage V , current a) will equal the total thermal load at the nodes in the prescribed temperature field of the weld pool. Since the total heat input is determined only by the temperature field in the elements at the liquid-solid boundary, the equivalence can be obtained by adjusting the weld pool size and the convection coefficient. As the weld pool size was determined from a macrograph, the value of the convection coefficient was varied to obtain the actual heat input data given by the experiment. Assuming a weld efficiency factor η of 0.65, the computed net heat input from the arc was compared to the total heat input for the experimental weld as given in Table 4.3. The convection coefficient that provided best agreement between weld nugget geometry and heat input data taken from PRC-185-9515 was 1500 $W/m^2 K$.

The thermal stress analysis computes an FEM approximate solution to the conservation of momentum and mass. The loads, material properties and geometry are time dependent. The 3D transient nonlinear thermal stress analysis used a viscoplastic material model and 8-node bricks. Temperature dependent Young's modulus, Poisson's ratio, yield strength, hardening modulus and viscosity were used, as shown in the following graphs (4.10 to 4.15).

Figure 4.10: Thermal conductivity vs. temperature.

Figure 4.11: Specific heat vs. temperature.

Figure 4.13: Yield strength vs. temperature.

Figure 4.12: Young's modulus vs. temperature.

Figure 4.14: Poisson's ratio vs. temperature.

Figure 4.15: Viscosity vs. temperature.

Rigid body modes in the pipe were constrained. The internal surface of the pipe was pressurized to 6.2 MPa (900 psig). The transient temperature field of the thermal analysis was applied to each time step. The displacement, strain and stress were computed for 40 time steps for each weld, i.e., the weld moved approximately 2.5

mm (0.1 in) in each time step. The stress analysis was done on the full vessel.

The first time step computed the displacement, strain and stress due to applying the internal pressure. Then the weld pool was positioned in the deformed geometry for each subsequent time step. If a groove formed under the weld, the thermal analysis took into account this thinning of the pipe wall. When the arc was extinguished, the weld was allowed to cool to room temperature. The final time step returned the internal pressure to 1 atm.

The evolution of microstructure is computed using the methodology described in [14] and [15].

4. Post-process results of each weld analysis to visualize the temperature, displacement, strain and stress.

Figure 4.16 shows a general view of the thermal field on the whole pipe as a result of the FEM analysis done consequently on all the welds.

Austenite, ferrite, pearlite and bainite isosurfaces as a result of the microstructure computation for weld E are shown below:

Figure 4.16: Temperature field distribution shown on the entire test pipe.

Figure 4.17: Microstructure analysis results for weld E: figure showing 0.9 and 0.1 fraction austenite isosurfaces.

Figure 4.19: Microstructure analysis results for weld E: figure showing 0.085 and 0.015 fraction pearlite isosurfaces.

Figure 4.18: Microstructure analysis results for weld E: figure showing 0.8 and 0.2 fraction ferrite isosurfaces.

Figure 4.20: Microstructure analysis results for weld E: figure showing 0.9 and 0.1 fraction bainite isosurfaces.

Figure 4.21: Microstructure analysis results for weld E: computed phase fractions along a line parallel to the welding direction.

5. Compare the temperature and deformation computed with CWM with the experimental temperature and deformation as estimated from macrographs in PRC-185-9515.

The following figures show the FEM thermal-stress computation results for the test welds in comparison with the experimental results. The computed distortion profile for all the welds agrees quite well with the experiment, which suggests that the FEM model is accurate.

From a visual standpoint based on the external appearance of the experimental welds, the various cross-sections were placed into three groups relative to their burn-through potential: safe welds - with no evidence of burn-through potential (welds A, B, D, G, H and I), marginal welds - an indication that there may be some risk and a need of absolute process control (welds J and L), and burn-through welds - the pipe wall has failed (weld E).

For the safe welds both experiment and FEM showed essentially no groove formation at all. The marginal welds showed internal surface deformation in both experiment and FEM analysis. The FEM analysis showed a deeper groove than experiment for some of the marginal and burn-through welds. Following are the pictures showing the computed results in comparison with the experimental results.

Figure 4.22: FEM analysis of weld A.

Figure 4.24: FEM analysis of weld B.

Figure 4.23: Experimental results of weld A.

Figure 4.25: Experimental results of weld B.

Figure 4.26: FEM analysis of weld D.

Figure 4.28: FEM analysis of weld E.

Figure 4.27: Experimental results of weld D.

Figure 4.29: Experimental results of weld E.

Figure 4.30: FEM analysis of weld G.

Figure 4.32: FEM analysis of weld H.

Figure 4.31: Experimental results of weld G.

Figure 4.33: Experimental results of weld H.

Figure 4.34: FEM analysis of weld I.

Figure 4.36: FEM analysis of weld J.

Figure 4.35: Experimental results of weld I.

Figure 4.37: Experimental results of weld J.

Figure 4.38: FEM analysis of weld K.

Figure 4.40: FEM analysis of weld L.

Figure 4.39: Experimental results of weld K.

Figure 4.41: Experimental results of weld L.

Discussion of the results

The FEM analyses for burn-through presented in this chapter consider a multitude of factors that influence the thermal stress response of the welded structure: the value of the internal pressure, the pipe diameter and wall thickness, the weld pool geometry and welding process parameters (welding speed, current, voltage), convection coefficient, the area of the internal surface with temperatures above temperatures of 900 C, and the time spent at high temperatures.

The accuracy of the thermal analyses is considered to be limited primarily by the values chosen for the convection coefficient on the internal surface of the pipe and the value of the high temperature viscosity for steel. The combined action of the error in the values used for the convection coefficient on the internal surface of the pipe and the viscosity could lead to the slightly deeper groove shown in the FEM analysis on the marginal and burn-through welds.

The results of the FEM analysis were also compared with the results from PRC-185-9515 obtained using the Batelle model for predicting the potential for burn-through. This model identifies the conditions where a burn-through is possible based on the achievement of an internal surface temperature of 1800 F (980 C, 1255 K) [10] [11]. In Table 4.4 the maximum temperature on the internal wall for the FEM analysis of each weld is given in comparison with the temperature predicted by the Battelle model taken from PR-185-9515.

	Battelle (F)	FEM (F)
A	1454	1565
B	1880	1901
D	2184	2200
E	2252	2414
G	1702	1797
H	1756	1968
I	1755	1855
J	1973	2120
K	1976	2148
L	2196	2362

Table 4.4: Maximum temperatures on the internal pipe wall.

Graphs 4.42 and 4.43 shows the temperature on the internal surface of the pipe. The horizontal line is the Batelle 1800 F line.

In conclusion, the agreement between experiment and FEM analysis for the total displacement of the deformed structure is considered good. The major discrepancy between the FEM and experimental results is that the local deformation is smaller in the experimental case. If all the input data required by the solver (the data used from the experiment, the convection coefficient value, the material properties) can be provided more accurately, the FEM computation could offer a solution closer to the real response of the welded structure.

All FEM analyzes were performed on Pentium III (600 MHz or a 450 MHz CPUs) with 512 Mb RAM. The operating system was Redhat Linux 6.1. Computing times did not exceed 10 hours for any weld. The results of the present analysis show that useful

Figure 4.42: Temperature profile on the inside wall surface for the “safe” welds.

Figure 4.43: Temperature profile on the inside wall surface for the “marginal” welds.

estimates of the risk of burn-through in the process of welding on pressurized thin walled steel pipes can be computed using the FEM methodology described in this chapter.

4.3.3 Solving on a composite mesh using implicit function representation of the weldpool

The performances of using a conforming weldpool mesh integrated in a global composite mesh were studied in this test case. The structure analyzed is the same as that presented in the previous section. The nonconforming mesh WP1 is replaced with a mesh in which the weldpool boundary has the exact shape and size as the real weld. It does not directly depend on the level of refinement of the mesh. The weld pool mesh is parameterized so that weld pool shape can be defined dynamically during the analysis. If the data describing the weld pool geometry is available for each computational step from experiment or from a previous arc physics analysis, the weld pool boundary can be meshed accurately for the starting transient, steady and end regim of the weld. The results of the thermal analysis are greatly improved if the weldpool can be described precisely, especially when, as in the case of the burn-through, a mesh error of half milimeter through the thickness of the pipe can change the verdict of the burn-through. Certainly almost the same accuracy can be obtained with a non-conforming mesh, but the number of elements needed will be enormously bigger. The algorithm used to obtain this mesh has been presented in section 3.2.

Figures 4.44 to 4.47 shows how the weldpool mesh is used in the form of an adaptive mesh. For each time step of the analysis, as the weldpool mesh is moving, the constraints insuring the continuity of temperature or displacement fields across the interface of the WP1 mesh and the MP1 mesh are evaluated.

Figure 4.44: The mesh shown for time step 5 of the analysis.

Figure 4.46: The mesh shown for time step 20 of the analysis.

Figure 4.45: The mesh shown for time step 10 of the analysis.

Figure 4.47: The mesh shown for time step 30 of the analysis.

The effect of different welding speeds and heat transfer coefficients on the thermal field distribution was analyzed. Figure 4.48 to 4.51 compare the shape and size of the 1800 K, 1140 K and 900 K temperature isosurfaces for two different welding speeds (0.7 cm/sec and 7 cm/sec) and two different heat transfer coefficients ($10 \text{ W/m}^2 \text{ K}$ corresponding to still air and $5000 \text{ W/m}^2 \text{ K}$ corresponding to high forced convection with water) assigned on the inner surface of the vessel.

Figure 4.48: The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $10 \text{ W/m}^2 \text{ K}$ and a welding speed of 0.7 cm/s .

Figure 4.50: The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $10 \text{ W/m}^2 \text{ K}$ and a welding speed of 7 cm/s .

Figure 4.49: The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $5000 \text{ W/m}^2 \text{ K}$ and a welding speed of 0.7 cm/s .

Figure 4.51: The 1800 K, 1140 K and 900 K are shown for a heat transfer coefficient of $5000 \text{ W/m}^2 \text{ K}$ and a welding speed of 7 cm/s .

Chapter 5

Conclusions

A methodology for computing the transient temperature, stress and microstructure fields for large welded structures with multiple welds has been developed. The welds can be multi-pass welds.

Special meshing algorithms were created to capture the essential geometric features of the welded structure and each weld joint. Composite meshing and adaptive meshing capabilities have been used to reduce the computational cost of the analysis. The methodology is a significant step towards the capability of a fully automatic analysis of a welded structure, given only the geometry of each part in the global coordinate system, the material type of each part, a curvilinear coordinate system for each weld joint, a weld procedure for each weld joint, the starting time, stopping time and starting point for each weld pass.

The interactions between thermal, mechanical and metallurgical processes present in a weld were included by coupling the thermal, stress and microstructural analysis. The input data for the analysis include geometry, thermal properties, initial temperature, initial microstructure, base metal composition, stress and strains, boundary conditions appropriate to the state of the welded structure.

The thermal analysis computes an FEM approximate solution to the transient 3D energy equation in a Lagrangian formulation. A weld pool model with prescribed temperatures was used to model the welding arc. For each of the weld passes, filler metal addition was added as the weld moved. The model gives the thermal history at every point in the welded structure.

The microstructure evolution in the HAZ was computed based on the thermal history of a point traveling along a flow line, from its upstream origin to its downstream exit. The ordinary differential equations governing phase transformations kinetics were integrated along flow lines.

The results of the thermal analysis were then used for the strain and stress field computation. Rate independent, rate dependent and linear viscous constitutive models were changed as a function of temperature to model the material behavior.

The analyses were validated against experimental results. The accuracy of the computation was limited primarily by the knowledge of input data: nonlinear material properties, convection coefficient, high temperature viscosity.

There are several opportunities for future work including the following. A significant gain in computing speed can be achieved by using a steady state model for the thermal stress analysis. The start and end transient regimes specific to the welding process can be modeled more accurately using data about the weld pool shape and size from a simulation of heat and fluid flow in the weld pool. Distributed computing would allow thermal, microstructure and stress to be computed in parallel using an iterative algorithm.

Bibliography

- [1] Carslaw, H.S., Jaeger, J.C., *Conduction of heat in solids*, Oxford, University Press 1973.
- [2] Ozisik, M.N., *Heat Conduction*, Second Edition, Wiley-Interscience Publication 1993.
- [3] Masubuchi, K., *Analysis of welded structures*, New York, Pergamon Press 1980.
- [4] Kreyszig, E., *Advanced Engineering Mathematics*, Fourth Edition.
- [5] Zienkiewicz, O.C., *The Finite Element Method in Engineering Science*, McGraw-Hill, London, 1971.
- [6] Ertas, A., Jones, J.C., *The Engineering Design Process*, Second Edition, John Wiley and Sons, Inc., 1996.
- [7] Ciarlet, P.G., Lions, J.L., *Handbook of Numerical Analysis*, Volume 4, 1996.
- [8] Goldak, J.A., Bibby, J., Downey, D., Gu, M., *Heat and fluid flow in welds*, Advanced joining technologies, North, T.H., (ed.), London, 1990.

- [9] PRC-185-9515, PRC Project No. August 3, 1995. *Repair of Pipelines by Direct Deposition of Weld Metal - Further Studies.*
- [10] Kiefner, J., *Effects of flowing products on line weldability*, Oil and Gas Journal, pp. 49-54, July 18, 1988.
- [11] Kiefner, J., Fischer, R., *Models aid in pipeline repair welding procedure*, Oil and Gas Journal, pp. 41-547, March 7, 1988.
- [12] Goldak, J.A., Breiguine, V., Dai, N., Zhou, J., *Thermal stress analysis in welds for hot cracking, Welding-induced residual stresses in pressure vessels and piping*, ASME PVP Conference, Montreal, Quebec, Canada, 1996.
- [13] Goldak, J.A., Breiguine, V., Dai, N., Zhou, J., *Thermal Stress Analysis in Welds for Hot Cracking*, Editor H. Cerjak, 3rd Seminar, Numerical Analysis of Weldability, Graz, Austria, 1995.
- [14] Henwood, C., Bibby, M.J., Goldak, J.A., Watt, D.F., *Coupled Transient Heat Transfer-Microstructure Weld Computations (Part B)*, Acta Met., vol. 36, No. 11, 1988.
- [15] Watt, D.F., Coon, L., Bibby, M.J., Goldak, J.A. and Henwood, C., *Modeling Microstructural Development in Weld Heat-Affected Zones, (Part A)*, Acta Met., vol. 36, No. 11, 1988.

- [16] Khoral, P., *Coupling Microstructure to Heat Transfer Computation in Weld Analysis*, M. Eng. thesis, Carleton University, Ottawa, Canada, 1989.
- [17] Goldak, J., Chakravarti, A., Bibby, M.J., *A New Finite Element Model for Welding Heat Sources*, Metallurgical Trans. AIME, 15B, 1984.
- [18] Gu, M., Goldak, J., Bibby, M.J., *Computational Heat Transfer in Welds with Complex Weld Pool Shapes*, Advanced Manufacturing Engineering, Vol. 3, 1991.