1

MICROD

National Library
of Canada

Bibliothèque nationale
du Canada

Canadian Theses Service

Service des thèses canadiennes

Ottawa, Canada
K1A 0N4

NOTICE

The quality of this microform is heavily dependent upon the quality of the original thesis submitted for microfilming. Every effort has been made to ensure the highest quality of reproduction possible.

If pages are missing, contact the university which granted the degree.

Some pages may have indistinct print especially if the original pages were typed with a poor typewriter ribbon or if the university sent us an inferior photocopy.

Previously copyrighted materials (journal articles, published tests, etc.) are not filmed.

Reproduction in full or in part of this microform is governed by the Canadian Copyright Act, R.S.C. 1970, c. C-30.

AVIS

La qualité de cette microforme dépend grandement de la qualité de la thèse soumise au microfilmage. Nous avons tout fait pour assurer une qualité supérieure de reproduction.

S'il manque des pages, veuillez communiquer avec l'université qui a conféré le grade.

La qualité d'impression de certaines pages peut laisser à désirer, surtout si les pages originales ont été dactylographiées à l'aide d'un ruban usé ou si l'université nous a fait parvenir une photocopie de qualité inférieure.

Les documents qui font déjà l'objet d'un droit d'auteur (articles de revue, tests publiés, etc.) ne sont pas microfilmés.

La reproduction, même partielle, de cette microforme est soumise à la Loi canadienne sur le droit d'auteur, SRC 1970, c. C-30.

Thermal-Stress Analysis of Contact Problems by the Finite Element Method,

Lemmy Lesse Meekisho
B.Sc. Eng. (Mech.)

*A Thesis submitted to the Faculty of Graduate
Studies and Research in partial fulfillment
of the requirements for the degree
Doctor of Philosophy*

**DEPARTMENT OF MECHANICAL AND AERONAUTICAL ENGINEERING
CARLETON UNIVERSITY
OTTAWA, ONTARIO
CANADA**

August 1988

©1988

Lemmy Lesse Meekisho

Permission has been granted to the National Library of Canada to microfilm this thesis and to lend or sell copies of the film.

The author (copyright owner) has reserved other publication rights, and neither the thesis nor extensive extracts from it may be printed or otherwise reproduced without his/her written permission.

L'autorisation a été accordée à la Bibliothèque nationale du Canada de microfilmer cette thèse et de prêter ou de vendre des exemplaires du film.

L'auteur (titulaire du droit d'auteur) se réserve les autres droits de publication; ni la thèse ni de longs extraits de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation écrite.

ISBN 0-315-46247-7

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

**Thermal-Stress Analysis of Contact Problems
by the Finite Element Method**

submitted by

Lemmy L. Meekisho, B.Sc. Eng. (Mech.)
in partial fulfillment of the requirements
for the degree of Doctor of Philosophy
in Mechanical Engineering

Chairman, Department of Mechanical
and Aeronautical Engineering

Thesis Supervisor

ABSTRACT.

The presence of internal boundaries or discontinuities is a common feature of many engineering problems both in stress analysis and heat transfer analysis. To analyze such problems by the Finite Element Method, special interface elements are developed to model discontinuities. Since contact is often localized, these elements are of the grading type so as to allow fine meshes in the contact region. To model the interactions between stress and heat transfer, a fully coupled analysis is developed. A computer programme is developed and tested for both 2D and 3D static and dynamic contact, on curved surfaces.

Testing is done by solving simple test cases, based on elasticity theory or heat transfer theory respectively, whose solutions are known or can be verified before attempting practical engineering problems.

The normal stress distribution for a long cylinder on a semi-infinite flat plate compares well with the closed form solution from Hertz's equations. Wheel-rail interaction problems particularly in sliding contact are examined.

Heat transfer across contacts is also modelled. The thermal properties and boundary conditions are specified carefully so as to mimic as accurately as possible the dependence of contact thermal resistance upon interface pressure, surface quality and gap size, as indicated by experimental results.

Practical application of interface elements in heat transfer is examined by modelling heat transfer across contacts and across layers of materials with different thermal properties. Specific problems of heat transfer across contacts and small air gaps in spring-loaded and bolted electronic circuit components are examined including a discussion of possible means of reducing contact thermal resistance based upon interacting geometries. A coupled thermal-stress analysis using contact elements is demonstrated by modelling the peening of welds. The analysis suggests that it is possible to produce welds that are nearly free of residual stress and distortion. This result could be important to industrial applications.

Acknowledgements.

I express my deepest gratitude to my thesis supervisor, Dr. John A. Goldak for his time, invaluable guidance, encouragement and dynamic ideas throughout the research and preparation of this thesis. I am very thankful to Balvant Patel, Alan Oddy, Moyra McDill, Dan Downey and other colleagues and friends at CASCADE and Carleton in general, past and present, whose cooperation and assistance has made my work enjoyable and fruitful. I also express gratitude to the Commonwealth Fellowship and Scholarship Scheme, my supervisor, and the Department of Mechanical and Aeronautical Engineering for supporting me financially. Thanks to my sister Rose whose support and confidence has always helped me a great deal.

Table of Contents

Abstract.	ii
Acknowledgements.	iii
Table of Contents.	iv
List of Figures.	vi
Nomenclature.	x
1. CONTACT STRESS ANALYSIS.	1
1.1 Introduction.	1
1.2 Present Study.	12
2. FINITE ELEMENT ANALYSIS OF CONTACTS.	13
2.1 Choice of Method.	13
2.2 Two-Dimensional Contact Analysis.	15
2.3 Three-Dimensional Contact Analysis.	22
2.4 Finite Element Solution.	28
2.5 Properties of Contact Elements.	31
2.6 Test Problems.	36
3. DYNAMIC ANALYSIS	48
3.1 Introduction	48
3.2 Central Difference Method.	49
3.3 The Newmark Implicit Algorithm.	51
3.4 Dynamic-Contact Analysis	54
3.5 Test Problems.	58
4. CONTACT THERMAL ANALYSIS.	70

4.1	<i>Introduction.</i>	70
4.2	<i>Thermal Contact Elements.</i>	71
4.3	<i>Two Dimensional F.E.A.</i>	74
4.4	<i>Three Dimensional F.E.A.</i>	76
4.5	<i>Test Problems.</i>	78
5.	<i>COUPLED CONTACT ANALYSES.</i>	108
5.1	<i>Thermal-Stress Analysis.</i>	108
5.2	<i>Thermal-Stress Analysis of Contact Problems.</i>	109
5.3	<i>Test Problems.</i>	110
6.	<i>CONCLUSIONS AND CONTRIBUTIONS.</i>	143
6.1	<i>Future Applications of Contact Analysis.</i>	144
	<i>REFERENCES.</i>	146
	<i>APPENDIX A.</i>	155
	<i>APPENDIX B.</i>	159

List of Figures.

Figure	Page
1.1 Stress boundary conditions.	3
2.1 Nonlinear boundary conditions in contact problems.	14
2.2 Topology of 2D contact elements.	16
2.3 Shape functions for 2D contact elements.	18
2.4 Topology of 3D contact elements (a, b).	23
2.4 Topology of 3D contact elements (c, d).	24
2.5 Bilinear material properties for contact elements.	33
2.6 Nonlinear material properties for contact elements.	34
2.7 Model for nonlinear properties of contact elements.	35
2.8 Model for Hertz's cylinder/plate interaction.	37
2.9 Simple test problem for analytical verification of contact algorithm.	38
2.10 Coarse F.E. mesh for Hertz's cylinder/plate problem.	43
2.11 Deformed shape of Figure 2.10	44
2.12 Graded F.E. mesh of a portion in Fig.2.10 near the contact zone.	45
2.13 Comparison of F.E. results with Hertz's solution.	46
2.14 3D graded mesh with 8-16 node contact bricks for testing algorithm.	47
3.1 A one element dynamics test case, constant x-acceleration.	60
3.2 A one element dynamics test case, constant x- and y-acceleration.	61
3.3 A multi-element dynamics test case with 4-8 node quadrilaterals.	62

3.4	Subelements (integration domains) from figure 3.3.	63
3.5	F.E. mesh for a simple wheel-rail interaction problem.	64
3.6	Free-body sliding of wheel over rail simulating braking conditions.	65
3.7	'Sticking' conditions in the interface of wheel-rail problem.	66
3.8	F.E. mesh for a simple test problem of rolling mill simulation.	67
3.9	'Frictionless' conditions in contact interfaces of the rolling mill problem.	68
3.10	'Sticking' conditions in contact interfaces of the rolling mill problem.	69
4.1	Variation of contact conductance with interface pressure.	73
4.2	A three element contact thermal analysis test case with gap size = 1	82
4.3	A three element contact thermal analysis test case with gap size = .1	83
4.4	A three element contact thermal analysis test case with gap size = .01	84
4.5	F.E. mesh for an initial temperature test problem with an air gap.	85
4.6	Temperature contours for Fig. 4.5 over 4 time steps.	86
4.7	Temperature contours for Fig. 4.5, heat source moved after each step.	87
4.8	Coarse F.E. mesh for a molten drop test problem.	88
4.9	Temperature contours for Fig. 4.8 after 0.1 seconds	89
4.10	Finer F.E. mesh for a molten drop test problem.	90
4.11	Temperature contours for Fig. 4.10 after 0.1 seconds.	91
4.12	F.E. mesh for Pammer's problem.	92
4.13	Temperature contours for Pammer's problem, low conductance in the gap.	93
4.14	Temperature contours for Pammer's problem, high conductance in the gap.	94
4.15	F.E. mesh for electronic clip problem, thermal boundary conditions.	95
4.16	Temperature contours in the clip, air in interface, force = 57 grammes.	96

4.17	Temperature contours in heat sink, air in interface, force = 57 grammes.	97
4.18	Temperature contours in the clip, He in interface, force = 114 grammes.	98
4.19	Temperature contours in heat sink, He in interface, force = 114 grammes.	99
4.20	Temperature change across gap versus interface force for Fig. 4.15.	100
4.21	Temp. change/Watt across gap versus interface force for Fig. 4.15.	101
4.22	Dimensions and layout of the transistor problem.	102
4.23	F.E. mesh for the transistor problem, stress boundary conditions	103
4.24	F.E. mesh for the transistor problem, thermal boundary conditions	104
4.25	Temp. contours in heat sink for transistor problem, force = 3.648 N	105
4.26	Temp. contours in heat sink for transistor problem, force = 29.148 N	106
4.27	Temp. change/Watt versus interface force for the transistor problem	107
5.1	Thermal boundary conditions for simple coupled contact analysis test	114
5.2	Undeformed & free expansion of Fig. 5.1, high interface shear resistance	115
5.3	Undeformed & free expansion of Fig. 5.1, low interface shear resistance	116
5.4	F.E. mesh of cylinder/plate test in coupled contact analysis.	117
5.5	Temperature contours for the cylinder/plate problem.	118
5.6	End view of temperature contours for Fig. 5.4.	119
5.7	End view of yy-stresses (normal to the gap) for Fig. 5.4.	120
5.8	End view of zz-stresses (cross-sectional) for Fig. 5.4.	121
5.9	zx-stresses (shear stresses in the plane of plate) for Fig. 5.4	122
5.10	F.E. mesh of cylinder/cylinder test in coupled contact analysis.	123
5.11	Temperature contours for the cylinder/cylinder problem	124
5.12	End view of temperature contours for Fig. 5.10.	125

5.13	End view of yy-stresses (normal to the gap) for Fig. 5.10.	126
5.14	End view of zz-stresses (cross-sectional) for Fig. 5.10.	127
5.15	zx-stresses (shear in plane tangent to cylinders) for Fig. 5.10.	128
5.16	F.E. mesh for a sphere/plate test in coupled contact analysis.	129
5.17	Temperature contours for the sphere/plate problem.	130
5.18	End view of temperature contours for the sphere/plate problem.	131
5.19	xy-stresses (shear in the plane of the plate) for Fig. 5.16.	132
5.20	F.E. mesh for a $4 \times 2 \times 1/4$ inch plate for edge- and butt-weld analysis.	133
5.21	zz-stresses for Fig. 5.20, after weld pass, & cooled to R.T.	134
5.22	xy-stresses for Fig. 5.20, after weld pass, & cooled to R.T.	135
5.23	zz-stresses for Fig. 5.20, after weld & peening at R.T.	136
5.24	xy-stresses for Fig. 5.20, after weld & peening at R.T.	137
5.25	zz-stresses for Fig. 5.20, after weld & cooling to $100 - 150^{\circ}C$	138
5.26	xy-stresses for Fig. 5.20 after weld & cooling to $100 - 150^{\circ}C$	139
5.27	F.E. mesh for butt-weld analysis showing strip used for peening.	140
5.28	zz-stresses for Fig. 5.27, after weld & peening at $100 - 150^{\circ}C$	141
5.29	xy-stresses for Fig. 5.27, after weld & peening at $100 - 150^{\circ}C$	142
A.1	Figure for demonstrating numerical ill-conditioning in contact analysis.	156

NOMENCLATURE.

σ, τ	Stress
F	Nodal forces
A	Cross sectional area
μ	Coefficient of friction
N, T	Normal and tangential load components at the interface
w	width of contact
d_i	diameters of circular sections
E	Young's modulus
ξ, η	Element level coordinates
N_i	Shape functions
Δu_n	Relative displacement normal to the gap
Δu_t	Relative displacement tangent to the gap
$\Delta \delta$	Relative displacement vector
ϵ	Global relative displacement vector for contact
ϵ_e	Element relative displacement vector for contact
T	Transformation matrix
Q	Determinant of the Jacobian transformation
K_t, K_n	Tangential and normal stiffness per unit length in 2D
K_x, K_y, K_n	Tangential, normal stiffness per unit area in 3D
G	Matrix for transforming absolute displacements into relative displacements
C	Force-relative displacement relationship matrix for contact analysis
D	Matrix of elastic constants
B	Shape function matrix
δ	The variational operator
Π	Total potential energy

- dL, dV Elemental length, elemental volume in numerical integration
 $NGAUS$ Number of Gauss points
 W_i Gauss point weights
 J Jacobian transformation matrix
 K The stiffness matrix
 R, P Applied external nodal load
 Ψ Residual load
 λ Fraction of applied external load
 $\ddot{X}$ Nodal acceleration vector
 $\dot{X}$ Nodal velocity vector
 X Nodal displacement vector
 Δt Time increment (step)
 M Mass matrix
 C Damping matrix
 α, δ Integration constants
 α_c Proportionality constant in lumped mass matrix calculations.
 δ_{ij} Kronecker delta
 A_i Integration parameters
 θ, β Scaling constants on M and K for computing damping matrix C
 ρ Density
 M_l Nodal mass for the master line or surface
 M_l' Nodal mass for master line or surface plus distributed slave mass
 r Mass factor
 Θ, T Nodal temperatures
 ΔT Temperature change
 k_x, k_y, k_z Thermal conductivities in the x-, y- and z-directions

Q	Heat source
c_p	Specific heat capacity
t	time
∇	The gradient operator
J	Thermal flux
C	Thermal load per unit temperature
h_j, h_c, h_g, h_r	Joint, contact, gas, radiative conductances respectively
p	pressure in the interface
α_{th}	coefficient of thermal expansion
ϵ^e	Elastic strain
ϵ^{pl}	Plastic strain
ϵ^{th}	Thermal strain
ϵ^{Tot}	Total strain
T^i	Surface tractions
n^i	Normal vector at the interface
D_{ijkl}	Elastic material tensor
CASCADE	Centre for Advanced Studies in Computer Aided Design and Engineering (Carleton).

NOTE: Some of the symbols were used to define more than one variable or function, however, every effort was made to define concisely the meaning of the symbols as they appear in different sections of the text.

1. CONTACT STRESS ANALYSIS

1.1 Introduction.

Internal boundaries are commonly encountered in components mechanically coupled by processes like welding, riveting, nailing, gluing, screws, nuts and bolts, etc. Uncoupled contact is also commonly encountered, for instance in machine components like ball and roller bearings, wheel-rail interaction, cam-follower interfaces, etc. Both coupled and uncoupled contact involve the transmission of loads from one component to another via contact interfaces. In electronic circuit components thermal loads are transmitted across contact interfaces. This presents a challenge in determining the proper service performance by load transmission between the components via points, lines or surfaces of contact. Contact stresses are now often incorporated into the design process of components that are expected to operate under static or dynamic contact with other solid components in service, the purpose being to avoid premature component failure due to high localized contact stresses which are often not included in the standard strength of material equations. In the simplest theory, contact between bodies with curved surfaces takes place at discrete points or lines. When pressed by forces large enough to cause finite elastic deflection or plastic yielding, the geometry of contact changes to finite areas. The contact areas so formed are normally much smaller than the surface areas of the individual bodies thus contact stresses can be extremely high.

To begin, recall the definition of stress,

$$F/A = \sigma \cdot n \quad (1.1)$$

Where σ is the stress, F , n and A the force, outward normal and cross sectional area respectively. In most problems A is specified in the data, however, in the case of generalized contact, A is usually unknown. In fact the crucial issue in contact analysis is to determine the shape and size of the contact surfaces.

Problems in mechanics usually require investigating the response of the interior of a body to a known set of forces or displacements acting on the boundary. The procedure for solving such problems involves setting down the known information in the form of

boundary conditions and then using differential equations to extend it to the interior of the body. An example by Fung^[1] of two blocks of materials of different properties, as shown in figure 1.1 will be used to demonstrate this procedure.

Equilibrium in the neighbourhood of a point P at the interface requires that the traction acting on the surface must be the same on both sides of the interface. Referring to free body diagrams in figure 1.1b and 1.1c this may be stated mathematically as:

$$T^{(1)} = T^{(2)} \quad (1.2)$$

Surface traction is defined as

$$T_i = \sigma_{ij} n_j \quad (1.3)$$

where T_i is the traction vector, σ_{ij} is the stress tensor in the material, and n_j is the outward normal to the surface.

For the model shown in figure 1.1, in order for the conditions of equation 1.3 to hold, stress components normal to the interface must be continuous across the interface. Hence $\sigma_{xy}^{(1)} = \sigma_{xy}^{(2)}$, and $\sigma_{yy}^{(1)} = \sigma_{yy}^{(2)}$. The other stress components need not be continuous across the interface. They are completely defined within each body as indicated by magnitudes of σ_{xx} in the two materials in figure 1.1d. The contact problem involves computing the geometry of the contact surface and the value of surface tractions T that enforce the value of σ near the surface.

1.1.1 First Generation of Contact Analysts.

The first documented successful analytical attempt at solving contact problems was by Hertz, in 1881. His model was based on two spheres in contact with each other, the region of contact being a circle whose diameter is dependent upon the load acting on them. This model was extended to one of a cylinder and plate in contact in which case the region of contact was approximated by a 'thin' rectangle whose width w was computed from the following relationship by Shigley^[2]:

$$w = \sqrt{\frac{2F}{\pi l} \frac{[(1 - \mu_1^2)/E_1] + [(1 - \mu_2^2)/E_2]}{(1/d_1 + 1/d_2)}} \quad (1.4)$$

FIG. 1.1: STRESS BOUNDARY CONDITIONS.

where E_1, E_2 are the Young's moduli for the cylinders.

μ_1, μ_2 are corresponding Poisson's ratios.

l is the length of the cylinders.

d_1, d_2 are the diameters of the cylinders.

Using the results of equation 1.4 the peak normal stress at the interface was computed as:

$$\sigma_n = \frac{2F}{\pi w l} \quad (1.5)$$

where σ_n is the peak normal stress and F is the total force acting on the cylinders, which can be the sum of the dead weight of the top cylinder and the applied load. Stress variation in the contact patch is elliptical with a maximum value along the line where the two cylinders have a common tangent, to zero where contact breaks off.

Castleberry^[3] and Seely^[4] both successfully pointed out the shortcomings of using Hertz's formulas alone in design calculations for machine components under contact stresses. This was based on laboratory studies in which a number of failures were attributed to states of stress and strain that were not adequately computed by Hertz's formulas.

Castleberry^[3] recommends that the use of the rather obscure Smith-Liu equations of 1953 for solving problems with line contact can be a feasible extension of Hertz's equations. Thirty five years ago these equations were unattractively complicated, but with the advent of powerful desk-top and pocket computers they can be used to compute the principle stresses σ_{xx}, σ_{yy} and σ_{zz} , and the maximum shear stress τ_{yx} . With σ_{xx}, σ_{yy} and σ_{zz} and τ_{yx} and the use of Mohr's circle it is possible to calculate critical stresses at and in the neighbourhood of contact.

Seely^[4] points out the added degree of complexity of contact introduced by cyclic loading. Fatigue failures are common in machine components in contact because often a localized discontinuity associated with localized contact stresses develops and leads to failure due to many repeated loading and unloading cycles. An example would be the transverse fissure failure of a rail wheel. This starts with a tiny transverse crack a few millimetres below the surface of the wheel. As the wheel rotates the crack is subjected to a cyclic load with maximum normal and shear stress components where the centreline of

the crack is normal to the axis of the rail, and diminishes as contact ceases at this point. The crack grows in an outward direction until finally, if not detected, the whole wheel fractures. Other examples would include fatigue failure which starts with the formation of pits at the surfaces of contact of gear teeth, ball and roller bearings, and fatigue failure which starts with the formation of microscopic cracks at the surface of tappets at the points of contact with cams.

Elegant descriptions of crack propagation mechanisms, fatigue and fracture mechanics are beyond the scope of this thesis. Details are readily available in publications^[5] and in standard design, solid mechanics and fracture mechanics textbooks.^{[2][4][6][7]}

1.1.2 Second Generation of Contact Analysts.

A second generation of researchers extended the long standing theories and equations of Hertz for analyzing contact problems. In the 1960's and 1970's J.J. Kalker^{[8][9]} published a number of papers based on a closed form integral approach for solving a wide variety of contact problems, in particular, rolling contact. The power of Kalker's approach (which is strongly mathematical and was coupled with an equally elegant treatment of the effects of frictional boundary conditions) is reflected in the accuracy of results he obtained by solving rolling contact problems of the type encountered in the railway industry. He developed a computer package for the analysis of fatigue and wear mechanisms in wheel-rail interactions which has been used by railway companies internationally.^[10]

Paul^[11] deviated from the classical Hertzian approach of 'antiformal' contact, in which the contact patch is assumed to be locally ellipsoidal, to one in which bodies hug each other more closely in a 'conformal' fashion, making the contact patch take a more arbitrary geometric shape. Examples of 'antiformal' contact would be the interface of new wheel-rail interface under elastic normal loading in dry friction conditions, new bearings and new cam-follower interface also under dry friction conditions. For 'conformal' contact a well worn wheel-rail interface or an underinflated tire would serve as examples. In a 1982 publication^[12] which summarizes his earlier work, Paul gives an elaborate geometric-trigonometric analysis of patches and shapes which lead to a more flexible approach to solving contact problems. Computer programmes associated with Paul's work like INSEP, CONFORM and INTERPEN are available in the public domain.

1.1.3 Third Generation of Contact Analysts.

The ability to model adequately contact problems by the first and second generation of Contact Analysts was evidently limited to contacting objects of geometries which can be described with simple mathematical equations. The generalized contact problem may involve arbitrary geometries. The contact area is usually unknown. As the objects are forced together the contact area changes continuously resulting in nonlinear boundary conditions. This is the phenomenon that makes the contact problem difficult to solve and requires special solution methods.

In the past, uncertainties caused by variations of geometric shapes in contacting structures and machine components led to the use of high safety factors both in design and fabrication to prevent failure by premature fatigue. This rule of thumb, though wasteful, worked when the safety factors were high enough, however when estimates of safety factors were inadequate, catastrophic failures could occur.

The solution to contact problems of the most general nature in terms of the type of loads, boundary conditions and interacting geometries, be it points, lines or surfaces, evolved in Civil Engineering practice. Many structures in Civil Engineering are assumed to a first approximation to be rigidly interconnected but in reality they are not. The service performance of structures like composite steel/concrete beams, rock joints, layered wooden structures joined by nails, etc. depend to a large extent upon the ability of the interface to transmit normal and tangential loads. Many examples can be cited from other fields, where interface conditions are crucial in service performance. In Mechanical Engineering, units like ball and roller bearings, cams and followers, wheel-rail interfaces, clutches, gears, etc. are but a few.

Numerical procedures such as the Finite Element Method, are the versatile tools that have been used by the third generation of Contact Analysts for solving the generalized problem of contact. Unfortunately these methods have often been plagued by numerical ill-conditioning caused by the rather high stiffnesses associated with the interfaces compared to the surrounding continuum. A demonstration by Wilson^[40] of such numerical ill-conditioning is given in Appendix A. The choice of interface elements with appropriate

behaviour can ease this problem. The interface elements ultimately chosen and used in this thesis will be described in detail later.

The first two-dimensional model with a linear variation of slip between interacting surfaces was proposed by Goodman et al^[13] in 1968. In the following year, Ghaboussi et al^[14] developed a more advanced interface element with a wider range of property values. With this more versatile element it was possible to solve significant problems such as determining the distribution of vertical stresses around embedded circular foundations in soils, and modelling the splitting of rocks using wedge shaped tools. Buragohain and Shah^[15] in 1977, developed curved isoparametric line and axisymmetric elements for the analysis of physical problems with straight or curved contacting surfaces using two dimensional models. In the following year the same authors^[16] extended their 2D model to 3D.

In the 1970's and 1980's the numerical analysis of contact problems progressed rapidly. Many papers were published with emphasis on different aspects of the large family of contact problems. The following is a brief outline of some of these categories.

There were those that were interested in the effects of contacts on service performance of structures and machine components by wear mechanisms and wear resistance.^{[17][18][19][20][21]} Others were interested in assessing the damage caused by abrasion and indentation of different materials in contact, or the same material with different hardness values.^{[22][23][24]}

A number of publications dealt with attempts to circumvent numerical difficulties often encountered when solving contact problems. Specific examples include the presence of nonlinearities, for example, in concrete cracking, crack tip analysis, etc. This category also includes the development of algorithms and other procedures for reducing the cost of computing:

- Zienkiewicz and Francavilla^[25], and Wilcox^[26] used a form of substructuring in which they computed flexibility matrices for the nodes of two elastic bodies coming into contact. This was done by first computing the individual element stiffness matrices, then in the frontal solver, the assembly of the global stiffness matrix and forward elimination is done

but variables associated with the contact nodes are not eliminated. The resulting global stiffness matrix (the upper triangular portion), is condensed to remove the zeros corresponding to the eliminated variables and inverted to obtain the flexibility matrix. The flexibility matrix is passed to a local coordinate system from which nodal displacements tangential to the line or surface of contact are readily computed without the need for back-substitution. This reduced the severity of the nonlinearity of the problem as well as the CPU time.

- The use of differential displacements by Wilson and Parsons^[27], and the use of relative displacements across the contact interfaces rather than absolute nodal displacements by Buragohain and Shah^{[15][16]} and Ghaboussi et al^[14] improved the numerical conditioning.
- It has been shown^[28] that the cost of computing in 3D for a uniform mesh with a direct solver is of $O(n^7)$ and drops to $O(\log_2 n)$ with the use of graded meshes, where n is the number of nodes along an edge of the uniform mesh. Since contact is often a localized phenomenon, graded contact elements were developed and used in this thesis. With graded meshes it is possible to discretize the contact zone using a very fine mesh and a coarse mesh away from the contact zone, an exercise which considerably reduces the cost of computing without compromising accuracy.
- The use of substructuring for computational efficiency without the penalty of loss of generality or accuracy has been discussed by Bathe^[29], and Noor et al.^[30] In substructuring, elements in a mesh being analyzed are numbered strategically such that elements representing the most likely source of numerical ill-conditioning in the problem (e.g. elements closest to the crack tip, or generally the smallest stiffest elements) sum their stiffness contribution to the lower right hand corner of the upper triangular global stiffness matrix. A normal forward elimination is done for the whole global stiffness matrix, but the back substitution phase is limited to the sub-structure. Here an iterative procedure may be performed on this small portion of the matrix until a specified tolerance is reached. After this, the rest of the back substitution is performed (often for this remaining portion the solution can be obtained in one step).
- Benson and Hallquist^[31] proposed a method of combining structural dynamics with rigid body dynamics in a Finite Element analysis of impact problems. In impact situations only

small areas of the bodies undergo finite deformation in the immediate neighbourhood of their contact. The rest serve to carry the momentum for dissipation by plastic work. They proposed that the areas of the bodies outside the impact zone should be modelled as a rigid body, while the impact zone is modelled using deformable body dynamics. The coupling between the two zones is made possible via sliding interfaces a phenomenon discussed in detail in NIKE2D manuals and in an earlier publication by Hallquist.^[33] For some problems, the loss of accuracy by using rigid body dynamics on a portion of the problem and deformable body dynamics in another portion is minimal, but the savings in the computation costs are quite significant.

Attempts to solve a wide variety of contact problems mainly excluding frictional conditions were documented in a number of publications.^{[34][35][36][37][38][39]} Contact problems closer to Civil Engineering in the areas of foundations, soil mechanics and Geotechnical Engineering have been examined.^{[13][40][41][42]} Kalker^{[8][9]} did extensive studies in rolling contact with frictional conditions. Other publications^{[43][44][45]} also document studies of contact problems with various frictional conditions.

Hughes^{[46][47]} studied contact-impact problems. By including inertia and damping terms to the equation of equilibrium along with a number of integration parameters associated with the Central Difference Method, Bathe^[29] suggested algorithms for solving the general dynamics problem. In this thesis production processes like rolling were mimicked by modelling the workpiece as a sliding rigid body having contact interfaces with the rolls, which were treated as static bodies. The Newmark implicit algorithm (one of the Direct Integration Methods) was used for the dynamics. Details and the step-by-step development of the computer programme for implementing the Finite Element Analysis of the Newmark Algorithm are given in section 3.3 of the thesis. Other algorithms for computing the dynamic response of structures by the Finite Element Method include^{[26][48][49][50][51][52][53]}

Oden^[54] gives a detailed description of the Finite Element approach to a class of contact problems generally referred to as Signorini's problem. The solid is usually deformable and is supported on one side by a rigid body. Oden^[54] also addresses the common problem in contact analysis, of the contact surface being unknown *a priori*. He does this by defining

three different regions of contact. These include contact between the deformable solid and rigid foundation, contact at places where the body folds over itself and contact at internal boundaries such as cracks. He discusses three types of Finite Element formulations, namely the displacement, mixed and penalty formulations.

In summary, contact analysis has been studied, in a wide variety of aspects in the past three decades. It would probably be fair to observe that the most elaborate Computer Aided Analysis of contact problems in their most general form has been done at Lawrence Livermore National Laboratories by Hallquist et al [31][33][55][56][57][58]. These publications along with associated Finite Element programmes like NIKE2D and DYNA3D, and accompanying graphics software like INGRID, TOPAZ, TAURUS are available in the public domain. The major disadvantage with the Livermore software is that they were written for supercomputers. This poses some difficulties for small to medium sized research units.

In this thesis a nonlinear algorithm based on the penalty method for modelling nonlinear contact element behaviour is developed. It will be discussed in detail in chapter 2 of the thesis. However, nonlinearity in the continuum part of contact analysis was done using standard nonlinear algorithms like Newton-Raphson, successive approximations, line search, etc. which are described elsewhere. [29][59][60][61]

1.1.4 Friction.

Friction is reviewed briefly since it greatly influences contact analysis. In Civil Engineering structures like foundations which are often under static loads, there is seldom sufficient relative movement between contacting surfaces to warrant an elegant treatment of friction boundary conditions. The laws of friction as described by the long standing formulas of Coulomb are used in these cases. In machine components which are under dynamic loads however, frictional conditions have to be treated very carefully according to the nature of the contact. In the case of gears, cams, pedestal bearings, etc. friction and wear are minimized by the use of appropriate lubricants. In the case of wheel-rail interfaces, belt drives, clutches, etc. considerably large frictional forces are needed for normal operation. Ball and roller bearings are more intricate because lubrication is essential between the balls or rollers and the inner and outer races. In addition, frictional interfaces are necessary between the outer race and housing, and between the inner race and the

shaft. Quite clearly the coefficient of friction in contact situations is very important.

The classical understanding of friction first published by Coulomb in 1785 has over the years been modified by a number of tribologists. A concise summary of the modifications leading to a state-of-art definition and understanding of friction is given by Tabor.^[62] Coulomb's law of friction may be represented by the following relationship:

$$T = \mu N \quad (1.6)$$

where μ is the coefficient of friction and N and T are the normal and tangential components of force for the two bodies in contact. In the current understanding of dry friction, N and T have to be modified quite considerably by:

1. The true area of contact between sliding surfaces
2. The nature and strength of the bonds formed at the interface where contact occurs.
3. The manner in which the materials in and around the contacting regions are deformed, sheared or ruptured in the process of sliding.

1.1.5 Friction Modelling in Production Processes.

In production processes like forging, reducing friction is beneficial since it reduces the forces and energy required to perform a given operation. The result of reduced friction is reduced stresses imposed on dies and may allow the use of smaller hammers or presses for a given part. In some situations it becomes necessary to control the magnitude of friction in order to control material flow hence promote die filling and reduce workpiece spreading. The right magnitude of friction or lubrication is thus evidently necessary in successfully forming the end product and indeed in achieving good quality of the end product.

To attain optimum friction levels the lubricating system must be carefully specified and controlled. Low friction levels help to promote homogenous deformation and hence increase the effective workability of a material and improve product quality by reducing the incidence of residual stresses and internal boundaries like porosity. The life of dies and hammers may also be extended considerably by the presence of a lubricant film which also acts as an insulating shield between the hot workpiece and the die.

There are four lubrication regimes that can occur in production processes like forging,

namely thin film, thick film, mixed and boundary regimes.^[63] These classifications are functions of:

1. Lubricant film thickness.
2. Surface roughness.
3. Load sharing between the bulk lubricant in surface valleys and boundary films at roughness peaks.

Different mechanisms characterize friction in each regime and these must be considered when developing appropriate friction models for different operations

In processes where friction variation is very small, simple friction models like the Coulomb constant friction coefficient are used to investigate the influence of friction on stresses. However, in cases where friction variation is significant it becomes necessary to develop more realistic models. This mainly involves determining the local lubrication regime and understanding the process of lubricant film formation and transportation in the workpiece-tool contact interface.

1.2 Present Study.

The present study is concerned with the development of Finite Element Analytical methods for solving contact problems. In chapter 2 a contact stress algorithm based on the penalty method will be developed and tested. The method proposed will be sufficiently general to handle arbitrary shaped contacting bodies. In chapter 3 an implicit dynamic algorithm will be implemented. This, interfaced with the contact algorithm, will be used to analyze dynamic-contact problems such as those found in production processes like rolling. Chapter 4 will be devoted to transient contact thermal analysis. The heat transfer model will be limited to conduction and will be used to analyze problems of heat transfer across contacts and small air gaps. Industrial application of contact thermal analysis will be demonstrated by solving problems from local electronics industries. A fully coupled contact thermal-stress analysis will be done in chapter 5. This will be used to model the peening of welds. Conclusions and suggestions will be covered in chapter 6.

2. FINITE ELEMENT ANALYSIS OF CONTACTS.

Hertz's problem of a cylinder/plate interaction (figure 2.1a) shows changes in the contact zone with loading. Prior to loading contact between the cylinder and plate is a line through point P . With increasing load the contact zone changes to rectangular surfaces of width w_2 and w_3 . The problem of a stiff punch with a sharp corner in contact with a softer deformable workpiece (figure 2.1b) also exhibits nonlinear boundary conditions. The position of an arbitrary point P on the workpiece changes continuously during the deformation. Typically at some instant after application of the load on the punch, the surface of the workpiece has stretched such that point P has moved to a new position P^* .

If the position of P is known at every stage of the deformation then material discretization becomes trivial and the F.E. solution may be computed with ease. Haber^[64] proposed a combined Eulerian-Lagrangian displacement model for solving large deformation contact problems. In the model he treats the initial and current configurations as unknown, and keeps track of points on the surface of the workpiece at all times. In this thesis a Lagrangian finite element formulation for analyzing contacting solids is devised. This involved creating new contact elements by pairing up nodes in the surface above the interface with those below it. The result of this amounted to 'filling' the interface with a one-element thick layer of a fictitious medium, then discretizing it in a manner compatible with the discretization of the solids on either side. The unique constitutive relationships for the interface material and other corresponding Finite Element derivations will be discussed in detail in the following sections.

2.1 Choice of Method.

The method chosen and used for stress analysis of contact problems in this thesis is related to the ones suggested by Buragohain and Shah,^{[15][16]} and Ghaboussi et al.^[14] The reasons which led to this choice were:

(a) the use of relative displacements rather than absolute displacements as degrees of freedom reduced the risk of numerical ill-conditioning and

(a): Hertz's Problem
(contact area exaggerated)

(b): Punch Problem.

FIG 2.1. NONLINEAR BOUNDARY CONDITIONS IN CONTACT PROBLEMS.

(b) contact elements fit naturally with regular elements thus implementation and compatibility with existing software in the computing environment at CASCADE was easier to achieve.

2.2 Two-Dimensional Contact Analysis..

In the global reference system $x-y$ (figure 2.2A) a gap is bounded by two quadrilaterals. Notice that the bottom edge of the gap is also the top edge of the bounding element from below, and the top edge is also the bottom edge of the bounding element from below. So globally, the gap (contact) element has four nodes if the bounding edges have no mid-edge nodes, or six nodes if the bounding edges have mid-edge nodes. When referred to the local ($t-n$) axes system (figure 2.2B), the global 4- or 6-node quadrilateral is collapsed onto a two-node or three-node line element respectively. The coordinates ' t ' and ' n ' refer to the tangential and normal directions to the mid-plane of the gap (figure 2.2C). The element level reference coordinate ξ varies along the mid-plane of the gap. The x - and y -coordinates of the local nodes 1, 2, (and 3 when applicable) are computed by averaging the x - and y -coordinates of the corresponding global nodes across the gap.

Grading elements^[28] were designed to support computer modelling of a wide variety of problems in both heat transfer and stress analysis.^{[65][66][67]} Since contact is often a localized phenomenon, the use of graded meshes in contact analysis becomes quite important. The regions in the neighbourhood of contact can be modelled by a very fine mesh and farther from the contact region the mesh becomes gradually coarser. The obvious advantage of using graded meshes is the lowering of computation cost without the risk of loss of accuracy. The finite element discretization of contacts was done by formulating graded contact elements that were compatible with the 4-8 node graded quadrilateral used to model the continuum on either side of the gap. This essentially meant that the isoparametric 2- or 3-node line element has shape functions computed as follows:

$$N_3 = 1 - |\xi| \quad (2.1)$$

$$N_2 = \frac{1}{2}(1 + \xi) - \frac{1}{2}N_3 \quad (2.2)$$

$$N_1 = \frac{1}{2}(1 - \xi) - \frac{1}{2}N_3 \quad (2.3)$$

A: Contact region between quadrilateral elements.

B: Contact element to model contact region.

C: Element level nodes and coordinate ξ .

FIG. 2.2: TOPOLOGY OF 2D CONTACT ELEMENTS.

Notice that from figures 2.3a-2.3c the shape functions N_1 , N_2 and N_3 are linear whether or not the mid-edge node is present. The corresponding derivatives of the shape functions are:

$$\frac{\partial N_3}{\partial \xi} = -1 \quad \text{for } \xi > 0 \quad (2.4)$$

$$\frac{\partial N_3}{\partial \xi} = 1 \quad \text{for } \xi < 0 \quad (2.5)$$

$$\frac{\partial N_2}{\partial \xi} = \frac{1}{2} - \frac{1}{2} \frac{\partial N_3}{\partial \xi} \quad (2.6)$$

$$\frac{\partial N_1}{\partial \xi} = -\frac{1}{2} - \frac{1}{2} \frac{\partial N_3}{\partial \xi} \quad (2.7)$$

The isoparametric relationships are:

(1) for geometry:

$$x(\xi) = \sum_{i=1}^{nn} N_i(\xi) x_i \quad (2.8)$$

$$y(\xi) = \sum_{i=1}^{nn} N_i(\xi) y_i \quad (2.9)$$

(2) for relative displacements:

$$\Delta u_t(\xi) = \sum_{i=1}^{nn} N_i \Delta u_{ti} \quad (2.10)$$

$$\Delta u_n(\xi) = \sum_{i=1}^{nn} N_i \Delta u_{ni} \quad (2.11)$$

where i is a node counter, nn is the number of nodes in the line element (2 or 3), and the subscripts t and n denote directions tangential and normal to the mid-plane of the gap. In the following sections, vectors and matrices will be given for the 3-node line element, but in the actual computer programme they take values corresponding to the 2- or 3-node elements as the analyst defines them in the Finite Element mesh.

2.2.1 The B-matrix.

The B-matrix to be used later in the volume integral for computing the stiffness matrix is assembled as follows:

$$[B] = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \quad (2.12)$$

2- (a)

2 - (b)

Basis functions if mid-edge is present.

(c): Basis functions if mid-edge node is absent.

FIG. 2 3: SHAPE FUNCTIONS FOR 2D CONTACT ELEMENTS.

2.2.2 Relative displacement transformations.

A relative nodal displacement vector for the gap element may be written as:

$$[\Delta\delta]^T = [\Delta u_{t1}, \Delta v_{n1}, \Delta u_{t2}, \Delta u_{n2}, \Delta u_{t3}, \Delta u_{n3}] \quad (2.13)$$

A vector ϵ of relative global displacements based on the isoparametric transformation may be computed by combining equations 2.12 and 2.13:

$$\epsilon = [B]\Delta\delta \quad (2.14)$$

A transformation of the global relative displacement vector ϵ to an element relative displacement vector ϵ_e may be defined such that:

$$\epsilon_e = [T]\epsilon \quad (2.15)$$

which from equation 2.14 may be written as:

$$\epsilon_e = [T][B]\Delta\delta \quad (2.16)$$

The matrix $[T]$ is a transformation which maps the 2D global quadrilateral onto the 1D line element and it is computed from the Jacobian transformation and its determinant as:

$$T = \frac{1}{\det J} \begin{bmatrix} \frac{\partial x}{\partial \xi} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial y}{\partial \xi} & \frac{\partial x}{\partial \xi} \end{bmatrix} \quad (2.17)$$

Q , the determinant of the Jacobian is computed as:

$$\det J = \left[\left(\frac{\partial x}{\partial \xi} \right)^2 + \left(\frac{\partial y}{\partial \xi} \right)^2 \right]^{\frac{1}{2}} \quad (2.18)$$

No further details of matrix transformations will be given here, however, Tong and Rossetos^[68] and Nye^[69] give a detailed coverage.

To satisfy the equilibrium equation in an integral sense along ξ , the nodal relative displacement vector is related to the nodal force vector via the constitutive property values for the contact element:

$$F_e = C\epsilon_e \quad (2.19)$$

In matrix notation, this takes the form:

$$\begin{bmatrix} F_t \\ F_n \end{bmatrix} = \begin{bmatrix} K_t & 0 \\ 0 & K_n \end{bmatrix} \begin{bmatrix} \Delta\delta_t \\ \Delta\delta_n \end{bmatrix} \quad (2.20)$$

where K_t and K_n are the tangential and normal stiffnesses per unit length of the gap. For rough contact K_t and K_n can have values as high as the shear modulus and Young's modulus, respectively, of the softer of the two materials in contact while in smooth contact they can have considerably lower values.

2.2.3 Relative to Absolute Displacements.

In order to connect gap elements with regular elements, the nodes they share must have the same degrees of freedom. To meet this requirement the three node gap element with 6 relative displacements is transformed into a 6-node element with 12 absolute displacements. This necessitates computing the relative displacement vector $\Delta\delta$ from the absolute displacement vector δ . To do this we need a transformation matrix G such that:

$$\Delta\delta = G\delta \quad (2.21)$$

where G essentially subtracts nodal displacements of the bottom edge from the top edge

$$[G] = \begin{bmatrix} -1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 1 \end{bmatrix}$$

Quite clearly since nodes 1, 2 and 5 define the bottom edge of the gap and nodes 3, 4 and 6 define the top edge, the absolute nodal displacement vector may be represented as:

$$[\delta]^T = [u_{t1}^B, u_{n1}^B, u_{t2}^B, u_{n2}^B, u_{t5}^T, u_{n5}^T, u_{t4}^T, u_{n4}^T, u_{t3}^B, u_{n3}^B, u_{t6}^T, u_{n6}^T] \quad (2.22)$$

Superscripts T and B on u denote top and bottom edges respectively, and superscript T on $[\delta]$ denotes the transpose operator.

By substituting equation 2.21 into equation 2.16 the element level relative displacement vector becomes:

$$\epsilon_e = [T][B][G]\delta \quad (2.23)$$

and the force-relative displacement relationships become:

$$F_e = [C][T][B][G]\delta \quad (2.24)$$

2.2.4 The stiffness matrix

The total potential energy of a body may be concisely described by the following relationship:

$$\Pi = \frac{1}{2} \int_V \epsilon^T C \epsilon dV - \int_V u^T f^B dV - \int_S u^{S^T} f^S dS - \sum_i u^{i^T} F^i \quad (2.25)$$

On the right hand side the first term is the strain energy, the second is the contribution to the potential energy from distributed body forces, the third is the contribution to the potential energy from distributed surface tractions, and the fourth is the contribution to the potential energy from discrete point forces.

From equation 2.23 the strain energy associated with contact is really the relative displacement interpolated at a point in the contact element using the isoparametric relationship. For convenience, equation 2.23 is rewritten as:

$$\epsilon_e = B^* \sigma$$

where $B^* = TBG$.

The minimum of Π is invoked by setting $\delta\Pi = 0$, which yields:

$$\int_V \delta u^{i^T} B^{*T} C B^* u^i dV = \int_V \delta u^{i^T} N^T f^B dV + \int_S \delta u^{i^T} N^T f^S dS + \sum_i \delta u^{i^T} F^i \quad (2.26)$$

By letting all the load terms on the R.H.S. sum their contributions into one vector P , equation 2.26 may be rewritten as:

$$\delta u^{i^T} \left(\int_V B^{*T} C B^* dV - \delta P \right) = 0 \quad (2.27)$$

In 2D contact analysis the strain energy integral is in fact a line integral. And since δ is arbitrary the minimum of the total potential energy in contact analysis becomes:

$$\int_L B^T C B dL \delta = P \quad (2.28)$$

Notice that equation 2.28 is the equilibrium equation for contact analysis and it takes the standard Finite Element Analysis format:

$$K \delta u = P$$

By using the definition of B^* from above, equation 2.28 becomes:

$$\int_L G^T B^T T^T C T B G dL \delta = P \quad (2.29)$$

From equation 2.29 the stiffness matrix may be evaluated as:

$$K = \int_L G^T B^T T^T C T B G dL \quad (2.30)$$

which at the element level is the closed integral:

$$K = \int_1^8 G^T B^T T^T C T B G \det J d\xi \quad (2.31)$$

This integral is computed numerically using Gaussian integration which is summarized in the following relationship:

$$K = \sum_{i=1}^{NG} W_i(\xi) G^T B^T T^T C T B G \det J \quad (2.32)$$

In which NG is the number of Gauss points, $W_i(\xi)$ are the Gauss point weights and $\det J$ is the determinant of the Jacobian transformation as computed from equation 2.18.

2.3 Three-Dimensional Contact Analysis.

Three-dimensional (3D) contact analysis was treated in a very similar fashion to 2D. The principle difference is that contact was on a curved surface in 3D instead of a line in 2D.

In the global x-y-z reference system (figure 2.4a), a gap is bounded by two bricks. The total number of nodes in the bottom and top surfaces of the bounding elements

A: -Contact region between two brick elements.

B: Contact element to model contact region.

FIG. 2.4: TOPOLOGY OF 3D CONTACT ELEMENTS.

C Contact surface quadrilateral

D: Element Level 2×2 integration domain.

FIG. 2.4: TOPOLOGY OF 3D CONTACT ELEMENTS.

determines the eventual number of nodes for the contact brick (fig. 2.4b). In this case bounding elements belong to the hierarchical 8-26 node brick family, generating 8-16 contact bricks. At the element level the contact brick is essentially collapsed onto a 4-8 node surface quadrilateral (figure 2.4c). In terms of the number of nodes, it is necessary that the bottom face of the contact brick be a mirror image of the top face. The element surface quadrilateral in the x-y-z coordinate system is mapped onto a 2×2 square in the integration domain denoted by the $\xi - \eta$ coordinate system (figure 2.4d). The shape functions for the 4-8 node contact quadrilateral take the form:

$$N_8 = \frac{1}{2} (1 + \xi) (1 - |\eta|) \quad (2.33)$$

$$N_7 = \frac{1}{2} (1 - |\xi|) (1 - \eta) \quad (2.34)$$

$$N_6 = \frac{1}{2} (1 - \xi) (1 - |\eta|) \quad (2.35)$$

$$N_5 = \frac{1}{2} (1 - |\xi|) (1 + \eta) \quad (2.36)$$

$$N_4 = \frac{1}{4} (1 + \xi) (1 - \eta) - \frac{1}{2} (N_7 + N_8) \quad (2.37)$$

$$N_3 = \frac{1}{4} (1 - \xi) (1 - \eta) - \frac{1}{2} (N_6 + N_7) \quad (2.38)$$

$$N_2 = \frac{1}{4} (1 - \xi) (1 + \eta) - \frac{1}{2} (N_5 + N_6) \quad (2.39)$$

$$N_1 = \frac{1}{4} (1 + \xi) (1 + \eta) - \frac{1}{2} (N_5 + N_8) \quad (2.40)$$

The corresponding derivatives of the shape functions:

$$\frac{\partial N_8}{\partial \xi} = \frac{1}{2} (1 - |\eta|) \quad (2.41)$$

$$\frac{\partial N_8}{\partial \eta} = -\frac{1}{2} (1 + \xi) \quad \text{for } \eta > 0 \quad (2.42)$$

$$\frac{\partial N_8}{\partial \eta} = \frac{1}{2} (1 + \xi) \quad \text{for } \eta < 0 \quad (2.43)$$

$$\frac{\partial N_7}{\partial \xi} = -\frac{1}{2} (1 - \eta) \quad \text{for } \xi > 0 \quad (2.44)$$

$$\frac{\partial N_7}{\partial \xi} = \frac{1}{2} (1 - \eta) \quad \text{for } \xi < 0 \quad (2.45)$$

$$\frac{\partial N_7}{\partial \eta} = -\frac{1}{2}(1 - |\xi|) \quad (2.46)$$

$$\frac{\partial N_6}{\partial \xi} = -\frac{1}{2}(1 - |\eta|) \quad (2.47)$$

$$\frac{\partial N_6}{\partial \eta} = -\frac{1}{2}(1 - \xi) \quad \text{for } \eta > 0 \quad (2.48)$$

$$\frac{\partial N_6}{\partial \eta} = \frac{1}{2}(1 - \xi) \quad \text{for } \eta < 0 \quad (2.49)$$

$$\frac{\partial N_5}{\partial \xi} = -\frac{1}{2}(1 + \eta) \quad \text{for } \xi > 0 \quad (2.50)$$

$$\frac{\partial N_5}{\partial \xi} = \frac{1}{2}(1 + \eta) \quad \text{for } \xi < 0 \quad (2.51)$$

$$\frac{\partial N_5}{\partial \eta} = \frac{1}{2}(1 - |\xi|) \quad (2.52)$$

$$\frac{\partial N_4}{\partial \xi} = \frac{1}{4}(1 - \eta) - \frac{1}{2} \left(\frac{\partial N_7}{\partial \xi} + \frac{\partial N_8}{\partial \xi} \right) \quad (2.53)$$

$$\frac{\partial N_4}{\partial \eta} = -\frac{1}{4}(1 + \xi) - \frac{1}{2} \left(\frac{\partial N_7}{\partial \eta} + \frac{\partial N_8}{\partial \eta} \right) \quad (2.54)$$

$$\frac{\partial N_3}{\partial \xi} = -\frac{1}{4}(1 - \eta) - \frac{1}{2} \left(\frac{\partial N_6}{\partial \xi} + \frac{\partial N_7}{\partial \xi} \right) \quad (2.55)$$

$$\frac{\partial N_3}{\partial \eta} = -\frac{1}{4}(1 - \xi) - \frac{1}{2} \left(\frac{\partial N_6}{\partial \eta} + \frac{\partial N_7}{\partial \eta} \right) \quad (2.56)$$

$$\frac{\partial N_2}{\partial \xi} = -\frac{1}{4}(1 + \eta) - \frac{1}{2} \left(\frac{\partial N_5}{\partial \xi} + \frac{\partial N_6}{\partial \xi} \right) \quad (2.57)$$

$$\frac{\partial N_2}{\partial \eta} = \frac{1}{4}(1 - \xi) - \frac{1}{2} \left(\frac{\partial N_5}{\partial \eta} + \frac{\partial N_6}{\partial \eta} \right) \quad (2.58)$$

$$\frac{\partial N_1}{\partial \xi} = \frac{1}{4}(1 + \eta) - \frac{1}{2} \left(\frac{\partial N_5}{\partial \xi} + \frac{\partial N_8}{\partial \xi} \right) \quad (2.59)$$

$$\frac{\partial N_1}{\partial \eta} = \frac{1}{4}(1 + \xi) - \frac{1}{2} \left(\frac{\partial N_5}{\partial \eta} + \frac{\partial N_8}{\partial \eta} \right) \quad (2.60)$$

The B-matrix takes the form:

$$[B] = \begin{bmatrix} N_1 & 0 & 0 & . & . & N_8 & 0 & 0 \\ 0 & N_1 & 0 & . & . & 0 & N_8 & 0 \\ 0 & 0 & N_1 & . & . & 0 & 0 & N_8 \end{bmatrix} \quad (2.61)$$

The Jacobian transformation which maps a 3D brick onto a 2D surface in space is computed by relationship 2.62. Note the thickness of contact elements is assumed to be negligible, hence the third row corresponding to the third element level dimension does not appear.

$$[J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \end{bmatrix} \quad (2.62)$$

The corresponding transformation which maps the global relative displacement vector to the element relative displacement vector is:

$$[T] = \begin{bmatrix} T_{1x} & T_{1y} & T_{1z} \\ T_{2x} & T_{2y} & T_{2z} \\ T_{3x} & T_{3y} & T_{3z} \end{bmatrix} \quad (2.63)$$

The first, third and second rows of the Transformation matrix in that order are computed by projections of the Jacobian matrix V_ξ and V_η as follows:

$$T_{1i} = \frac{V_{\xi i}}{|V_\xi|} \quad (2.64)$$

$$T_{3i} = \frac{V_\xi \times V_\eta}{|V_\xi||V_\eta|} \quad (2.65)$$

$$T_{2i} = T_{3i} \times T_{1i} \quad (2.66)$$

The force relative displacement relationships in 3D become:

$$\begin{bmatrix} F_x \\ F_y \\ F_z \end{bmatrix} = \begin{bmatrix} K_x & 0 & 0 \\ 0 & K_y & 0 \\ 0 & 0 & K_z \end{bmatrix} \begin{bmatrix} \Delta \delta_x \\ \Delta \delta_y \\ \Delta \delta_z \end{bmatrix} \quad (2.67)$$

The G-matrix is of the same form in 3D as it was in 2D, however, its dimensions range from 12×24 to 24×48 corresponding to the 8-16 global nodes in the contact brick.

2.4 Finite Element Solution.

At this point the element stiffness matrix and load vector have been computed. The upper triangular portion of the stiffness matrix and the load vector as is, are 'sent' to the frontal solver for:

- (i) Assembly of the global stiffness matrix and global load vector for the entire Finite Element mesh.
- (ii) A sequence of trial solutions is computed for each step in the analysis.

From this stage the computed results are written to an output file and a graphics database after equilibrium and convergence checks are performed. If the problem is nonlinear a post-processing procedure begins whereby residual loads are computed from one of a number of nonlinear algorithms (one used commonly was the Newton-Raphson algorithm) and sent back to the solver for iterative computations. The cycle repeats until a predetermined tolerance from one of a number of convergence norms is reached. Examples of such convergence norms include force, energy and displacement.

2.4.1 Iteration Process by the Newton-Raphson Method.

The state of equilibrium requires that the sum of internal nodal forces of a continuum balances the applied external load, such that:

$$F + P = 0 \quad (2.68)$$

where F is the internal nodal force vector and P is the applied external load vector.

Since F is computed from the displacement dependent stiffness matrix we may write

$$K(u)u + P = 0 \quad (2.69)$$

where K denotes the stiffness matrix and u the nodal displacement vector. This is clearly a nonlinear relationship for which values of K and u that will satisfy the equilibrium condition exactly might not be computable in one step. An approximation method is chosen for which initial values of K and u are used, and an iterative procedure is used to refine the solution until it converges to a desired level of accuracy.

For the Newton-Raphson method a residual force vector Ψ is computed by substituting the value of the current displacement into the equilibrium equation 2.69

$$K(u^n)u^{n+1} + P = \Psi(u^n) \quad (2.70)$$

where superscript n denotes the n^{th} iteration. The residual force for the next iteration is obtained from the truncated Taylor expansion:

$$\Psi(u^{n+1}) = \Psi(u^n) + (d\Psi/du)_n \Delta u^n \quad (2.71)$$

In equation 2.71 $(d\Psi/du)_n$ implies the change of residual force with displacement evaluated at the n^{th} iteration.

Mathematically

$$(d\Psi/du)_n = K(u^n) \quad (2.72)$$

The improved value of displacement is related to the current one by:

$$u^{n+1} = u^n + \Delta u^n \quad (2.73)$$

and

$$\Delta u^n = -K(u)^{-1} \Psi(u^n) \quad (2.74)$$

2.4.2 Incremental methods.

In this approach the starting point is again the equilibrium equation. The known case of zero displacement for zero load is often used for the initial values of K and u . The external load is applied in small known increments for which corresponding updated values of displacement are computed.

Equation 2.69 is conveniently rewritten as:

$$K(u)u + \lambda f_o = 0 \quad (2.75)$$

where λ is the fraction of external load applied currently. Differentiating with respect to λ :

$$\frac{d}{d\lambda} (K(u)u) + f_o = 0 \quad (2.76)$$

note:

$$\frac{d}{d\lambda} (K(u)u) = K(u) \frac{du}{d\lambda}$$

Therefore solving for $\frac{du}{d\lambda}$:

$$\frac{du}{d\lambda} = -K(u)^{-1} f_o \quad (2.77)$$

Next the load is incremented by $\Delta\lambda$:

$$u_{m+1} - u_m = -K(u_m)^{-1} f_o \Delta\lambda_m$$

or:

$$u_{m+1} = -K(u_m)^{-1} \Delta f_m + u_m \quad (2.78)$$

The subscript m refers to the m^{th} load increment. At each increment, the external load vector can be changed to follow a given load path. Smaller step lengths will give higher accuracy until roundoff damage becomes the dominant error.

2.4.3 Combined Incremental and Iterative Methods.

The two procedures are generally combined so as to be able to iterate within each load increment. Equation 2.73 can thus be modified as follows:

$$u_{m+1}^{n+1} = u_{m+1}^n + \Delta u_{m+1}^n \quad (2.79)$$

where subscripts represent increments and superscripts represent iteration steps.

In this thesis the Updated Lagrangian procedure was used to handle loads in a multiple step analysis. The first iteration of the first load increment is generally taken to be stress free, but for subsequent load increments the initial stress state becomes the stress state at the end of the preceding load increment. During the entire analysis three displacement vectors were maintained:

- (1) A total displacement vector 'DISTOT' which is an accumulation of displacements from the start of the analysis, i.e., from the first iteration of the first increment to the end of the previous load step. This displacement vector defines the deformed geometry at the beginning of the increment. DISTOT was used mainly for graphics.

(2) An incremental displacement vector 'DISINC' which is an accumulation of iterative displacements within a load increment. This is the displacement that defines the geometry at the end of the increment for the current iteration. It is written to the output file.

(3) The most recent iterative displacement fresh from the frontal solver was kept as the vector 'DISITR'.

2.5 Properties of Contact Elements.

2.5.1 Linear-Rigid Behaviour.

At the interface of two solids, a contact element could have linear properties such that it will permit the two solids to come together and rest on each other without surface penetration. In order to behave thus, the contact element must have a force-relative displacement relationship of the type shown in figure 2.5 such that as long as a gap exists (i.e. $\Delta u > 0$), the stiffness of the gap element is 'rubbery' and linear. However, as soon as the gap closes (i.e. $\Delta u = 0$) the stiffness changes and becomes infinite. An infinite stiffness cannot be represented on a digital computer. A penalty method reduces the infinite stiffness to a 'large' stiffness, say 5 times the stiffness of the contacting materials.

This particular feature is not very useful in Engineering applications since often surface indentation does in fact take place. When this happens the nature of the problem changes considerably causing potential nonlinearity. This occurs because new contact areas and surface conditions emerge. Typical problems that may be encountered when contact areas and surface conditions change include solution oscillation or divergence.

2.5.2 Nonlinear Behaviour

A more attractive feature is one in which surface deformation and possible penetration is anticipated hence provision for surface release is incorporated. A close analogy to the compound spring problem that can model this feature, is summarized in figure 2.6. As long as the applied load produces a displacement less than or equal to the gap width 'a' the small spring stiffness k is operative. As soon as the displacement becomes greater than 'a' and a surface penetration of magnitude δ takes place, the larger stiffness K becomes operative. Note $K \gg k$. This more realistic feature was chosen and used. From the force-relative displacement relations if the loading or unloading cycle is entirely on either

part of the curve (i.e. either k or K' is operative) then quite clearly the problem is linear and the solution is obtained in one computation step. If, however, the loading or unloading cycle involves crossing the 'knee' then the solution will have to be obtained in more than one step iteratively. The algorithm that does this is summarized by figure 2.7 whereby:

1. F_{app} is the externally applied load
2. A corresponding displacement u is computed, the stiffness value used being k .
3. An internal load F_{int} is calculated.
4. The difference between F_{int} and F_{app} is the out of balance force Ψ which is used as the applied load in computing the iterative displacement.

After each iterative step a convergence check is performed by one of three commonly used convergence norms namely energy, force and displacement. The solution for the original applied force F_{app} is point B. In iterative problems the 'exact' solution is seldom attained, rather convergence is approached to a specified tolerance level in the chosen norm(s).

FIG. 2.5: BILINEAR MATERIAL PROPERTIES FOR CONTACT ELEMENTS.

FIG. 2.6: NONLINEAR MATERIAL PROPERTIES FOR CONTACT ELEMENTS.

FIG. 2.7: MODEL FOR NONLINEAR PROPERTIES OF CONTACT ELEMENTS.

2.6 Test Problems.

A Finite Element idealization of contact problems in stress analysis along with an algorithm for computing interface stiffnesses akin to the physical problem of a compound spring was developed. The accuracy of the proposed method was tested on simple test problems before realistic engineering applications were attempted.

The ultimate test problem would probably be to solve Hertz's problem of a cylinder/plate interaction (figure 2.8) analytically, then discretize the same geometry and solve it by the Finite Element Method incorporating the contact analysis for the interface, and compare the results. Equation 1.3 is used to compute Hertz's analytical solution of two cylinders in contact. By setting d_2 to ∞ it was possible to use it to solve the problem of a cylinder/plate interaction. First however, simple verifiable test problems had to be run to establish the physical soundness of the analysis. One such problem is shown in figure 2.9. For generality, arbitrary material properties in the continuum elements (1 & 3) were: Young's modulus $E = 1.4 \times 10^8 \text{ N/m}^2$, $\nu = .3$, dimensions $1\text{m} \times 1\text{m}$. For the gap element the stiffness per unit length normal to the gap was $K_n = 1.4 \times 10^8 \text{ N/m}$. A total compressive load 0.05 N was applied, other boundary conditions can be seen on the diagram.

2.6.1 Displacements.

By applying elasticity relationships within each element:

$$u_1^3 = u_3^7 = -\frac{Fl}{AE} = -\frac{0.05 \times 1}{1 \times 1.4 \times 10^8} = -3.5714 \times 10^{-8} \text{ m}$$

$$u_2^6 = \frac{F}{K_n} = -\frac{0.05}{1.4 \times 10^8} = -3.5714 \times 10^{-8} \text{ m}$$

where the superscripts refer to the node number and the subscripts refer to the element number.

Globally:

$$u^3 = u_1^3 = -3.5714 \times 10^{-8} \text{ m}$$

$$u^6 = u_1^3 + u_2^6 = -7.14286 \times 10^{-8} \text{ m}$$

$$u^7 = u_1^3 + u_2^6 + u_3^7 = -1.07143 \times 10^{-7} \text{ m}$$

FIG. 2.8: MODEL FOR HERTZ'S CYLINDER/PLATE INTERACTION.

FIG. 2.9: SIMPLE TEST PROBLEM FOR ANALYTICAL VERIFICATION OF CONTACT ALGORITHM.

TEST 2.9 OUTPUT

Displacements for right hand side no. 1 Nel = 1

Node No.	1	2	3	4
X-Disp.	0.00000E+00	0.10714E-09	0.10714E-09	0.00000E+00
Y-Disp.	0.00000E+00	0.00000E+00	-.35714E-09	-.35714E-09

Displacements for right hand side no. 1 Nel = 2

Node No.	4	3	6	5
X-Disp	0.00000E+00	0.10714E-09	0.10714E-09	0.00000E+00
Y-Disp	-.35714E-09	-.35714E-09	-.71429E-09	-.71429E-09

Displacements for right hand side no 1 Nel = 3

Node No	5	6	7	8
X-Disp	0.00000E+00	0.10714E-09	0.10714E-09	0.00000E+00
Y-Disp.	-.71429E-09	-.71429E-09	-.10714E-08	-.10714E-08

Output for element 1, for iteration 1 and increment 1

G.P.	XX-STRESS	YY-STRESS	XY-STRESS	ZZ-STRESS	MAX P.S.
1	-.152260E-09	-.500000E-01	-.549190E-10	0.000000E+00	-.152260E-09
2	-.167844E-10	-.500000E-01	-.621321E-10	0.000000E+00	-.167844E-10
3	-.158443E-09	-.500000E-01	-.750239E-11	0.000000E+00	-.158443E-09
4	-.229671E-10	-.500000E-01	-.147155E-10	0.000000E+00	-.229671E-10

Stresses in Element 2, Increment 1, Iteration 1.

G.P.	X-STRESS	Y-STRESS	XDISP	YDISP	KX
1	0.380487E-09	-.500000E-01	0.257491E-17	-.357143E-09	0.140000E+09

Output for element 3, for iteration 1 and increment 1

G.P.	XX-STRESS	YY-STRESS	XY-STRESS	ZZ-STRESS	MAX P.S.
1	-.500626E-10	-.500000E-01	-.794367E-10	0.000000E+00	-.500626E-10
2	0.403862E-09	-.500000E-01	-.794367E-10	0.000000E+00	0.403862E-09
3	-.500627E-10	-.500000E-01	0.794367E-10	0.000000E+00	-.500627E-10
4	0.403862E-09	-.500000E-01	0.794367E-10	0.000000E+00	0.403861E-09

Iterative increment converged.

RIGHT HAND SIDE 1

NQDE	CODE	REACTIONS DUE TO PRESCRIBED VARIABLES
1	11	0.34817220E-10 0.25000000E-01
2	1	0 00000000E+00 0.25000000E-01
4	10	- .12744691E-09 0.00000000E+00
5	10	0.33439952E-11 0.00000000E+00
8	10	- .27682657E-17 0.00000000E+00

2.6.2 Stresses within the elements.

For elements 1 & 3 $\sigma_{yy} = -\frac{F}{A} = -.05 \text{ Pa}$

It is negative because the applied force was compressive.

For element 2 (the interface element) stresses are computed from the global displacements of the nodes it shares with the bounding continuum elements. Thus from compatibility and continuity normal stress in element 2 should be $-.05 \text{ Pa}$

2.6.3 Force equilibrium.

The sum of applied forces should be equal in magnitude but of opposite sign to the reactions at the appropriate fixed nodes. Mathematically $\sum F_y = -\sum R_y$

By referring to 'TEST 2.9 OUTPUT' which is part of the output file from the F.E.A. run of the problem shown in figure 2.9, it is evident that the Finite Element solution reproduced the analytical solution exactly. Many other simple and verifiable test problems aimed at testing the conditioning of the analysis with graded meshes having variable node quadrilaterals were successfully completed. In all cases, the exact solution was computed.

At this point sufficient confidence in the analysis had been established, so a bigger, more practical problem (Hertz's problem of a cylinder/plate interaction) illustrated in figure 2.10 was attempted. Notice only half of the problem was discretized because of symmetry. For this problem, $F = 2.244 \times 10^6 \text{ N}$, the cylinder diameter was 1 m and $\nu = .3$.

An interesting result was observed, in that the cylinder flattened noticeably on the plate. As well, it was noticed that the normal stresses in the gap were high and compressive near the tangent point ($x = 0$) but rapidly dropped off to zero then became tensile as contact broke off (figure 2.11). The mesh was obviously too coarse to show any realistic results.

A very small portion ($40\text{mm} \times 40\text{mm}$) in the neighbourhood of contact was discretized into a fine graded mesh, figure 2.12. With this mesh, F.E.A. results of normal interface stresses compared very well with those from analytical calculations. With a deeper plate the results were even better, which is logical because ideally Hertz's analytical method is

one of a long cylinder interfacing a semi-infinite plate. A summary of results is given in figure 2.13.

After extending the contact analysis to 3D it was also necessary to run a series of test problems to ascertain the accuracy and confidence in the analysis existed in 3D as it did in 2D. This was done successfully. Figure 2.14 shows a graded 3D mesh with 8-26 node bricks in the continuum and corresponding 8-16 node contact bricks in the interface. The dimensions of the problem were arbitrary and a uniform compressive load in the z-direction was applied on the slab. The result was a constant normal stress in the z-direction in the continuum elements as well as the interface elements. This was a very significant result because irrespective of the element size or the number of nodes, the normal stress was constant indicating that the contact algorithm was well conditioned.

FIG. 2.10: COARSE F.E. MESH FOR HERTZ'S CYLINDER/PLATE PROBLEM.

FIG. 2.11: DEFORMED SHAPE OF FIGURE 2.10.

FIG. 2.12: GRADED F.E. MESH OF A PORTION ($40\text{mm} \times 40\text{mm}$)
IN FIGURE 2.10 NEAR THE CONTACT ZONE.

FIG. 7 13: COMPARISON OF F.E. RESULTS WITH HERTZ'S SOLUTION.

FIG. 2.14: 3D GRADED MESH WITH 8-16 NODE CONTACT BRICKS USED FOR TESTING CONDITIONING OF CONTACT ALGORITHM.

3. DYNAMIC ANALYSIS

3.1 Introduction

The dynamic response of a Finite Element assemblage is computed from the governing general equilibrium equation:

$$M\ddot{X} + C\dot{X} + KX = R \quad (3.1)$$

where M , C and K are the mass, damping and stiffness matrices respectively, and $\ddot{X}$, $\dot{X}$, X and R are the acceleration, velocity, displacement and load vectors for the Finite Element assemblage. Each term in equation 3.1 is a function of time and terms on the left hand side are the internal forces (inertia, damping and elastic) which at any time are in equilibrium with the external forces on the right hand side.

Equation 3.1 represents a set of second order differential equations whose solution is obtainable through standard procedures. Details for seeking the solutions of differential equations, which can be as varied as the nature and order the coefficient matrices M , C and K , will not be discussed in this thesis. There are however two methods which have been adapted to Finite Element analysis models, namely Direct Integration and Mode Superposition.

In direct integration, numerical integration procedures are performed in a step by step fashion directly on the differential equations, that is, no transformation is done on the differential equation before the integration. 'Step by step' refers to two ideas which are in essence the backbone of all direct integration methods. First, the solution of the equilibrium equation is sought at discrete times within the solution domain. So at these discrete times (separated by time intervals Δt) the solution is similar to that of a static equilibrium, only the left hand side includes inertia and damping forces additional to elastic forces. Second, the variation of displacements, velocities and accelerations within each time interval is assumed. The form of these variations which translates into a number of algorithms, determines the accuracy of the solution method, and just as importantly, the cost and sensitivity to the time step length.

3.2 Central Difference Method.

In this thesis some aspects of the central difference method were used for computing the solution of contact problems in stress analysis. The main assumptions of the central difference method are based on the numerical values of the response over two time steps:

$${}^t\ddot{X} = \frac{1}{\Delta t^2} ({}^{t-\Delta t}X - 2{}^tX + {}^{t+\Delta t}X) \quad (3.2)$$

$${}^t\dot{X} = \frac{1}{2\Delta t} ({}^{t+\Delta t}X - {}^{t-\Delta t}X) \quad (3.3)$$

In solving the equilibrium equation at time 't' equation 3.1 may be more correctly rewritten as

$$M{}^t\ddot{X} + C{}^t\dot{X} + K{}^tX = {}^tR \quad (3.4)$$

Where the superscript refers to the acceleration, velocity, displacement and load vectors at time 't'.

Substituting equations 3.2 and 3.3 into equation 3.4 the equilibrium equation over two time steps $t - \Delta t$: t , t : $t + \Delta t$ may be written as:

$$M \left[\frac{1}{\Delta t^2} ({}^{t-\Delta t}X - 2{}^tX + {}^{t+\Delta t}X) \right] + C \left[\frac{1}{2\Delta t} ({}^{t+\Delta t}X - {}^{t-\Delta t}X) \right] + K [{}^tX] = {}^tR \quad (3.5)$$

Finally since we are interested in solving equation 3.5 at $t + \Delta t$ it is convenient to rearrange it as follows:

$$\left[\frac{1}{\Delta t^2}M + \frac{1}{2\Delta t}C \right] {}^{t+\Delta t}X = {}^tR - \left[K - \frac{2}{\Delta t^2}M \right] {}^tX - \left[\frac{1}{\Delta t^2}M - \frac{1}{2\Delta t}C \right] {}^{t-\Delta t}X \quad (3.6)$$

Some of the direct integration methods are also called forward difference methods because they use equilibrium conditions at time t to compute the solution at time $t + \Delta t$. This category includes explicit integration. Implicit integration methods like the Newmark, Houbolt and Wilson methods use equilibrium conditions at time $t + \Delta t$ and compute the solution at time $t + \Delta t$.

3.2.1 Implicit/Explicit Integration.

The choice of implicit or explicit integration for solving a particular problem is not trivial. It depends upon the size and nature of the problem, the frequency of loads and

the computer resources available. Explicit integration has the advantage that it does not involve the inversion of the effective stiffness matrix, so it is not quite as computational intensive as the implicit method for which matrix inversion is necessary.^[29] The explicit method however, is very sensitive to the time step length.^{[70][71][72]}

For the implicit method, the integration parameters and the ratio of the time step length to the period ($\Delta t/T$) determines stability. The unconditional stability of an integration method exists if for any given set of initial conditions the solution will not diverge for any time step Δt , especially if Δt is large. Conditional stability holds if the solution does not diverge provided that Δt less than the stability limit. The Central Difference Method is conditionally stable. With the choice of appropriate integration parameters, the Houboldt, Wilson and Newmark methods are unconditionally stable. However, the accuracy of the solution is not guaranteed. The choice of time step size that will give an accurate solution should depend upon the frequency of the loads. Thus to evaluate the transient response of high frequency loads small time steps are necessary while larger time steps should suffice for low frequency loads. A poor choice of time step size may cause errors in the computed response because it may cause period elongation and/or amplitude decay.^[29]

Recent publications^{[73][74][75][76][77][78]} indicate that a mixed implicit-explicit solution method would be more effective for solving large dynamic response problems. This would necessitate dividing the mesh into portions to be solved by the implicit frontal solver and portions to be solved by the explicit solver.

Hallquist and Goudreau^{[33][55][56][57]} of Lawrence Livermore National Laboratories give a very elaborate procedure for solving the most general contact-impact problem. These, along with the respective computer programmes like NIKE2D, DYNA3D and mesh generating and graphics software are available in the public domain. More recently Benson et al^[31] published a rigid body algorithm for structural dynamics. These publications and software are quite elegant, however their implementation may be limited to supercomputers because they were written in a vectorized format.

3.3 The Newmark Implicit Algorithm.

The Newmark implicit algorithm assumes that acceleration over a time step is linear. It further assumes that variation of velocity and displacement may be computed as follows:

$${}^{t+\Delta t}\ddot{X} = {}^t\ddot{X} + [(1 - \delta){}^t\ddot{X} + \delta{}^{t+\Delta t}\ddot{X}] \Delta t \quad (3.7)$$

$${}^{t+\Delta t}\dot{X} = {}^t\dot{X} + {}^t\dot{X}\Delta t + \left[\left(\frac{1}{2} - \alpha \right) {}^t\ddot{X} + \alpha {}^{t+\Delta t}\ddot{X} \right] \Delta t^2 \quad (3.8)$$

α and δ are integration constants to be determined *a priori* to ensure stability and accuracy. In this thesis $\delta = .5$ and $\alpha = .25$ were chosen for they give an unconditionally stable scheme with acceleration over a time step being constant. In the computer code however provision was made for flexibility of choice of integration parameters.

For problems demonstrated in this thesis it was assumed that physical damping was zero. However in the computer code provision was made for cases where C is not zero. It is generally understood that the damping matrix is a linear combination of the mass and the stiffness matrices:

$$C = \theta M + \beta K \quad (3.9)$$

The elastic stiffness matrix was derived from the principle of virtual work in a standard volume integral in Finite Element Analysis as:

$$K = \int_{vol} B^T D B \det J dv \quad (3.10)$$

The lumped mass matrix was computed on the basis of dimensions of the element, the density and the requirement of mass conservation within the element.^[32] Lumped mass matrices have proven to work well in structural and solid mechanics problems. The total mass of the element is computed as

$$M = \int_{\Omega} \rho d\Omega \quad (3.11)$$

Entries of the lumped mass matrix are assigned to be proportional to the diagonal entries of the consistent mass matrix and they are positive definite. Entries of the consistent mass matrix are generally computed as:

$$m_{ab} = \int_{\Omega} N_a \rho N_b d\Omega \quad (3.12)$$

The proportionality constant α_c between the lumped and consistent mass matrices is computed as:

$$\alpha_c = \int_{\Omega} \rho d\Omega / \left(\sum_{a=1}^n \int_{\Omega} \rho N_a^2 d\Omega \right) \quad (3.13)$$

The denominator in equation 3.13 is the sum of diagonal entries of the consistent mass matrix.

Lumping is done by combining equations 3.11, 3.12 and 3.13 as follows:

$$m_{ab} = \alpha_c \delta_{ab} \int_{\Omega} \rho N_a^2 d\Omega \quad a = b \quad (3.13a)$$

$$m_{ab} = 0 \quad a \neq b \quad (3.13b)$$

δ_{ij} is the Kronecker delta.

In implementing the Newmark algorithm into a computer programme it was necessary to rewrite equation 3.4 more appropriately with all the time superscripts associated with the acceleration, velocity, displacement and load vectors being replaced by $t + \Delta t$ as follows

$$M^{t+\Delta t} \ddot{X} + C^{t+\Delta t} \dot{X} + K^{t+\Delta t} X = {}^{t+\Delta t} R \quad (3.14)$$

In effect the Newmark algorithm computes ${}^{t+\Delta t} X$ from equation 3.14 by substituting values from equations 3.7 and 3.8. It then updates ${}^{t+\Delta t} \ddot{X}$ and ${}^{t+\Delta t} \dot{X}$ for use in computing the response of the Finite Element assemblage during the next time step. The process then continues. The procedure is summarized concisely in the following sections.

3.3.1 Initial Conditions.

Equation 3.14 has to be satisfied at all times especially at the commencement of the computation, that is, at time $t = 0$. Thus ${}^0 X$, ${}^0 \dot{X}$ and ${}^0 \ddot{X}$ have to be given as data.

3.3.2 Integration Constants.

A number of constants resulting from substitutions of equations 3.7 and 3.8 into 3.14, involving Δt , α and δ are calculated up front as well. Such constants include:

$$A_0 = \frac{1}{\alpha \Delta t^2}, A_1 = \frac{\delta}{\alpha \Delta t}, A_2 = \frac{1}{\alpha \Delta t}, A_3 = \frac{1}{2\alpha} - 1, A_4 = \frac{\delta}{\alpha} - 1, A_5 = \frac{\Delta t}{2} \left(\frac{\delta}{\alpha} - 2 \right),$$

$$A_6 = \Delta t(1 - \delta) \text{ and } A_7 = \delta \Delta t$$

3.3.3 Effective Stiffness Matrix.

An effective stiffness matrix is computed from the elastic stiffness matrix, the mass and damping matrices as follows:

$$K_{eff} = K + A_0 M + A_1 C \quad (3.15)$$

3.3.4 Effective Load Vector.

For each time step an effective load vector is computed as:

$${}^{t+\Delta t}R = {}^tR + M(A_0 {}^tX + A_2 {}^t\dot{X} + A_3 {}^t\ddot{X}) + C(A_1 {}^tX + A_4 {}^t\dot{X} + A_5 {}^t\ddot{X}) \quad (3.16)$$

3.3.5 Time Step Response.

The effective stiffness matrix computed from equation 3.15 and the effective load vector computed from equation 3.16 are passed to the frontal solver for computation of ${}^{t+\Delta t}X$. In preparation for the next time step the acceleration and velocity vectors are updated as follows:

$${}^{t+\Delta t}\ddot{X} = A_0 ({}^{t+\Delta t}X - {}^tX) - A_2 {}^t\dot{X} - A_3 {}^t\ddot{X} \quad (3.17)$$

$${}^{t+\Delta t}\dot{X} = {}^t\dot{X} + A_6 {}^t\ddot{X} + A_7 {}^{t+\Delta t}\ddot{X} \quad (3.18)$$

3.3.6 Nonlinear Dynamic Response.

The procedure described so far is for solving linear second order ordinary differential equations. A problem is considered linear if in equation 3.14 M, C, K and R are independent of X . Otherwise, it becomes nonlinear. The same procedure is used for nonlinear calculations except that within each time step the solution is refined through an iterative procedure until a specified level of accuracy is reached.

$$K_{eff} {}^{t+\Delta t}\Delta X^k = {}^{t+\Delta t}F^{k-1} \quad (3.19)$$

$${}^{t+\Delta t}X^k = {}^{t+\Delta t}X^{k-1} + \Delta X^k \quad (3.20)$$

where ΔX^k is the iterative displacement which is summed into the incremental (time step) displacement ${}^{t+\Delta t}X$, and k is the iteration number. F is the internal nodal load vector computed for each iteration during the nonlinear calculations:

$$F^k = \int B^T \sigma \quad (3.21)$$

3.4 Dynamic-Contact Analysis

In this thesis, dynamic interaction problem analysis was limited to small displacement theory in sliding contact. In order to do a large displacement Finite Element analysis of a generalized contact-impact problem, it is essential to write an efficient programme for accurately keeping track of material points at lines and surfaces of interaction. It may become necessary to change the nodal connectivity (a phenomenon called rezoning) at various stages in the analysis since, as the bodies deform, the shapes of elements become distorted. This could lead to erroneous results or numerical ill-conditioning. Explicit analysis is generally less computational intensive than implicit analysis because it does not involve matrix inversion. This obviously reflects the overall computing cost and in particular the memory requirements.

Hallquist^[33] developed a vectorized implicit programme, NIKE2D for doing large displacement contact analysis. This programme has the ability of keeping track of interface nodes by a slide-line analysis. Together with programmes like DYNA2D and DYNA3D, Hallquist et al^{[55][56][57]} have been able to solve general large displacement contact-impact problems. Contact-impact analysis which was implemented in three methods is described briefly as follows:

3.4.1 Nodal Constraint Method.

In this method, constraints are imposed on the global equilibrium equations by transforming the nodal displacement components of the slave nodes along the contact interface. The transformation eliminates the degree of freedom of the slave node normal to the interface and distributes its normal force component to adjacent master nodes. This method is no longer used because of a number of problems associated with it, notably slide-line kinking and poor zoning caused by different fineness on either side of the interface.

3.4.2 Penalty Method.

In the slide-line analysis by Hallquist^[33] the slave side of the interface is constrained to slide on but not penetrate the master surface. For the penalty method, at places in the interface where surface penetration takes place or the potential exists, penetration is

precluded by the insertion of interface springs into the global stiffness matrix.

In the contact algorithm developed in this thesis a concept akin to the use of interface springs was used. Here, when the slave surface penetrates the master surface or approaches it to within a specified tolerance, the stiffness component of the contact element normal to the interface was scaled up by an order of magnitude of 5. The effect of this was to reduce the possibility of surface penetration. The penalty method was implemented in NIKE2D for large displacement contact analysis with sliding.

3.4.3 Distributed Parameter Method.

This method was implemented in DYNA3D for solving large displacement contact-impact problems with sliding, typically in high pressure explosive situations. It has similarities to the penalty method particularly the definition and handling of slide lines. The enforcement of momentum conservation in the distributed parameter method is briefly described below:

Definition of a slide line. The slide line is first defined by designating 'master' nodes on one side of the interface and 'slave' nodes on the other. Vector analysis is used throughout slide line analysis, hence orientation has to be established at the start. The master-slave designation has to be such that as one moves along the contact zone, the slave nodes are on the left. Algorithms for determining surface penetration and interface pressure are based on this convention. One prerequisite of slide line analysis is that slave nodes may slide on, but not penetrate the master line (or surface).

Slave zone and node search. During each time step the location of each slave node is updated by identifying the master node closest to it, if it is not on the master surface, or by associating it with the first node on the master segment on which it lies.

In addition during each time step a complete search of the interface is done so as to identify the first slave zone that overlaps a given master segment. This information is used at a later stage to compute a distributed pressure for each master segment. With current algorithms these searches consume much CPU time and computer memory.

Tensile interface force calculation. For each slave node that lies in contact with the master segment a check is performed to determine if a tensile force has been developed.

The development of a tensile interface force would suggest the onset of surface penetration at which point procedures for releasing such a node are implemented.

Distribution of slave mass to the master segment. Mass distribution is done in three different ways:

- (i) if a slave segment is located wholly on a master segment, half of its mass is distributed to each of the two nodes that define that master segment,
- (ii) if one slave node is in contact with a master segment half the mass lying between the contacting node and the centre of the slave segment is distributed to the nearest corresponding master node, and
- (iii) if no slave node is in contact no slave mass is distributed to the master segment.

This is an important step in that conservation of mass is checked at every step.

Momentum calculation. Not only is it important to ascertain that mass is conserved, but because in slide line analysis impact conditions are encountered, momentum has to be conserved as well. Thus, if a slave node reached the master segment in the preceding time step, interface conditions are applied to determine post-impact conditions for the master segment in the current time step.

Slave zone stress and master surface nodal accelerations. Knowing the states of stress in the slave zones a distributed pressure is computed by interpolation for each master segment. The change in nodal accelerations is computed and nodal accelerations for the master surface are updated at the same time.

Slave mass and master surface nodal accelerations.

Global acceleration components of each master node are resolved into components normal and tangential to the master line. The normal components are multiplied by a z -factor and transformed back to the global system. The z -factor is computed as:

$$z = \frac{M_l}{M_l^*}$$

where M_l is the master nodal mass and M_l^* is the master nodal mass plus the distributed slave zone mass.

Slave nodal accelerations and vectors. For each slave node in contact with a master segment nodal accelerations and velocities normal and tangential to the master line are computed. With these, global components are readily computed by use of an appropriate transformation matrix.

Time step calculations. For each slave node not in contact with the master line in the current time step, a check for potential contact in the next time step is done. If the potential exists, a time step value is computed such that the slave node may reach the master line but not penetrate it.

3.5 Test Problems.

3.5.1 Rigid Body Dynamics.

For the Newmark Algorithm acceleration is constant over a time step, that is

$$\ddot{x} = a \quad (3.22)$$

For a particle under constant acceleration the expression for the distance covered over a time interval may be computed by integrating equation 3.22 twice with respect to time:

$$x = \frac{at^2}{2} + \dot{x}_0 t + x_0 \quad (3.23)$$

Often initial conditions x_0 , and $\dot{x}_0$ are zero thus equation 3.23 may be rewritten as follows.

$$x = \frac{at^2}{2} \quad (3.24)$$

The first test of the Finite Element code for the Newmark algorithm would be to reproduce the analytical solution of a particle under constant acceleration. A test problem, figure 3.1, of a single 4-node quadrilateral was subjected to a constant unit x-acceleration over a number of time steps gave the exact response as the analytical method. A one element test problem, figure 3.2 was subjected to a combined x- and y-acceleration and again the exact solution was reproduced. A test problem of a combination of six 4-8 node quadrilaterals, figure 3.3 which got divided into a number of individual 4-node subelements, figure 3.4 also gave the exact answer. The creation of subelements is done by the element grading algorithm if the parent element has mid-edge nodes.

3D tests were also done and the results were excellent. No specific test problems for 3D will be described here, however, applications of dynamics analysis to production processes and actual situations involving rigid body and finite strain interactions involving contact interfaces will be described in a later section.

3.5.2 Static-Dynamic Contact.

Some test problems were devised to examine situations in which static and dynamic interactions occur across contact interfaces. One such test problem was of a wheel-rail

interaction, figure 3.5. Two types of interface conditions were examined. First the whole wheel was fixed while the rail was subjected to dynamic nodal body forces, the resistance to shear in the interface being negligible (frictionless conditions). The result was rigid body sliding of the rail as can be seen in figure 3.6. This situation is similar to the sudden braking of a train in which case the wheels are suddenly stopped from rotating but due to momentum wheels slide as a rigid body along the rail.

Secondly, only the centre of the wheel was fixed and the shear resistance in the interface was raised (hence simulating stick conditions) while the rail was again subjected to dynamic nodal body forces. As shown in figure 3.7 the wheel underwent quite a noticeable amount of straining; this is exactly what was expected. This situation would represent the kind of wheel-rail interaction of a train beginning to move from a stationary position.

Another problem examined was one simulating a simple rolling mill, figure 3.8. The middle element which represented the workpiece was subjected to translational nodal body forces and was separated from the outer elements which were fixed completely (representing the dies), by contact interfaces. By specifying different magnitudes of shear terms in the contact interface, it was possible to simulate frictionless interaction whereby the workpiece underwent rigid body translation as seen in figure 3.9. In the case of sticking friction the workpiece underwent significant deformation as seen in figure 3.10.

FIG. 3.1: A ONE ELEMENT DYNAMICS TEST CASE, CONSTANT X-ACCELERATION OVER FIVE TIME STEPS.

FIG. 3.2: A ONE ELEMENT DYNAMICS TEST CASE, CONSTANT
X- AND Y-ACCELERATION OVER FIVE TIME STEPS.

FIG. 3.3: A MULTI-ELEMENT DYNAMICS TEST CASE WITH 4-8 NODE QUADRILATERALS.

FIG. 3.4: SUBELEMENTS (INTEGRATION DOMAINS) FROM FIGURE 3.3
GENERATED DUE TO PRESENCE OF MID-SIDE NODES.

FIG. 3.5: F.E. MESH FOR A SIMPLE WHEEL-RAIL INTERACTION PROBLEM.

FIG. 3.6: FREE-BODY SLIDING OF WHEEL OVER RAIL SIMULATING BRAKING CONDITIONS.

FIG. 3.7: 'STICKING' CONDITIONS IN THE INTERFACE OF WHEEL-RAIL
PROBLEM SIMULATING LOW WHEEL VELOCITY.

FIG 3 8: F.E MESH FOR A SIMPLE TEST PROBLEM OF ROLLING
MILL SIMULATION.

FIG 3.9. 'FRICTIONLESS' CONDITIONS IN CONTACT INTERFACES OF THE ROLLING MILL PROBLEM.

FIG. 3.10: 'STICKING' CONDITIONS IN CONTACT INTERFACES OF THE ROLLING MILL PROBLEM.

4. CONTACT THERMAL ANALYSIS.

4.1 Introduction.

The transient response of many physical field problems can be studied using the quasi-harmonic equation which in three dimensions takes the form:

$$\frac{\partial}{\partial x} \left(k_x \frac{\partial \Theta}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial \Theta}{\partial y} \right) + \frac{\partial}{\partial z} \left(k_z \frac{\partial \Theta}{\partial z} \right) + Q = \rho c \frac{\partial \Theta}{\partial t} \quad (4.1)$$

where Θ is the state variable being studied for the particular problem in which k_x, k_y, k_z, ρ and c are constants associated with the field problem which must be known *a-priori* and supplied as data. Examples of such field problems include heat transfer, gas diffusion, Reynolds' film lubrication, torsion, compressible flow and ground water seepage.^[79]

For transient heat transfer analysis the quasi-harmonic equation is referred to as the heat equation. The thermal conductivities in the x- y- and z-directions, are k_x, k_y, k_z respectively, Θ is the temperature, Q is the heat source and t, ρ and c are the time, density and volumetric specific heat capacity respectively.

The details of the Finite Element approach for seeking the solution of the transient heat equation and for the general field problem are given in standard text books.^{[29][80][81][82]} As well, over the past five years, significant advances have been made in research work in heat transfer related to real production processes such as welding.^{[28][65][66][67][83][84][85]} All the work referred to involved the analysis of bodies of complex (but continuous) geometries. This section of the thesis was aimed at examining real life high temperature manufacturing problems such as welding, casting and rolling. Such processes involve bodies with geometries that include discontinuities (gaps) or bodies that are continuous but interact at finite geometric locations like points, lines and surfaces.

The principle philosophy for solving the problem of discontinuities was to 'fill' the gap with a fictitious medium with constitutive property values that would allow the transmission of thermal loads across the gap. In formulating the ensuing contact thermal elements, and in seeking the solution to the heat equation, special care had to be taken in specifying the constitutive property values and boundary conditions so as to satisfy physical laws of

thermodynamics and heat transfer. In this case it was necessary to be able to compute heat flow through layers of materials of different size and material properties. The gap was modelled as a material with a finite computable resistance to heat flow.

4.2 Thermal Contact Elements.

Contact joint conductance h_j is influenced by a number of parameters such as interface pressure, surface finish, micro-hardness, presence of fluids, oxides or impurities.^{[87][89][90][91]} According to Yovanovich,^[87] joint conductance has contributions from:

1. Solid-solid conductance h_c
2. Gap conductance h_g
3. Radiative conductance h_r

In the simplest model the joint conductance is computed as:

$$h_j = h_c + h_g + h_r \quad (4.2)$$

The radiative heat transfer rate may often be neglected.

4.2.1 Contact Conductance.

The contact conductance coefficient h_c may be defined as:

$$h_c = \frac{Q/A_a}{\Delta T_c} \quad (4.3)$$

Q/A_a is the steady-state heat flux based on the apparent contact area A_a and ΔT_c is the temperature drop at the interface.

The total contact conductance h_c of N contact spots thermally connected in parallel:

$$h_c = \sum_{i=1}^N h_{c,i} = 2k_s \sum_{i=1}^N a_i / \Psi_{c,i} \quad (4.4)$$

$k_s = 2k_1k_2/(k_1 + k_2)$ is the harmonic mean thermal conductivity, a is the mean asperity radius and Ψ_i is the thermal spreading parameter. Detailed geometric analysis is given in his paper^[87] under four main subjects which influence h_c , namely:

- Contact conductance parameter
- Relative-real contact area
- Contact spot density
- Mean contact spot radius

These parameters were combined together to form a single relationship for computing contact conductance as a function of interface pressure P and microhardness H . For type 304 stainless steel the relationship took the form:

$$h_c = 1.25 (P/H)^{0.95} \quad (4.5)$$

Graphically, this result is summarized by figure 4.1. This result was verified by the experimental work of Hegazy.^[88]

4.2.2 Elastogap Conductance Model.

The thermal gap conductance h_g is proportional to the local gap thickness, the thermal conductivity of the gas and the temperature difference. Heat flow across the gap is assumed to be perpendicular to the plane of contact.

The differential gap heat flow rate is computed as:

$$dQ_g = \frac{k_g \Delta T}{\delta} dA \quad (4.6)$$

Where k_g is the local gas conductivity, δ is the local gap thickness and ΔT is the temperature difference. The total heat flow across the gap is computed as:

$$Q = \int_A dQ_g \quad (4.7)$$

The gap thickness δ was not allowed to approach zero. This avoided unbounded conductance and unbounded flux. A more careful analysis of the physics of gap conductance would contain a gap area/thickness ratio, A_g/δ . It is this ratio that is bounded. In other words the gap area must approach zero more quickly than the gap thickness. The use of a gap thickness cutoff value in this thesis, is a crude approximation of this phenomena.

FIG. 4.1: VARIATION OF CONTACT CONDUCTANCE WITH INTERFACE PRESSURE.^[87]

Globally, in two dimensions (2D) the contact thermal element, like the contact element in stress analysis described earlier, is a four-node or six-node quadrilateral (no mid-edge nodes for the vertical edges) as shown in figure 2.2.

It is sandwiched between two regular quadrilateral elements such that nodes 3, 4 (and 6 for the 6-node quadrilateral) also belong to the regular element bounding from the top while nodes 1, 2 (and 3) also belong to the element bounding from the bottom. However at the element level the contact thermal element is a two-node or three-node line element that lies in the mid-plane of the gap.

The choice of shape functions for the 3-node element was the bilinear form rather than the fully quadratic. This is the form that can be used by the subelement creation routine which is the principle feature of graded meshes. Recent studies by Pammer^[92] indicate that not only does the use of graded meshes reduce the cost of computing but actually enhances the accuracy of the Finite Element modelling of a certain class of heat transfer problems, specifically transient temperature fields that develop in bodies that experience sudden temperature changes like quenching of heat treatment specimens. In Pammer's findings, the best accuracy was attained if the element size changes by a size quotient of a geometric progression $r = 2$. Thus as we move from an area of stable thermal transients such as insulated boundaries towards areas of steep transients where the hot specimen surface comes into contact with the low temperature fluid, the mesh gets finer. Contact thermal elements were designed not to have any heat storage capacity, a property that gave them the ability to transmit high thermal gradients without the risk of numerical instability. Grading contact thermal elements were developed so as to be compatible with the regular continuum elements (with finite specific heat capacity) with which they interface. With these it was possible to do a thermal analysis of problems involving solids in contact or solids separated by small air gaps.

Figure 2.3 shows the shape functions for the element.

4.3 Two Dimensional F.E.A.

The following description of the Finite Element formulation of contact thermal elements closely follows the step-by-step computer programme development. Details are

given for the 2-node line element. The 3-node element follows a very similar procedure to the 2-node element so details will not be given here.

Equation 4.3 may be rearranged to take the form:

$$F = C\Delta T \quad (4.8)$$

Where F is the thermal load, C is the stiffness matrix and ΔT is the temperature change.

Algebraically equation 4.8 may be written as:

$$\begin{bmatrix} F_1^e \\ F_2^e \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} \Delta T_1^e \\ \Delta T_2^e \end{bmatrix} \quad (4.9)$$

where F_i^e are the element thermal loads, ΔT_i^e are the relative temperatures across the gap and k_{ij} are the entries of the element stiffness matrix, computed in a volume integral as follows:

$$[K] = \int B^T C B dvol \quad (4.10)$$

where $dvol = \det J d\xi$

$$[B] = [N_1, N_2], \quad N = f(\xi) \quad (4.11)$$

$$[C] = A \begin{bmatrix} h_j & 0 \\ 0 & h_j \end{bmatrix} \quad (4.12)$$

h_j is the joint conductance at the Gauss point.

It would be attractive to solve equation 4.9 for the temperature drop or rise (ΔT) across the contact thermal element with two unknowns to be determined. Nevertheless, it will be necessary to formulate the element stiffness matrix and load vector in terms of the global geometry and hence compute the absolute global nodal temperatures instead (in this case four unknowns to be determined). To do this a transformation matrix G is required such that:

$$\begin{bmatrix} \Delta T_1^e \\ \Delta T_2^e \end{bmatrix} = [G] \begin{bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{bmatrix} \quad (4.13)$$

where:

$$[G] = \begin{bmatrix} -1 & 0 & 0 & 1 \\ 0 & -1 & 1 & 0 \end{bmatrix} \quad (4.14)$$

With the transformation matrix G the volume integral for computing the stiffness matrix becomes:

$$[K] = \int G^T B^T C B G \det J d\xi \quad (4.15)$$

4.4 Three Dimensional F.E.A.

In three dimensions (3D), globally the contact thermal elements are 8-16-node grading bricks (without mid-edge nodes on the vertical edges). At the element level they are 4-8-node grading quadrilaterals, as shown in figure 2.4. Notice that the top face is a mirror image of the bottom face. The element level surface quadrilaterals' topology is computed by averaging the coordinates of corresponding global nodes of the top and bottom faces. For example the element level coordinate of node 4 would be $X^e(4) = X^g(4) + X^g(8)$, similarly for $Y^e(4)$ and $Z^e(4)$.

The Finite Element formulation of 3D elements follows a very similar pattern to 2D, the primary difference is that in 3D the thermal response across surfaces is computed whereas in 2D the thermal response across lines is sought hence arrays are longer and matrices are bigger. As an example, for the global 16-node brick which corresponds to the element level 8-node quadrilateral the shape functions, nodal temperature, and relative temperature arrays respectively are:

$$[B] = [N_1, N_2, N_3, N_4, N_5, N_6, N_7, N_8] \quad (4.16)$$

$$[T]^T = [T_1, T_2, T_3, T_4, T_5, T_6, T_7, T_8] \quad (4.17)$$

$$[\Delta T]^T = [\Delta T_1, \Delta T_2, \Delta T_3, \Delta T_4, \Delta T_5, \Delta T_6, \Delta T_7, \Delta T_8] \quad (4.18)$$

and the C-matrix is:

$$C = A \begin{bmatrix} h_c & 0 & 0 \\ 0 & h_c & 0 \\ 0 & 0 & h_c \end{bmatrix} \quad (4.19)$$

and the G-matrix is:

$$[G] = \begin{bmatrix} 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & -1 \end{bmatrix} \quad (4.20)$$

Thus the stiffness matrix K is 16×16 while the Jacobian transformation matrix is 2×3 compared to the usual 3×3 in 3D analysis. The reason for this is that a 3D brick is mapped onto a 2D quadrilateral in a 3D space (a surface quadrilateral). Thus:

$$[J] = \begin{bmatrix} \frac{dx}{d\xi} & \frac{dy}{d\xi} & \frac{dz}{d\xi} \\ \frac{dx}{d\eta} & \frac{dy}{d\eta} & \frac{dz}{d\eta} \end{bmatrix} \quad (4.21)$$

4.5 Test Problems.

After formulating a Finite Element procedure for computing heat flow across contacts the next step was to test it by using it to solve simple problems that could be verified by analytical heat transfer calculations. The case of unit thermal load through a unit dimension material of unit thermal conductivity producing a unit temperature change was examined. A simple test problem is shown in figure 4.2. It has continuum elements (1 & 3) with thermal conductivity 1 W/mK , separated by a gap element (2). A 1 J thermal load was applied on the top edge while the bottom edge was fixed at 0°C . The thermal conductivity of the fluid in the gap was set at 1 W/mK .

The size of all elements was $1 \text{ m} \times 1 \text{ m}$. The result as expected was a temperature change of 1°C across each element.

The height of the gap element was reduced to $1/10$ the height of the continuum elements. The corresponding result was a temperature change of 0.1°C across the gap, as shown in figure 4.3. Figure 4.4 illustrates that a further reduction of the gap size to $1/100$ gave a temperature change of 0.01°C across the gap.

Similar test problems in 3D were solved and the F.E.A. results were identical to analytical results. Other rigorous tests involving combinations of variable node elements in 2D and 3D in graded meshes were completed successfully thus testing the conditioning of the procedure. Other common problems in heat transfer involving contact were examined to test the physical soundness of the analysis before attempting to solve real industrial problems.

An initial temperature problem of two bodies at different temperatures, separated by a small air gap was attempted. The situation is shown in figure 4.5. The transient temperature fields of the bodies were examined over four time steps and the result according to figure 4.6 was quite logical, in that heat was conducted through the gap and along the cooler body. The same problem was repeated but the hot body was gradually moved to the right. That is, during the first time step the hot body was coupled to the cooler body via the element on the extreme left and in the fourth step it was coupled via the element on the extreme right. The result which is summarized by figure 4.7 is quite realistic in that

as the heat source is moved the thermal response of the cooler body moves accordingly.

A 3D test problem of heat transfer from a high temperature source to a large heat sink via a contact interface, a common situation in arc welding, was examined. A simple coarse mesh showing the initial conditions for such a problem is shown in figure 4.8. The transient temperature field after 0.1 seconds is given in figure 4.9. Notice that the temperature variation in the heat source is linear while in the heat sink it looks roughly quadratic. The mesh was obviously too coarse to reproduce the expected elliptical shaped temperature contours. The mesh was refined, figure 4.10 and the corresponding result, figure 4.11 was remarkably better. These test problems further reinforced confidence in the contact thermal analysis of being able to simulate physical phenomena realistically.

A test problem involving two rods at different temperature separated by a small air gap was devised to investigate the ability of contact thermal analysis to model physical phenomena involving very high thermal gradients. This is the family of problems described by Pammer^[92], figure 4.12. Notice that as we approach the gap separating the rods the element size is halved, hence satisfying the quotient ratio of 2 requirement. The results were quite realistic and close to what was expected. With a low contact thermal conductivity, there was very little heat exchange between the rods. This resulted in a high temperature change across the gap (90°C) as illustrated in figure 4.13. With a higher contact thermal conductivity however, considerably more heat could flow across the gap. This reduced the temperature change across the gap (74°C) as seen in figure 4.14.

At this point the F.E. procedure for contact thermal analysis had given sufficiently good results both in terms of the physical soundness and accuracy to warrant attempts at actual industrial problems.

The first such problem was one of analyzing the heat build-up in a spring loaded electronic circuit connecting clip assembly. The internal and external radii of the semi-circular clip were 0.05 and 0.075 inches respectively. The spring force pressing the clip onto the heat sink ranged between 57 and 114 grammes. Both the clip and heat sink were of a copper-beryllium alloy, but on the surface of the heat sink were 25 and 100 micro-inches of gold and nickel platings respectively. A small sector (22.5°) of the clip was discretized along with a corresponding size heat sink system. In the thermal analysis,

a heat q flux was prescribed on the inside surface of the clip and the temperature of the base of the heat sink was prescribed to 0°C . The rest of the surfaces were insulated as shown in figure 4.15. The analysis was first done with air in the interface, then it was repeated with helium in the interface.

Temperature contours for $f = 57$ grammes with air in the interface are shown in figures 4.16 and 4.17. This represented the worst case because it gave the highest temperature rise per unit operating power. Temperature contours for $f = 114$ grammes with helium in the interface are shown in figures 4.18 and 4.19. This represented the best case because it gave the lowest temperature rise per unit operating power. A graphical summary of temperature rise versus interface force is given in figure 4.20, and temperature rise per watt versus interface force is given in figure 4.21.

With this analysis it is possible to analyze different geometries of interaction to determine the one that gives the least temperature build-up so as to allow such devices to operate at much higher power settings.

The second problem also involved analyzing temperature build-up in electronic circuits but the geometries of interaction were different. In the problem whose dimensions and layout are given by figure 4.22, four transistors were symmetrically clamped onto a heat sink system of an aluminum plate connected to four heat pipes. During operation each transistor releases heat, which had to be transmitted across the contacting surfaces and thin air gap to the heat sink system. Heat extraction is crucial to the operation of transistors since heat build up can destroy them or the epoxy resins used to connect electronic components to the circuit board.

The r.m.s. surface finish of the mating surfaces was 125 micro-inches. One quarter of the problem (one transistor and associated heat sink) was discretized. The mesh giving stress analysis boundary conditions is given in figure 4.23 and the thermal analysis boundary conditions are given in figure 4.24. A range of tensions of 3.648 – 29.168 newtons was used for the analysis.

The resulting temperature contours for the least interface force are shown in figure 4.25, and for the highest interface force in figure 4.26. Results are summarized graphically in figure 4.27.

The significance of the results, was, as expected, that heat conduction across contacts improves with interface pressure. With this analysis it is possible to recommend accurately the tightening tension of the assembly bolts that predetermine the operating capacity of the device.

```

OUTPUT FOR ELEMENT NO. 3
  For right hand side no. 1
  Node No. 6 5 7 8
  Tempera. 2.000 2.000 3.000 3.000
OUTPUT FOR ELEMENT NO. 2
  For right hand side no. 1
  Node No. 4 3 5 6
  Tempera. 1.000 1.000 2.000 2.000
OUTPUT FOR ELEMENT NO. 1
  For right hand side no. 1
  Node No. 1 2 3 4
  Tempera. 0.000 0.000 1.000 1.000
Max. temperature change at any node = 0.000 in iteration 2.

```

FIG. 4.2: A THREE ELEMENT CONTACT THERMAL ANALYSIS TEST CASE
WITH GAP SIZE = 1

FIG. 4.8: COARSE F.E. MESH FOR A MOLTEN DROP TEST PROBLEM.

OUTPUT FOR ELEMENT NO. 3

83

For right hand side no. 1

Node No. 6 5 7 8
Tempera. 1.100 1.100 2.100 2.100

OUTPUT FOR ELEMENT NO. 2

For right hand side no. 1

Node No. 4 3 5 6
Tempera. 1.000 1.000 1.100 1.100

OUTPUT FOR ELEMENT NO. 1

For right hand side no. 1

Node No. 1 2 3 4
Tempera. 0.000 0.000 1.000 1.000

Max. temperature change at any node = 0.000 in iteration 2.

FIG. 4.3: A THREE ELEMENT CONTACT THERMAL ANALYSIS TEST CASE
WITH GAP SIZE = .1

```

-OUTPUT FOR ELEMENT NO.      3
  For right hand side no.    1
  Node No.                   6      5      7      8
  Tempera.                   1.010  1.010  2.010  2.010
OUTPUT FOR ELEMENT NO.      2
  For right hand side no.    1
  Node No.                   4      3      5      6
  Tempera.                   1.000  1.000  1.010  1.010
OUTPUT FOR ELEMENT NO.      1
  For right hand side no.    1
  Node No.                   1      2      3      4
  Tempera.                   0.000  0.000  1.000  1.000
Max. temperature change at any node = 0.000 in iteration 2.

```

FIG. 4.4: A THREE ELEMENT CONTACT THERMAL ANALYSIS TEST CASE
WITH GAP SIZE = .01

FIG. 4.5: F.E. MESH FOR AN INITIAL TEMPERATURE TEST PROBLEM
WITH AN AIR GAP.

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.214E+02 AT NODE 10
 MAX= 0.713E+02 AT NODE 14

CONTOUR VALUES
 A= 0.214E+02
 B= 0.314E+02
 C= 0.414E+02
 D= 0.514E+02
 E= 0.614E+02
 F= 0.713E+02

TIME = 0.2000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.341E+02 AT NODE 10
 MAX= 0.541E+02 AT NODE 14

CONTOUR VALUES
 A= 0.341E+02
 B= 0.441E+02
 C= 0.541E+02

TIME = 0.3000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.278E+02 AT NODE 10
 MAX= 0.453E+02 AT NODE 14

CONTOUR VALUES
 A= 0.278E+02
 B= 0.378E+02
 C= 0.453E+02
 D= 0.507E+02
 E= 0.571E+02
 F= 0.635E+02
 G= 0.699E+02
 H= 0.763E+02
 I= 0.827E+02

TIME = 0.4000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.233E+02 AT NODE 10
 MAX= 0.426E+02 AT NODE 14

CONTOUR VALUES
 A= 0.233E+02
 B= 0.333E+02
 C= 0.426E+02
 D= 0.526E+02
 E= 0.626E+02
 F= 0.726E+02
 G= 0.826E+02
 H= 0.926E+02
 I= 1.026E+02

FIG. 4.6: TEMPERATURE CONTOURS FOR THE INITIAL TEMPERATURE TEST PROBLEM OVER 4 TIME INCREMENTS.

TIME - 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN- 0.215E+02 AT NODE 10
 MAX- 0.715E+02 AT NODE 14

TIME - 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN- 0.225E+02 AT NODE 13
 MAX- 0.775E+02 AT NODE 14

TIME - 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN- 0.415E+02 AT NODE 10
 MAX- 0.915E+02 AT NODE 14

TIME - 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN- 0.205E+02 AT NODE 10
 MAX- 0.675E+02 AT NODE 14

FIG. 4.7: TEMPERATURE CONTOURS FOR THE INITIAL TEMPERATURE TEST PROBLEM, THE HEAT SOURCE MOVED AFTER EACH STEP.

FIG. 4.8: COARSE F.E. MESH FOR A MOLTEN DROP TEST PROBLEM.

TIME = 0.1000E+00
 CONTOURS OF TEMPERATURE
 MIN= 0.221E+02 AT NODE 17
 MAX= 0.383E+02 AT NODE 26

CONTOUR VALUES
 A= 2.37E+01
 B= 2.54E+01
 C= 2.70E+01
 D= 2.86E+01
 E= 3.02E+01
 F= 3.18E+01
 G= 3.35E+01
 H= 3.51E+01
 I= 3.67E+01

FIG. 4.9: TEMPERATURE CONTOURS AFTER 0.1 SECONDS WITH THE
 TWO SECTIONS INITIALLY BEING AT 100 AND 20°C.

FIG. 4.10: FINER F.E. MESH FOR A MOLTEN DROP TEST PROBLEM.

TIME = 0.1000E+00
 CONTOURS OF TEMPERATURE
 MIN= 0.221E+02 AT NODE 52
 MAX= 0.432E+02 AT NODE 40

CONTOUR VALUES
 A= 2.42E+01
 B= 2.63E+01
 C= 2.84E+01
 D= 3.05E+01
 E= 3.26E+01
 F= 3.48E+01
 G= 3.69E+01
 H= 3.90E+01
 I= 4.11E+01

FIG. 4.11: TEMPERATURE CONTOURS FOR THE FINER MESH AFTER 0.1 SECONDS, SIMILAR CONDITIONS THE COARSE MESH.

TIME = 0.10000E+01

FIG. 4.12: F.E. MESH FOR PAMMER'S PROBLEM.

TIME = 0.10000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.545E+01 AT NODE 46
 MAX= 0.946E+02 AT NODE 29

CONTOUR VALUES
 A= 1.44E+01
 B= 2.33E+01
 C= 3.22E+01
 D= 4.11E+01
 E= 5.00E+01
 F= 5.89E+01
 G= 6.78E+01
 H= 7.67E+01
 I= 8.56E+01

FIG. 4.13: TEMPERATURE CONTOURS FOR PAMMER'S PROBLEM WITH
 "LOW CONDUCTANCE IN THE GAP"

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.145E+02 AT NODE 46
 MAX= 0.855E+02 AT NODE 29

CONTOUR VALUES
 A= 2.16E+01
 B= 2.87E+01
 C= 3.58E+01
 D= 4.29E+01
 E= 5.00E+01
 F= 5.71E+01
 G= 6.42E+01
 H= 7.13E+01
 I= 7.84E+01

FIG. 4.14: TEMPERATURE CONTOURS FOR PAMMER'S PROBLEM WITH A HIGHER CONDUCTANCE IN THE GAP.

FIG. 4.15: F.E. MESH FOR THE ELECTRONIC CONNECTING CLIP PROBLEM
SHOWING THERMAL ANALYSIS BOUNDARY CONDITIONS.

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 90
 MAX= 0.329E+02 AT NODE 33

CONTOUR VALUES
 A= 3.10E+01
 B= 3.13E+01
 C= 3.15E+01
 D= 3.18E+01
 E= 3.20E+01
 F= 3.23E+01
 G= 3.25E+01
 H= 3.28E+01
 I= 3.30E+01

FIG. 4.16: TEMPERATURE CONTOURS IN THE CONNECTING CLIP, AIR IN THE INTERFACE, TOTAL FORCE ON CLIP = 57 GRAMMES.

TIME = 0.10000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 90
 MAX= 0.329E+02 AT NODE 33

CONTOUR VALUES
 A= 0.00E+00
 B= 1.25E-01
 C= 2.50E-01
 D= 3.75E-01
 E= 5.00E-01
 F= 6.25E-01
 G= 7.50E-01
 H= 8.75E-01
 I= 1.00E+00

FIG. 4.17: TEMPERATURE CONTOURS IN THE HEAT SINK, AIR IN THE INTERFACE, TOTAL FORCE ON CLIP = 57 GRAMMES.

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 90
 MAX= 0.165E+02 AT NODE 33

CONTOUR VALUES
 A= 1.50E+01
 B= 1.53E+01
 C= 1.55E+01
 D= 1.58E+01
 E= 1.60E+01
 F= 1.63E+01
 G= 1.65E+01
 H= 1.68E+01
 I= 1.70E+01

FIG. 4.18: TEMPERATURE CONTOURS IN THE CONNECTING-CLIP, H_c
 IN THE INTERFACE, TOTAL FORCE ON CLIP = 114 GRAMMES.

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 90
 MAX= 0.165E+02 AT NODE 33

CONTOUR VALUES
 A= 0.00E+00
 B= 1.25E-01
 C= 2.50E-01
 D= 3.75E-01
 E= 5.00E-01
 F= 6.25E-01
 G= 7.50E-01
 H= 8.75E-01
 I= 1.00E+00

FIG. 4.19: TEMPERATURE CONTOURS IN THE HEAT SINK, HELIUM
 IN THE INTERFACE, TOTAL FORCE ON CLIP = 114 GRAMMES.

FIG. 4.20: TEMPERATURE CHANGE ACROSS THE GAP VERSUS INTERFACE
FORCE FOR THE ELECTRONIC CONNECTING CLIP PROBLEM.

FIG. 4.21: TEMPERATURE CHANGE PER WATT VERSUS INTERFACE FORCE FOR THE ELECTRONIC CONNECTING CLIP PROBLEM.

FIG. 4.22: DIMENSIONS AND LAYOUT OF THE TRANSISTOR PROBLEM.

FIG. 4.23: F.E. MESH FOR THE TRANSISTOR PROBLEM SHOWING

STRESS ANALYSIS BOUNDARY CONDITIONS:

FIG. 4.24: F.E. MESH FOR THE TRANSISTOR PROBLEM SHOWING
THERMAL ANALYSIS BOUNDARY CONDITIONS.

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 168
 MAX= 0.626E+01 AT NODE 232

CONTOUR VALUES
 A= 0.00E+00
 B= 3.75E-02
 C= 7.50E-02
 D= 1.13E-01
 E= 1.50E-01
 F= 1.88E-01
 G= 2.25E-01
 H= 2.62E-01
 I= 3.00E-01

FIG. 4.25: TEMPERATURE CONTOURS IN THE HEAT SINK OF THE
 TRANSISTOR PROBLEM, INTERFACE FORCE = 3.648 N.

TIME = 0.10000E+01
 CONTOURS OF TEMPERATURE
 MIN= 0.000E+00 AT NODE 168
 MAX= 0.400E+01 AT NODE 237

CONTOUR VALUES
 A= 0.00E+00
 B= 3.75E-02
 C= 7.50E-02
 D= 1.13E-01
 E= 1.50E-01
 F= 1.88E-01
 G= 2.25E-01
 H= 2.62E-01
 I= 3.00E-01

FIG. 4.26: TEMPERATURE CONTOURS IN THE HEAT SINK OF THE
 TRANSISTOR PROBLEM, INTERFACE FORCE = 29.168 N.

FIG. 4.27: TEMPERATURE CHANGE PER WATT VERSUS INTERFACE
FORCE FOR THE TRANSISTOR PROBLEM.

5. COUPLED CONTACT ANALYSES.

In the preceding chapters, emphasis was placed on the theoretical developments of the Finite Element solutions for Engineering problems in static, dynamic and thermal contact analysis individually. Often, Engineering problems require coupling two or more of these regimes. In strongly coupled situations like static-dynamic contact, analysis is done concurrently. Examples include isothermal forging, rolling, etc. In weakly coupled situations such as welding, analysis is done in successive steps.

5.1 Thermal-Stress Analysis.

To do a full weld analysis it is necessary to have a constitutive model capable of modelling the deformation of a body under combined loading. In a typical welding process the specimen deforms due to thermally induced strains and externally applied constraints such as clamping. The heat affected zone and immediate neighborhood undergoes plastic deformation while generally the far zone undergoes elastic deformations. Clearly a thermal-elasto-plastic constitutive model is necessary.

Here, weld analysis was done in a weakly coupled sense. That is, a thermal analysis was first done to obtain the transient thermal history. With this the temperature dependent mechanical properties such as yield stress, specific heat capacity and coefficient of linear expansion were computed. As well, the transient thermal strains were computed as:

$$\epsilon^{th} = \alpha \Delta T \quad (5.1)$$

α is the coefficient of linear expansion and ΔT is the temperature change in the time step.

Patel^[83] gives a detailed description of the thermal-elasto-plastic constitutive model and algorithm implemented in ASGARD. The finite strain version of the algorithm in ASGARD is given by Oddy.^[86] In this version strains were evaluated at the mid-point of the time step.

Briefly, in the constitutive model, the body undergoes a net strain resulting from a possible number of contributions including elastic, thermal, plastic, etc.:

$$\epsilon^{Tot} = \epsilon^{el} + \epsilon^{th} + \epsilon^{pl} + \dots \quad (5.2)$$

The elastic strain is expressed in terms of all the other strains:

$$\epsilon^{el} = \epsilon^{Tot} - \epsilon^{th} - \epsilon^{pl} - \dots \quad (5.3)$$

The stress in the body due to the elastic strain is computed in indicial notation as follows:

$$\sigma_{ij} = D_{ijkl} \epsilon_{kl}^{el} \quad (5.4)$$

D_{ijkl} is the elastic material tensor.

5.2 Thermal-Stress Analysis of Contact Problems.

Coupled analysis of contact problems was done by a similar procedure to that of the ordinary continuum. The geometric treatment of contact problems in heat transfer and stress analysis is similar. So in the thermal analysis phase the element types for contact thermal analysis are specified along with appropriate material properties. In the stress analysis phase, the Finite Element mesh remains the same as that defined earlier so the only changes are the element types and material properties.

A number of simple and verifiable test problems for thermal-stress analysis of contact problems were done. Following these test cases, peening as a form of stress relief in welds was modelled. To simulate this, a thin strip of material was 'placed' on the weld specimen a short while after the weld pass was completed. A compressive load was applied to the weld specimen via the strip and contact interface. This process was repeated for a number of temperature ranges to determine one that would give the best results.

Modelling peening as a form of stress relief in welds was done in three stages. The first was a regular stress analysis to determine the contact surface, size of gap and interface pressure. With this information the thermal conductivity of contact, which is dependent on these surface conditions, was computed. Finally, transient thermal strains were computed and used to compute the stress state in the weld specimen.

5.3 Test Problems.

Coupled thermal-stress analysis involving contact was examined. Many high temperature manufacturing processes like hot rolling, forging, welding etc. may be simulated under this category. A simple 3D test problem devised to examine thermal-stress analysis with contact is shown in figure 5.1. During the first stage of the analysis, a thermal flux q was prescribed on the top face of the test specimen, while the bottom face was fixed at 0°C . In the second stage (stress analysis), the bottom face was constrained in z -direction sufficiently to eliminate rigid body modes, then the test specimen's response to the thermal loading was observed under two different conditions in the interface. Figure 5.2 shows the deformed shape of the specimen with fairly high resistance to shear straining in the x - y plane of the contact interface. Figure 5.3 shows the deformed shape with low resistance to shear straining.

This test revealed that, with high resistance to shear straining, the expansion of the test specimen due to the heating effect was concentrated in the z -direction. With low resistance to shear straining however, expansion occurred evenly in all three directions.

3D versions of the spring-loaded connecting clip in electronic circuits discussed earlier in thermal analysis were examined in more detail and variation. The aim was to explore other shapes of interaction in a coupled analysis.

The smallest temperature rise per unit operating power could be determined by varying the shape of interaction, guided by the thermal analysis portion of the programme. The stress analysis portion gives the necessary information on the state of stress resulting from externally applied loads (springs, bolts, etc.) and thermal loads. Coupled contact analysis can thus give useful information for a more effective design and operation of interacting components.

5.3.1 Cylinder/Plate Interaction.

Figure 5.4 shows the thermal boundary conditions for a 3D coupled thermal-stress analysis of a cylinder/plate interaction. The thermal flux loading on the inner surface of the cylinder was in fact applied on each node on that surface. The temperature of the bottom surface of the plate was fixed at 0°C . Apart from the interacting surfaces,

all others were insulated. In the stress analysis phase, the bottom face of the plate was constrained in the y -direction sufficiently to eliminate rigid body modes and a uniform compressive load was applied on the inner surface of the cylinder. Figure 5.5 shows the resulting temperature contours, from a total thermal flux of 100 W. Figure 5.6 shows the end-view of the temperature contours from figure 5.5. Of the computed stress state only 3 components considered of major significance will be presented, namely the yy -stresses (normal to the gap) as shown in figure 5.7, zz -stresses (cross sectional), figure 5.8, and zx -stresses (shear stresses in the plane of the plate), figure 5.9.

5.3.2 Cylinder/Cylinder Interaction.

Figure 5.10 shows the thermal boundary conditions of a cylinder/cylinder interaction. This was in fact a replica of the cylinder/plate problem both in the thermal and stress boundary conditions. Figures 5.11 and 5.12 show the 3D- and end-views of the temperature contours. Stresses normal to the gap (yy -stresses) are shown in figure 5.13 while figure 5.14 gives the zz -stresses (cross sectional). Shear stresses in the plane tangent to the cylinders are shown in figure 5.15.

5.3.3 Sphere/Plate Interaction.

A similar problem to the preceding two, involving the interaction of a sphere and a plate, was attempted. The thermal analysis indicated that very little heat was transferred from the sphere to the plate. This is due to the fact that a sphere and a plate interact at a point, with the contact area growing into a circle whose size is proportional to the applied load. There was such a high temperature rise per unit operating power that it was useless to use the result for comparison with the preceding two. However, for purposes of completeness, the thermal load was reduced to 10 Watts so as to have reasonable results that could be displayed. Figure 5.16 shows the thermal boundary conditions. 3D- and end-views of the temperature contours are given in figures 5.17 and 5.18 respectively. Shear stresses in the plane of the plate are given in figure 5.19. The rest of the stress components will not be presented since, due to the complex geometry of the sphere, it would require intricate sectioning to display stress contours in an easy-to-visualize manner.

5.3.4 Weld Analysis

In production processes such as welding, the undesirable result is residual stress and strain. A number of methods have been developed to make welding processes more efficient, with minimum residual stresses in the final product. Since practically it is impossible to develop a welding process that yields stress-free products, methods of relieving stress have been devised, among them peening. In peening, hammer blows are applied to the welded area a little while after the weld pass is completed.

A very ambitious application of coupled thermal-stress analysis in contact was to attempt to model stress relief procedures in weld specimens by peening. In devising the test problem it was proposed to have a thin strip of material, possibly copper, that was placed on top of the weld specimen. Peening blows were applied on the strip as tractions rather than directly on to the weld specimen. This produced a smooth weld surface. Two weld specimens were used.

5.3.4.1 Edge Weld on a Plate.

A weld bead was run in the middle of a $4 \times 2 \times 1/2$ inch mild steel plate. Because of symmetry only the half of the plate ($4 \times 2 \times 1/4$) inch was discretized as shown in figure 5.20. In the thermal analysis the weld bead was run from points marked 'START' to 'FINISH'. In the stress analysis phase, stresses in the plate were computed from thermal strains emanating from transient temperature fields, and used as externally applied loads. Figure 5.21 shows the zz -stresses (normal to the thickness of the plate). It also shows the thin strip (stress-free at the present stage) on which peening blows were applied. Figure 5.22 shows the xy -stresses (shear stresses in the plane of the plate) at room temperature. The third stage of the analysis was regular stress analysis in which peening blows were applied on the weld specimen via the copper strip.

The results were remarkable. As figures 5.23 and 5.24 indicate, virtually all the stress was removed from the plate onto the copper strip. In fact both figures 5.23 and 5.24 indicate that the welded plate remained with a low net compressive stress which in some applications is preferred to stress-free situations (which are very hard to achieve anyway).

5.3.4.2 Butt Weld on a Plates.

The second weld analysis test problem examined was a bit more practical in that it modelled butt welding of two plates. Two plates $4 \times 2 \times 1/2$ inches were welded together, since two planes of symmetry existed the same mesh from the preceding edge weld analysis was used. In the stress analysis phase however, boundary conditions had to be specified differently so as to satisfy the symmetry requirements. The F.E. mesh used for butt weld analysis showing a slightly longer copper strip on which peening blows were applied is shown in figure 5.27.

Normal stresses in the thickness direction of the plate at $190^{\circ}\text{C} - 150^{\circ}\text{C}$ are shown by figure 5.25, and shear stresses in the plane of the plate are shown in figure 5.26. After the peening blows were applied; the plate was virtually free of normal stresses in the thickness direction, figure 5.28. Similar results were observed in the plane of the plate, figure 5.29.

Peening was performed at a number of temperature ranges but the one that gave the best results was $100^{\circ}\text{C} - 150^{\circ}\text{C}$. The reason that peening at elevated temperatures works as a form of stress relief is that the compressive stress due to peening is approximately equal to the tensile stresses that can be removed from the specimen upon cooling.

The significance of the results from stress relief analysis in welds by a coupled F.E.A. in contact is quite far reaching. First it can be used to reduce or remove guess work in the magnitude of peening blows, and the temperature range to perform the stress relief procedure for best results.

Secondly, stress relief might be attempted in two stages of the analysis. The copper strip may be used in the thermal analysis as a cold shoe to cool the weld specimen. This should remove some of the excess heat that may be responsible for residual stresses and strains. The advantage of this is that the magnitude of peening blows may be reduced considerably. This particular feature was not studied but its potential is obvious. Care of course must be taken not to cool too much otherwise it might result in very high thermal gradients. The effect of this would be the introduction of very high residual strains (the opposite of what was intended).

FIG. 5.1: THERMAL BOUNDARY CONDITIONS FOR A SIMPLE TEST PROBLEM OF COUPLED CONTACT THERMAL-STRESS ANALYSIS.

TIME = 0.10000E+01

DISP. SCALE FACTOR = 0.500E+02

FIG. 5.2: UNDEFORMED STATE AND FREE EXPANSION OF FIGURE 5.1
WITH HIGH SHEAR RESISTANCE IN THE INTERFACE.

TIME = 0.10000E+01

DISP. SCALE FACTOR = 0.500E+02

FIG. 5.3: UNDEFORMED STATE AND FREE EXPANSION OF FIGURE 5.1
WITH LOW SHEAR RESISTANCE IN THE INTERFACE.

FIG. 5.4: 3D F.E. MESH FOR A CYLINDER/PLATE TEST PROBLEM
IN A COUPLED CONTACT THERMAL-STRESS ANALYSIS.

CONTOUR VALUES

A = $5.86E+00$
 B = $1.17E+01$
 C = $1.76E+01$
 D = $2.34E+01$
 E = $2.93E+01$
 F = $3.52E+01$
 G = $4.10E+01$
 H = $4.69E+01$
 I = $5.27E+01$

FIG. 5.5: TEMPERATURE CONTOURS FOR THE CYLINDER/PLATE PROBLEM.

TIME = 0.10000E+03

CONTOURS OF TEMPERATURE

MIN= 0.000E+00 IN ELEMENT 357

MAX= 5.86E+02 IN ELEMENT 434

CONTOUR VALUES

A= 5.86E+00

B= 1.17E+01

C= 1.76E+01

D= 2.34E+01

E= 2.93E+01

F= 3.52E+01

G= 4.10E+01

H= 4.69E+01

I= 5.27E+01

FIG. 5.6: END VIEW OF TEMPERATURE CONTOURS FOR THE

CYLINDER/PLATE PROBLEM.

TIME = 0.10000E+03

CONTOURS OF $\gamma\gamma$ -STRESS

MIN = -0.118E+05 IN ELEMENT 408

MAX = 0.741E+04 IN ELEMENT 221

CONTOUR VALUES

A = -9.85E+03

B = -7.93E+03

C = -6.01E+03

D = -4.09E+03

E = -2.18E+03

F = -2.59E+02

G = 1.66E+03

H = 3.58E+03

I = 5.49E+03

FIG. 5.7: END VIEW OF $\gamma\gamma$ -STRESSES (NORMAL TO THE GAP)

FOR THE CYLINDER/PLATE PROBLEM.

TIME = 0.1000E+03

CONTOURS OF ZZ-STRESS

MIN = 0.173E+05 IN ELEMENT 2
 MAX = 0.135E+05 IN ELEMENT 221

CONTOUR VALUES
 A = -1.42E+04
 B = -1.11E+04
 C = -8.04E+03
 D = -4.96E+03
 E = -1.88E+03
 F = 1.19E+03
 G = 4.27E+03
 H = 7.35E+03
 I = 1.04E+04

FIG. 5.8: END VIEW OF ZZ-STRESSES (CROSS SECTIONAL)

FOR THE CYLINDER/PLATE PROBLEM.

FIG. 5.9: ZX-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE) FOR THE CYLINDER/PLATE PROBLEM.

FIG. 5.10: 3D F.E. MESH FOR A CYLINDER/CYLINDER TEST PROBLEM
IN A COUPLED CONTACT THERMAL-STRESS ANALYSIS.

FIG. 5.11: TEMPERATURE CONTOURS FOR THE CYLINDER/CYLINDER PROBLEM.

TIME = 0.10000E+03

CONTOURS OF TEMPERATURE

MIN= 0.000E+00 IN ELEMENT 406

MAX= 0.720E+02 IN ELEMENT 472

CONTOUR VALUES

A= 7.20E+00

B= 1.44E+01

C= 2.16E+01

D= 2.88E+01

E= 3.60E+01

F= 4.32E+01

G= 5.04E+01

H= 5.76E+01

I= 6.48E+01

FIG. 5.12: END VIEW OF TEMPERATURE CONTOURS FOR THE CYLINDER/CYLINDER PROBLEM.

TIME = 0.10000E+03
 CONTOURS OF YY-STRESS
 MIN=-0.166E+05 IN ELEMENT 420
 MAX= 0.140E+05 IN ELEMENT 228

CONTOUR VALUES
 A=-1.35E+04
 B=-1.05E+04
 C=-7.42E+03
 D=-4.35E+03
 E=-1.28E+03
 F= 1.78E+03
 G= 4.85E+03
 H= 7.92E+03
 I= 1.10E+04

FIG. 5.13: END VIEW OF YY-STRESSES (NORMAL TO THE GAP)
 FOR THE CYLINDER/CYLINDER PROBLEM.

TIME = 0.1000E+03
 CONTOURS OF ZZ-STRESS
 MIN=-0.173E+05 IN ELEMENT 2
 MAX= 0.153E+05 IN ELEMENT 50

CONTOUR VALUES
 A=-1.40E+04
 B=-1.08E+04
 C=-7.53E+03
 D=-4.27E+03
 E=-1.02E+03
 F= 2.24E+03
 G= 5.50E+03
 H= 8.76E+03
 I= 1.20E+04

FIG. 5.14: END VIEW OF ZZ-STRESSES (CROSS-SECTIONAL)
 FOR THE CYLINDER/CYLINDER PROBLEM.

FIG. 5.15: ZX-STRESSES (SHEAR STRESSES IN THE PLANE TANGENT TO THE CYLINDERS) FOR THE CYLINDER/CYLINDER PROBLEM.

FIG. 5.16: 3D F.E. MESH FOR A SPHERE/PLATE TEST PROBLEM
IN A COUPLED CONTACT-THERMAL-STRESS ANALYSIS.

CONTOUR VALUES

A = 1.70E+00

B = 3.40E+00

C = 5.10E+00

D = 6.80E+00

E = 8.50E+00

F = 1.02E+01

G = 1.19E+01

H = 1.36E+01

I = 1.53E+01

FIG. 5.17 TEMPERATURE CONTOURS FOR THE SPHERE/PLATE PROBLEM.

TIME = 0.1000E+03

CONTOURS OF TEMPERATURE

MIN= 0.000E+00 IN ELEMENT 584

MAX= 0.170E+02 IN ELEMENT 1

CONTOUR VALUES

A= 1.70E+00

B= 3.40E+00

C= 5.10E+00

D= 6.80E+00

E= 8.50E+00

F= 1.02E+01

G= 1.19E+01

H= 1.36E+01

I= 1.53E+01

FIG. 5.18: END VIEW OF TEMPERATURE CONTOURS FOR THE SPHERE/PLATE PROBLEM.

CONTOUR VALUES

$A = -9.31E+02$
 $B = -7.41E+02$
 $C = -5.50E+02$
 $D = -3.60E+02$
 $E = -1.69E+02$
 $F = 2.10E+01$
 $G = 2.11E+02$
 $H = 4.02E+02$
 $I = 5.92E+02$

FIG. 5.19: XY-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE) FOR THE SPHERE/PLATE PROBLEM.

FIG. 5.20: F.E. MESH FOR A $4 \times 2 \times .25$ inch PLATE USED
FOR EDGE- AND BUTT-WELD ANALYSIS.

TIME = 0.10000E+21

CONTOURS OF ZZ-STRESS

MIN=-0.394E+08 IN ELEMENT 342

MAX= 0.112E+09 IN ELEMENT 161

CONTOUR VALUES

A=-2.43E+07

B=-9.15E+06

C= 5.96E+06

D= 2.11E+07

E= 3.62E+07

F= 5.13E+07

G= 6.64E+07

H= 8.15E+07

I= 9.67E+07

FIG. 5.21: ZZ-STRESSES (NORMAL TO THE THICKNESS DIRECTION)

AFTER THE EDGE WELD PASS & COOLING TO R. T.

TIME = 0.10000E+21

CONTOURS OF XY-STRESS

MIN=-0.145E+08 IN ELEMENT 343

MAX= 0.728E+07 IN ELEMENT 939

CONTOUR VALUES

A=-1.24E+07

B=-1.02E+07

C=-8.00E+06

D=-5.82E+06

E=-3.64E+06

F=-1.45E+06

G= 7.28E+05

H= 2.91E+06

I= 5.09E+06

FIG. 5.22: XY-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE)

AFTER THE EDGE-WELD PASS & COOLING TO ROOM TEMP.

TIME = 0.1000E+21
 CONTOURS OF ZZ-STRESS
 MIN=-0.256E+09 IN ELEMENT 383
 MAX= 0.256E+09 IN ELEMENT 450

CONTOUR VALUES
 A= -2.04E+08
 B= -1.53E+08
 C= -1.02E+08
 D= -5.09E+07
 E= 3.02E+05
 F= 5.15E+07
 G= 1.03E+08
 H= 1.54E+08
 I= 2.05E+08

FIG. 5.23: ZZ-STRESSES (NORMAL TO THE THICKNESS DIRECTION)
 AFTER PEENING AT ROOM TEMPERATURE.

TIME = 0.10000E+21
 CONTOURS OF XY-STRESS
 MIN=-0.232E+09 IN ELEMENT 427
 MAX= 0.232E+09 IN ELEMENT 393

CONTOUR VALUES
 A=-1.86E+08
 B=-1.39E+08
 C=-9.31E+07
 D=-4.67E+07
 E=-2.97E+05
 F= 4.61E+07
 G= 9.25E+07
 H= 1.39E+08
 I= 1.85E+08

FIG. 5.24: XY-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE)
 AFTER PEENING AT ROOM TEMPERATURE.

TIME = 0.10000E+21
 CONTOURS OF ZZ-STRESS

MIN=-0.582E+09 IN ELEMENT 262
 MAX= 0.439E+09 IN ELEMENT 136

CONTOUR VALUES

A=-4.80E+08
 B=-3.78E+08
 C=-2.76E+08
 D=-1.74E+08
 E=-7.15E+07
 F= 3.07E+07
 G= 1.33E+08
 H= 2.35E+08
 I= 3.37E+08

FIG. 5.25: ZZ-STRESSES (NORMAL TO THE THICKNESS DIRECTION)

AFTER THE BUTT-WELD PASS & COOLING TO 100 - 150°C.

TIME = 0.10000E+21

CONTOURS OF XY-STRESS

MIN=-0.970E+08 IN ELEMENT 211

MAX= 0.125E+09 IN ELEMENT 225

CONTOUR VALUES

A=-7.48E+07

B=-5.26E+07

C=-3.04E+07

D=-8.16E+06

E= 1.41E+07

F= 3.63E+07

G= 5.85E+07

H= 8.07E+07

I= 1.03E+08

FIG. 5.26: XY-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE)

AFTER THE BUTT-WELD PASS & COOLING TO 100 ~ 150°C.

FIG. 5.27: F.E. MESH USED FOR BUTT-WELD ANALYSIS WITH THE STRIP
ON WHICH PEENING WAS APPLIED.

TIME = 0.1000E+21

CONTOURS OF ZZ-STRESS

MIN=-0.923E+09 IN ELEMENT 492

MAX= 0.106E+10 IN ELEMENT 481

CONTOUR VALUES

A=-7.25E+08

B=-5.27E+08

C=-3.30E+08

D=-1.32E+08

E= 6.63E+07

F= 2.64E+08

G= 4.62E+08

H= 6.60E+08

I= 8.58E+08

FIG. 5.28 ZZ-STRESSES (NORMAL TO THE THICKNESS DIRECTION) AT
ROOM TEMP., PEENING WAS DONE AT 100 - 150°C.

TIME = 0.100005+21

CONTOURS OF XY-STRESS

MIN=-0.301E+09 IN ELEMENT 415

MAX= 0.295E+09 IN ELEMENT 482

CONTOUR VALUES

A=-2.41E+08

B=-1.82E+08

C=-1.22E+08

D=-6.27E+07

E=-3.12E+06

F= 5.65E+07

G= 1.16E+08

H= 1.76E+08

I= 2.35E+08

FIG. 5.29: XY-STRESSES (SHEAR STRESSES IN THE PLANE OF THE PLATE)

AT ROOM TEMP., PEENING WAS DONE AT 100 - 150°C.

6. CONCLUSIONS AND CONTRIBUTIONS.

This thesis was principally concerned with the development of F.E. analytical methods for solving contact problems. In the opening chapter an extensive literature search on this family of Engineering problems was presented. In the second chapter, a F.E. algorithm for contact stress analysis was developed. This code was linked into a large multi-purpose F.E. programme ASGAR with which preliminary test problems were run to establish confidence in the code. Hertz's problem of a cylinder/plate interaction was solved and agreement with the analytical solution was good. The method developed evidently extends the long-standing capability for solving Hertzian type problems with arbitrary geometric shapes and boundary conditions.

Contact problems are generally difficult to solve because of complications introduced by shapes and boundary conditions. When such problems are attempted by numerical methods like the Finite Element Method, graded meshes become necessary because of the localized nature of contact. Without grading ability, a sufficiently accurate solution would require the use of a fine mesh throughout the analysis domain, which implies high computing costs. Problems solved in this thesis demonstrate that the use of grading contact elements alongside grading continuum elements resulted in much lower computing costs without the loss of accuracy or generality.

This thesis presents a coupled thermal-stress analysis of contact problems by the Finite Element Method. Some specific applications of this capability include:-

- Modelling of peening as a stress relief process in welding.
- Modelling heat buildup in electronic circuits. By analyzing different possible shapes of interaction for a particular application, it is possible to determine the interaction that gives the least heat buildup, hence the highest safe operating capacity.
- Modelling the state of stress of electronic circuit connectors that are under combined stress and thermal loading. The significance of this is that, knowing the state of stress it is possible to reduce the guess work in designing and fabricating electronic components.

Results of a preliminary study of the application of appropriate interface conditions in the Computer Aided modelling of production processes such as metal rolling were presented. Likewise, appropriate interface conditions for wheel-rail interaction problems, especially in sliding contact were examined.

6.1 Future Applications of Contact Analysis.

6.1.1 Thermal Analysis.

In a 1986 publication Tekriwal and Mazumder^[93] presented results of a Finite Element modelling of arc welding processes. A F.E.M. programme ABAQUS, was used to successfully predict the size of the heat affected zone (HAZ) and the melt-pool-zone (MPZ). In principle a continuously growing mesh due to the added metal from the moving welding rod was used. The one thing that was not mentioned was how numerical difficulties that must have been faced due to very high thermal gradients between the MPZ and the HAZ were avoided.

The use of a continuously growing layer of contact thermal elements between the MPZ and the HAZ would be very useful in absorbing the high thermal gradients and hence reducing the possibility of numerical difficulties in the solution. This could be examined since the potential exists.

6.1.2 Stress Analysis.

An area which was not examined but whose potential exists with the developed software, is the use of contact interfaces to analyze practical problems involving rolling contact.

Adhesion joints are commonly used as permanent linkages between a variety of Engineering components and units whose main mode of load transmission is tension. Examples include layered wooden structures like plywood and the rubber-derivative and other materials used in conveyor belts. The strength of the adhesion joint in shear is crucial. It has been demonstrated by the problems discussed in this thesis that contact stress analysis by normal compressive load transmission can be done quite accurately. Shear load transmission by the contact analysis developed was not as thoroughly explored. However, in the course of compiling this thesis load transmission by shear has been examined. F.E.A.

results for a double lapped joint under tensile loads compared very well with experimental results. Analysis of other types of adhesion joints is possible and should be investigated.

REFERENCES.

- [1] Y.C. Fung: *A First Course in Continuum Mechanics*, 2nd. Ed. Englewood Cliffs, N.J., Prentice-Hall Inc. 1977.
- [2] J.E. Shigley: *Mechanical Engineering Design*, 3rd. Ed. McGraw-Hill, New York, 1977.
- [3] G.A. Castleberry: *Analyzing Contact Stresses More Accurately*. Machine Design (magazine), April 12, 1984.
- [4] F.B. Seely: *Advanced Mechanics of Materials*, 3rd. Ed., 1978.
- [5] T.W. Crooker: *The Role of Fracture Mechanics in Fatigue Design*. A.S.M.E. Publication No. 76-DE-5, 1977, p. 1-7.
- [6] R.W. Hertzberg: *Deformation and Fracture Mechanics of Engineering Materials* 2nd. Ed. John Wiley and Sons, 1983
- [7] A.P. Boresi, O.M. Sidebottom: *Advanced Mechanics of Materials* 4th. Ed. Chapter 14, John Wiley and Sons, 1985.
- [8] J.J. Kalker: *The Computation of Three-Dimensional Rolling Contact with Dry Friction*. International Journal for Numerical Methods in Engineering, Vol. 14, 1979, p. 1293-1307.
- [9] J.J. Kalker: *A Minimum Principle for the Law of Dry Friction; Part 2: Application to Nonsteady Rolling Elastic Cylinders*. Journal of Applied Mechanics, Transactions of the ASME, 1971, 881-887.
- [10] J.J. Kalker: *Review of Wheel-Rail Contact Theories*. The Winter Annual meeting of the A.S.M.E., Chicago Illinois, Nov.16-21, 1980, p. 77-92.
- [11] B. Paul: *Surface and Subsurface Stresses Associated with Non-Hertzian Wheel-Rail Contact*. Symposium on Contact Mechanics and Wear of Rail/Wheel Systems, Vancouver, B.C., July 6-9, 1982, p. 197-207, p. 93-106.
- [12] B. Paul, J. Hashemi: *Contact Geometry Associated with Arbitrary rail and wheel profiles*. Winter Annual meeting of the A.S.M.E., Chicago Illinois, Nov.16-21, 1980.
- [13] R.E. Goodman, R.L. Taylor, T. Brekke: *A Model for Mechanics of Jointed Rocks*.

Journal of the Soil Mechanics and Foundations Division, ASCE, Vol.94, No SM3, Proceedings Paper 5937. March 1968.

- [14] J. Ghaboussi, E.L. Wilson, J. Isenberg: *Finite Elements for Rock Joints and Interfaces*. Journal of the Soil Mechanics and Foundations Division, ASCE Publications, No. SM10, Oct. 1973, p. 832-848.
- [15] D.N. Buragohain, V.L. Shah: *Curved Interface Elements for Interaction Problems*. Proceedings, International Symposium on Soil Structure Interaction, University of Roorkee, Roorkee India, 1977.
- [16] D.N. Buragohain, V.L. Shah: *Curved Isoparametric Interface Surface Element*. Proceedings, American Society of Civil Engineers Vol. 104, Jan. 1978.
- [17] S. Ramanath, M.C. Shaw: *Role of Abrasive and Workpiece Properties relative to the Surface Temperature and Residual Stress*. Proc. NAMRC No. 14, University of Minnesota, May, 1986.
- [18] T.G. Pearce, K.A. Rose: *Tangential Force-Creepage Relationships in Theory and Practice*. Symposium on Contact Mechanics and Wear of Rail/Wheel Systems, Vancouver, B.C., July 6-9, 1982, p. 183-196.
- [19] R.A. Lindberg: *Processes and Materials of Manufacture 2nd. Ed.* Allyn and Bacon Inc. 1977
- [20] N. Tunca: *Wear Mechanisms of High Speed Roller Bearings and Finite Element Analysis of Contact Stress and Crack Propagation*. Ph. D. Thesis, Technical University of Nova Scotia, Halifax, N.S., July 1985.
- [21] S.C. Lim, M.F. Ashby: *Wear Mechanism Maps*. Acta Metallurgica, Vol. 35, 1987, p. 1-24.
- [22] G. Dumas, C.N. Baronet: *Elasto-plastic indentation of a Half-space by an infinitely long Rigid Circular Cylinder*. International Journal of Mechanical Sciences, Vol. 13, 1971, p. 519-530.
- [23] C. Hardy, C.N. Baronet, G.V. Tordion: *The Elasto-plastic indentation of a Half-space by a Rigid Sphere*. International Journal for Numerical Methods in Engineering, Vol. 3, 1971, p. 451-462.

- [24] Y. Yokouchi, T.W. Cou, I.G. Greenfield: *Elastic-Plastic Analysis of Indentation Damages in Copper: Work Hardening and Residual Stress*. Metallurgical Transactions A, Vol. 14A, Nov. 1983, p. 2415-2421.
- [25] A. Francavilla, O.C. Zienkiewicz: *A Note on the Computation of Elastic Contact Problems*. International Journal for Numerical Methods in Engineering, Vol. 9, 1975, p. 913-924.
- [26] L.E. Wilcox: *An Exact Analytical Method for Calculating Stresses in Bevel and Hypoid Gear Teeth*. Gleason Machine Division, Rochester, New York.
- [27] E.A. Wilson, B. Parsons: *Finite Element Analysis of Elastic Problems using Differential Displacements*. International Journal for Numerical Methods in Engineering, Vol. 2, 1970, p. 387-395.
- [28] M. McDill, J.A. Goldak, A. Oddy, M.J. Bibby: *Isoparametric Quadrilaterals and Hexahedrons for Mesh Grading Algorithms*. Communications in Numerical Methods, Vol. 2, 1986.
- [29] K.J. Bathe *Finite Element Problems in Engineering*. Prentice Hall 1982, chapters 5, 9 and 13.
- [30] A.K. Noor, H.A. Kamel, R.E. Fulton: *Substructuring Techniques, Status and Projections*. International Journal of Computers and Structures, Vol. 8, 1978.
- [31] D.J. Benson, J.O. Hallquist: *A Simple Rigid Body Algorithm for Structural Dynamics Programs*. International Journal for Numerical Methods in Engineering, Vol. 22, March 1986, p. 723-749.
- [32] T.J.R. Hughes: *The Finite Element Method*. Linear Static and Dynamic Finite Element Analysis, Prentice-Hall Inc., N.J., 1987.
- [33] J.O. Hallquist: *A Procedure for the Solution of Finite Deformation, Contact-Impact Problems by the Finite Element Method*. Paper for U.S. Energy Research and Development, University of California, April 1976, p. 2-25.
- [34] S.K. Chan, I.S. Tuba: *A Finite Element Method for Contact Problems of Solid Bodies; Part I: Theory and Validation*. International Journal of Mechanical Sciences, Vol. 13, 1977, p. 615-625.

- [35] T.J.R. Hughes, I. Levit, J. Winget: *An Element by Element Solution Algorithm for Problems of Structural and Solid Mechanics*. Journal of Computer Methods in Applied Mechanics and Engineering, No. 36, 1983, p. 241-256.
- [36] P.J. Yoder, R.G. Whirley: *On the Numerical Implementation of Elasto-plastic Models*. Journal of Applied Mechanics, ASME, No. 84-apm-33, 1984, p. 283-288.
- [37] S.K. Chan, I.S. Tuba: *A Finite Element Method for Contact Problems of Solid Bodies; Part II: Application to Turbine Blade Fastenings*. International Journal of Mechanical Sciences, Vol. 13, 1971, p. 627-639.
- [38] K.P. Oh, S.M. Rohde: *Numerical Solution of the Point Contact problem using the Finite Element Method*. International Journal for Numerical Methods in Engineering, Vol.11, 1977, p. 1507-1518.
- [39] K.J. Bathe, N. Chaudhary: *A Solution Method for Planar and Axisymmetric Contact Problems*. International Journal for Numerical Methods in Engineering, Vol.21, 1985, p. 65-88.
- [40] E.L. Wilson: *Finite Elements for Foundations, Joints and Fluids*. Finite Elements in Geomechanics, chapter 10.
- [41] G.W. Clough, J.M. Duncan: *Finite Element Analysis of Retaining Wall Behavior*. Journal of the Soil Mechanics and Foundations Division, ASCE Publications, No SM12, Dec. 1971.
- [42] R.E. Goodman, C. St.John: *Finite Element Analysis of Discontinuous Rocks*. Numerical Methods in Geotechnical Engineering, McGraw-Hill, New York, 1977.
- [43] R. Gaertner: *Investigation of Plane Elastic Contact Allowing for Friction*. Second National Symposium on Computerized Structural Analysis and Design, George Washington University, Washington D.C., 29-31 March 1976, p. 59-63.
- [44] N.D. Hung, G. de Saxce: *Frictionless Contact of Elastic Bodies by Finite Element and Mathematical Programming Technique*. International Computers and Structures, Vol. 11, 1980.
- [45] N. Okamoto, M. Nakazawa: *Finite Element Incremental Contact Analysis with various Frictional Conditions*. International Journal for Numerical Methods in Engineering,

Vol.14, 1979, p. 337-357.

- [46] T.J. Hughes, R.L. Taylor, J.L. Sackman: *Finite Element Formulation and Solution of Contact-Impact Problems in Continuum Mechanics*. Paper for the National Civil Engineering Laboratory, University of California, Jan. 1975.
- [47] T.J.R. Hughes, R.L. Taylor, J.L. Sackman, A. Curnier, W. Kanoknukulchai: *A Finite Element Method for a Class of Contact-Impact Problems*. Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 8, 1976.
- [48] N. Masden: *Numerically Efficient Procedures for Dynamic Contact Problems*. International Journal for Numerical Methods in Engineering, Vol. 20, 1984, p. 1-14.
- [49] W.H. Enright: *On the Efficient Time Integration of Systems of Second-Order Equations arising in Structural Dynamics*. International Journal for Numerical Methods in Engineering, Vol. 16, 1980, p. 13-18.
- [50] O.C. Zienkiewicz, W.L. Wood, N.W. Hines, R.L. Taylor: *A Unified Set of Single Step Algorithms: Part 1; General Formulation and Applications*. International Journal for Numerical Methods in Engineering, Vol. 20, 1984.
- [51] B. Torstenfelt: *Contact Problems with Friction in General Purpose Finite Element Computer Programs*. International Journal of Computers and Structures, Vol. 16, 1983, p. 487-493.
- [52] E. Zoliti: *A Finite Element Procedure to Time Dependent Contact Analysis*. International Journal of Computers and Structures, Vol.17, 1983, p. 555-561.
- [53] W.L. Wood: *A Unified set of Single Step Algorithms: Part 2; Theory*. International Journal for Numerical Methods in Engineering, Vol. 20, 1984, p. 2303-2309.
- [54] J.T. Oden, G.F. Carey: *Finite Elements*. Special Problems in Solid Mechanics, Volume V, Prentice-Hall Inc., N.J., 1984, p. 158-212.
- [55] J.O. Hallquist: *A Numerical Treatment of Sliding Interfaces and Impact*. Winter Annual Meeting of the ASME, San Francisco California. Dec 1978, p. 117-133.
- [56] J.O. Hallquist, G.L. Goudreau, D.J. Benson: *Sliding Interfaces with Contact-Impact in Large-Scale Lagrangian Computations*. Journal of Computer Methods in Applied Mechanics and Engineering, No. 51, 1985, p. 107-137.

- [57] J.O. Hallquist, G.L. Goudreau: *Recent Developments in Large-Scale Finite Element Lagrangian Hydrocode Technology*. Journal of Computer Methods in Applied Mechanics and Engineering. Vol. 33, 1982, 725-757.
- [58] E.C. Flower, J.O. Hallquist: *Computer Aided simulation of Metal Forming*. Paper prepared for 'American Machinist', presented in a short course at Carleton University, May 1986
- [59] D.R.J. Owen, E. Hinton: *Finite Elements in Plasticity, Theory and Practice*. Pineridge Press, 1980.
- [60] K.J. Bathe, A.P. Cimento: *Some Practical Procedures for the Solution of Nonlinear Finite Element Equations*. Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 22, 1980.
- [61] R.W. Hornbeck: *Numerical Methods*, Quantum Publishers Inc., 1975
- [62] D. Tabor: *Friction - The Present state of our Understanding*. Journal of Lubrication Technology. Volume 103, April, 1981, p. 169-179.
- [63] W.R.D. Wilson: *Friction Modelling in Forging*. Computer Simulation in Materials Science, 1986 ASM Materials Science Seminar, Lake Buena Vista, Florida, October 4-5, 1986, p. 237-267.
- [64] R.B. Haber: *A Mixed Eulerian-Lagrangian Displacement Model for Large Deformation Analysis in Solid Mechanics*. Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 43, 1984, p. 277-292.
- [65] J.A. Goldak, B. Patel, M.J. Bibby, J. Moore: *Computational Weld Mechanics*. Internal Report, Department of Mechanical and Aeronautical Engineering, Carleton University, 1985.
- [66] J.A. Goldak, M. McDill, A. Oddy, M.J. Bibby, R. House, X. Chi: *Computational Heat Transfer for Weld Mechanics*. Advances in Welding Science Technology, TWR '86 Proceedings, Gatlinburg, Tennessee, May 18-22, 1986.
- [67] J.A. Goldak, A. Oddy, M. McDill, R. House: *Progress in Computing Residual Stress and Strain in Welds*. Advances in Welding Science Technology, TWR '86 Proceedings, Gatlinburg, Tennessee, May 18-22, 1986.

- [68] P. Tong, J.N. Rossetos: *Finite Element Method, Basic Technique and Implementation*: M.I.T. Press, Cambridge MAS. 1977, p. 155-159.
- [69] J.F. Nye: *Physical Properties of Crystals, Their Representation by Tensors and Matrices*. Oxford Univ. Press, 1964.
- [70] A.H. Cottrell: *The Mechanical Properties of Matter*. John Wiley and Sons, 1964.
- [71] L. Brillouin: *Wave Propagation in Periodic Materials*. Dover Publications 1946.
- [72] G. Dhatt, G. Touzot: *The Finite Element Method Displayed*. John Wiley and Sons, 1984
- [73] W.K. Liu: *Development of Mixed Time Partition Procedures for Thermal Analysis of Structures*. International Journal for Numerical Methods in Engineering, Vol. 19, 1983, p. 125-140.
- [74] W.K. Liu, T. Belytschko: *Mixed-Time Implicit-Explicit Finite Elements for Transient Analysis*. International Journal of Computers and Structures, Vol. 15, No. 4, 1982, p. 445-450.
- [75] W.K. Liu, T. Belytschko, Y.F. Zhang: *Implementation and Accuracy of Mixed-Time Implicit-Explicit Methods for Structural Dynamics*. International Journal of Computers and Structures, Vol. 19, No. 4, 1984, p. 521-530.
- [76] W.K. Liu: *Mixed Time Integration Methods for Transient Thermal Analysis of Structures*. NASA Contractor Report 172209, Sept. 1983.
- [77] W.K. Liu, Y.F. Zhang, T. Belytschko: *Implementation of Mixed-Time Partition Algorithms for Nonlinear Thermal Analysis of Structures*. Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 48, 1985, p. 1-26.
- [78] W.K. Liu, Y.F. Zhang: *Improvement of Mixed Time Implicit-Explicit Algorithms for Thermal Analysis of Structures*. Journal of Computer Methods in Applied Mechanics and Engineering, No. 37, 1983, p. 207-223.
- [79] E. Hinton, D.R.J. Owen: *An Introduction to Finite Element Computations*. Pineridge Press, 1981.
- [80] O.C. Zienkiewicz; *The Finite Element Method*. McGraw-Hill, London, 1977.

- [81] R.D. Cook: *Concepts and Application of Finite Element Analysis*. 2nd. Ed., Wiley, 1981.
- [82] B. Irons, S. Ahmad: *Techniques of Finite Elements*. John Wiley and Sons, 1984
- [83] B. Patel: *Thermo-Elasto-Plastic Finite Element Analysis of Deformations and Residual Stresses due to Welds*. Ph. D. Thesis, Carleton University, Ottawa Ont. Jan. 1985.
- [84] R. House: *A Finite Element Formulation for Heat Transfer Incorporating Latent Heat Effects*. M.Eng Thesis, Carleton University, 1986.
- [85] L.A. Beresford: *Finite Element Analysis of Three Dimensional Heat Flow in Solids*. M.Eng. Thesis, Carleton University, 1982
- [86] A.S. Oddy: *Three Dimensional, Finite Deformation, Thermal-Elasto-Plastic Finite Element Analysis*. Ph.D. Thesis, Carleton University, Ottawa Ont. Oct. 1987.
- [87] M.M. Yovanovich: *Recent Developments in Thermal Contact, Gap and Joint Conductance Theories and Experiment*. Keynote paper, International Heat Transfer Conference, San Francisco Ca., 1986.
- [88] A.A. Hegazy: *Thermal Joint Conductance of Conforming Rough Surfaces: Effect of Surface Micro-Hardness Variation*. Ph.D. thesis, University of Waterloo, Waterloo, Ontario, Canada, 1985.
- [89] M.G. Cooper, B.B. Mikic, M.M. Yovanovich: *Thermal Contact Conductance*. International Journal of Heat and Mass Transfer, Vol. 12, 1969, p. 279-300.
- [90] B.B. Mikic: *Thermal Contact Conductance; Theoretical Considerations* International Journal for Numerical Methods in Engineering, Vol. 17, 1974, p. 205-214.
- [91] R.G. Miller, L.S. Fletcher: *A Facility for the Measurement of Thermal Contact Conductance*. Proceedings of Southeastern Seminar on Thermal Science SETS, Tulane University, New Orleans La., 1974, p. 263-285.
- [92] Z. Pammer: *A Mesh Refinement Method for Transient Heat Conduction Problems solved by Finite Elements*. International Journal for Numerical Methods in Engineering, Vol. 15, 1980, p. 495-505.

- [93] P. Tekriwal, J. Mazumder: *Finite Element Modelling of Arc Welding Processes*. Advances in Welding Science Technology, TWR '86 Proceedings, Gatlinburg, Tennessee, May 18-22, 1986, p. 71-80.

APPENDIX A

Numerical sensitivity of Finite Element Systems.

The interface of jointed structures often results in quite significant changes in the stiffness of the individual components. Using nonlinear analysis, in the neighborhood of a crack in a plate, as the crack closes, the stiffness value normal to the crack surface becomes very large.^[40] A simple one dimensional truss test case (figure A1) with simple enough coefficients to compute the solution of the resulting simultaneous equations by a hand calculator was used to illustrate this effect.

The joint was assigned a much larger stiffness than the two adjoining elements because it is physically smaller hence stiffer. Equilibrium equations are formulated in terms of the absolute displacements u_1 and u_2 on either side of the joint as follows:

$$P_1 = K_1 u, \quad [K_1] = \begin{bmatrix} 1.5 & 0 \\ 0 & 0 \end{bmatrix} \quad (A.1)$$

$$P_2 = K_2 u, \quad [K_2] = \begin{bmatrix} 0 & 0 \\ 0 & 1.5 \end{bmatrix} \quad (A.2)$$

$$P_3 = K_3 u, \quad [K_3] = \begin{bmatrix} 1000 & -1000 \\ -1000 & 1000 \end{bmatrix} \quad (A.3)$$

Assembling the global nodal equilibrium equations for which the external loads have to balance the internal forces,

$$R = P_1 + P_2 + P_3 = \sum_{i=1}^3 K_i u \quad (A.4)$$

If, say, the external loads are assigned values $R_1 = 0.0$ and $R_2 = 1.0$ then the two equations are as follows:

$$\begin{bmatrix} 1001.5 & -1000 \\ -1000 & 1001.5 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 0.0 \\ 1.0 \end{bmatrix} \quad (A.5)$$

(A): External loads and absolute displacements of the joint.

(B): Free body diagram showing internal nodal forces.

FIG.A1: THREE ELEMENT TRUSS TEST CASE FOR DEMONSTRATING NUMERICAL ILL-CONDITIONING DUE TO SMALL, STIFF JOINT ELEMENTS.

Solving this set of simultaneous equations using simple variable elimination and maintaining five significant figures in the coefficients of the equations, the solution was found to be $u_1 = 0.33309$ and $u_2 = 0.33359$.

The same set of simultaneous equations was solved again but maintaining four significant figures (i.e. dropping .5 from equation A.5):

$$\begin{bmatrix} 1001 & -1000 \\ -1000 & 1001 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 0.0 \\ 1.0 \end{bmatrix} \quad (A.6)$$

and the solution was $u_1 = 0.49975$ and $u_2 = 0.50025$.

Equations A.5 were solved again but only three significant figures were maintained, (in effect neglecting the contribution to the stiffness matrix of the larger elements bounding the gap):

$$\begin{bmatrix} 1000 & -1000 \\ -1000 & 1000 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 0.0 \\ 1.0 \end{bmatrix} \quad (A.7)$$

and the equations became singular.

This illustration of numerical sensitivity was for a simple one-dimensional case using a hand calculator. Modern digital computers use a large number of significant figures, typically 8 or 16. However, in the Finite Element discretization of contact problems there is often the need to use very fine graded meshes near the region of contact, resulting in very small, very stiff elements which can cause numerical ill-conditioning.

The use of relative displacements instead of absolute displacements as degrees of freedom helps to avoid this numerical ill-conditioning. The same three element one dimensional test case was used to demonstrate this.

Let u_1 be the absolute displacement of node 1, and δ the relative displacement of node 2 with respect to node 1. We may thus transform the displacement vector such that:

$$u = Tu^* \quad (A.8)$$

where

$$\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 1.0 & 0.0 \\ 1.0 & 1.0 \end{bmatrix} \begin{bmatrix} u_1 \\ \delta \end{bmatrix} \quad (A.9)$$

The equilibrium equation is likewise transformed in a similar manner.

$$R^* = K^* u^* \quad (A.10)$$

for which

$$R^* = T^T R = \begin{bmatrix} 1.0 & 1.0 \\ 0.0 & 1.0 \end{bmatrix} \begin{bmatrix} 0.0 \\ 1.0 \end{bmatrix} = \begin{bmatrix} 1.0 \\ 1.0 \end{bmatrix} \quad (A.11)$$

and the individual element stiffnesses are likewise transformed as follows:

$$K_1^* = T^T K_1 T = \begin{bmatrix} 1.0 & 1.0 \\ 0.0 & 1.0 \end{bmatrix} \begin{bmatrix} 1.5 & 0.0 \\ 0.0 & 0.0 \end{bmatrix} \begin{bmatrix} 1.0 & 0.0 \\ 1.0 & 1.0 \end{bmatrix} = \begin{bmatrix} 1.5 & 0.0 \\ 0.0 & 0.0 \end{bmatrix} \quad (A.12)$$

$$K_2^* = T^T K_2 T = \begin{bmatrix} 1.0 & 1.0 \\ 0.0 & 1.0 \end{bmatrix} \begin{bmatrix} 0.0 & 0.0 \\ 0.0 & 1.5 \end{bmatrix} \begin{bmatrix} 1.0 & 0.0 \\ 1.0 & 1.0 \end{bmatrix} = \begin{bmatrix} 1.5 & 1.5 \\ 1.5 & 1.5 \end{bmatrix} \quad (A.13)$$

$$K_3^* = T^T K_3 T = \begin{bmatrix} 1.0 & 1.0 \\ 0.0 & 1.0 \end{bmatrix} \begin{bmatrix} 1000 & -1000 \\ -1000 & 1000 \end{bmatrix} \begin{bmatrix} 1.0 & 0.0 \\ 1.0 & 1.0 \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & 1000 \end{bmatrix} \quad (A.14)$$

The global stiffness matrix is assembled:

$$K^* = K_1^* + K_2^* + K_3^* = \begin{bmatrix} 3.0 & 1.5 \\ 1.5 & 1001.5 \end{bmatrix} \quad (A.15)$$

The transformed equilibrium equations become:

$$\begin{bmatrix} 3.0 & 1.5 \\ 1.5 & 1001.5 \end{bmatrix} \begin{bmatrix} u_1 \\ \delta \end{bmatrix} = \begin{bmatrix} 1.0 \\ 1.0 \end{bmatrix} \quad (A.16)$$

The solution for equations A.16 with 5 significant figures maintained in the coefficients of the equations was, $u_1 = 0.33308$, $\delta = 0.0005$. The solution with three significant figures was $u_1 = 0.33311$, $\delta = 0.00067$. Thus no numerical ill-conditioning was encountered when relative displacements were used. It may in fact be generalized that

as $K_3^* \rightarrow \infty$ $u_1 \rightarrow 0.33$ and as $K_3^* \rightarrow \infty$ $\delta \rightarrow 0$

APPENDIX B

Pseudo-code for Contact Stress Analysis.

Pre-processing.

Compute global geometry for contact elements from the global nodal list and coordinates.

Compute by interpolation, nodal coordinates of the mid-plane of the gap.

Determine the number of subdomains for Gaussian quadrature.

Determine the Gauss point weights.

Enter the Gauss loop.

Call the shape function routine.

Create the B-matrix from the shape functions.

Compute the stiffness matrix K in a volume integral;

$$K = \int G^T B^T T D T B G dvol$$

Complete the Gauss loop.

Assemble the load vector.

Compute the solution for the current load increment. - FRONTAL SOLVER

Post-processing.

Compute the internal nodal loads from the current stress state.

$$R = \int B^T \sigma$$

Perform the convergence check. (Is $K \Delta u = F$ to a predetermined tolerance by a chosen convergence norm?)

If, so proceed by writing the solution to the output file and graphics database.

Compute the stress at the end of each increment. This will be used as the initial stress for the next increment.

Proceed to the next increment.

If not, compute the residual load ($\Psi = F - R$) and use it as the applied

load to compute the iterative displacement.

Accumulate iterative displacements into an incremental displacement vector.

Repeat convergence check.

Pseudo-code for Dynamic Analysis.

Pre-processing.

Define the global nodal coordinate vector and element connectivity from input data.

Compute integration parameters for the Newmark Algorithm (A_0 - A_6).

Update the nodal acceleration and velocity vectors.

Determine the number of subdomains for Gaussian quadrature.

Determine the Gauss point weights.

Enter the Gauss loop.

Call the shape function routine.

Create the B-matrix from shape function derivatives.

Compute the stiffness matrix K in a volume integral.

$$K = \int B^T D B \, dvol$$

Complete the Gauss loop.

Compute the lumped mass matrix.

Compute the Effective stiffness matrix.

$$K_{eff} = [K] + A_0 * [MASS] + A_1 * [DAMP]$$

Compute the Effective Load Vector.

$$W1 = A_0 * {}^1U + A_2 * {}^2\dot{U} + A_3 * {}^2\ddot{U}$$

$$W2 = A_1 * {}^1U + A_4 * {}^2\dot{U} + A_5 * {}^2\ddot{U}$$

$$W3 = [MASS] * W1$$

$$W4 = [DAMP] * W2$$

$$EXTLOD = W3 + W4$$

(For first iteration of first increment)

$$EFFLOD = EXTLOD + W3 + W4$$

(For first iteration of each increment)

Compute the solution for the current load increment. - **FRONTAL SOLVER**

Post-processing.

Compute the internal nodal loads from the current stress state.

$$R = \int B^T \sigma$$

Perform the convergence check. (Is $K\Delta u = F$ to a predetermined tolerance by a chosen convergence norm?)

If so, proceed by writing the solution to the output file and graphics database.

Compute the stress at the end of each increment. This will be used as the initial stress for the next increment.

Proceed to the next increment.

If not compute the residual load ($\Psi = F - R$) and use it as the applied load to compute the iterative displacement.

Accumulate iterative displacements into an incremental displacement vector.

Repeat convergence check.

Pseudo-code for Contact Thermal Analysis.

The code structure for contact thermal analysis was quite similar to the structure for stress analysis. Equilibrium and convergence checks were based on the maximum temperature change at a node.

END

250789

FIN

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN = 0.223E+02 AT NODE 14
 MAX = 0.775E+02 AT NODE 14

CONTOUR VALUES
 A = 0.270E+01
 B = 0.340E+01
 C = 0.410E+01
 D = 0.480E+01
 E = 0.550E+01
 F = 0.620E+01
 G = 0.690E+01
 H = 0.760E+01
 I = 0.830E+01

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN = 0.223E+02 AT NODE 14
 MAX = 0.775E+02 AT NODE 14

CONTOUR VALUES
 A = 0.270E+01
 B = 0.340E+01
 C = 0.410E+01
 D = 0.480E+01
 E = 0.550E+01
 F = 0.620E+01
 G = 0.690E+01
 H = 0.760E+01
 I = 0.830E+01

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN = 0.223E+02 AT NODE 14
 MAX = 0.775E+02 AT NODE 14

CONTOUR VALUES
 A = 0.270E+01
 B = 0.340E+01
 C = 0.410E+01
 D = 0.480E+01
 E = 0.550E+01
 F = 0.620E+01
 G = 0.690E+01
 H = 0.760E+01
 I = 0.830E+01

TIME = 0.1000E+01
 CONTOURS OF TEMPERATURE
 MIN = 0.223E+02 AT NODE 14
 MAX = 0.775E+02 AT NODE 14

CONTOUR VALUES
 A = 0.270E+01
 B = 0.340E+01
 C = 0.410E+01
 D = 0.480E+01
 E = 0.550E+01
 F = 0.620E+01
 G = 0.690E+01
 H = 0.760E+01
 I = 0.830E+01

FIG. 4.7: TEMPERATURE CONTOURS FOR THE INITIAL TEMPERATURE TEST PROBLEM, THE HEAT SOURCE MOVED AFTER EACH STEP.