

CRANFIELD UNIVERSITY



Axel LANDENBERGER

**THE COUPLING OF THE FEM AND THE BEM
FOR THE SOLUTION OF ELASTOPLASTICITY
AND CONTACT PROBLEMS**

98

School of Mechanical Engineering

PhD THESIS

CRANFIELD UNIVERSITY

School of Mechanical Engineering

**Department for Turbomachinery and
Engineering Mechanics**



PhD Thesis

Academic Year 1997-1998

Axel LANDENBERGER

**THE COUPLING OF THE FEM AND THE BEM
FOR THE SOLUTION OF ELASTOPLASTICITY
AND CONTACT PROBLEMS**

Supervisor:

Dr. A. M. EL-ZAFRANY

March 1998

This thesis is submitted in candidature for
the degree of Doctor of Philosophy

ABSTRACT

This thesis introduces a method for the coupling of the elastic Boundary Element Method (BEM) and the elastoplastic Finite Element Method (FEM) and the extension of this method to contact problems by using gap finite elements.

A method for the coupling of BEM subregions and FEM subregions is derived which limits the transformation of nodal forces into nodal traction to the interface degrees of freedom. This method is also capable of modelling body forces. Then the coupling procedure is extended to allow for elastoplastic material models in the FEM parts of the structure. Additionally contact conditions are included by defining an extra artificial subregion which consists of gap finite elements. Incremental iterative algorithms are used to overcome the present nonlinearities resulting from plasticity, non-conforming contact and friction.

A computer program based on the developed methods and algorithms was coded in FORTRAN77 and tested on PCs. Patch tests and case studies were run in order to validate the developed package against known analytical solutions or the commercial finite element package ABAQUS. It can be concluded that all the original ideas and methods developed in this work are successful. The coupled FEM/BEM is as accurate and reliable as the pure FEM for elastoplastic analysis, with FEM/BEM having the advantage of minimising the modelling effort. The use of gap elements in an artificial subregion was found to be an ideal and accurate way of including contact conditions into the environment of the elastoplastic coupled FEM/BEM.

ACKNOWLEDGEMENTS

Most of all the author would like to thank Dr. El-Zafrany for his excellent guidance and supervision.

Moreover the author is very grateful to his girl friend Phillippa and his family and friends in Germany who have been a great moral support throughout his work on this project.

LIST OF CONTENTS

ABSTRACT	i
ACKNOWLEDGEMENTS	ii
LIST OF FIGURES	vii
LIST OF TABLES	xii
LIST OF NOTATIONS	xiii
CHAPTER ONE	
1. INTRODUCTION	1
1.1 General introduction	1
1.2 Objectives	2
1.3 Thesis layout	4
1.4 Publications	5
CHAPTER TWO	
2. LITERATURE REVIEW	6
2.1 The coupling of the FEM and the BEM	6
2.1.1 General aspects	6
2.1.2 Direct coupling	11
2.1.3 Coupling from first principles (variational coupling)	18
2.1.4 Iterative coupling	19
2.2 Elastoplastic boundary element analysis	21
2.3 Contact analysis	23
2.3.1 General aspects	23
2.3.2 Gap elements	27
2.4 Conclusions	29
CHAPTER THREE	
3. ELASTICITY PROBLEMS	35
3.1 Governing equations	36

3.2	Finite element theory	37
3.3	Boundary element theory	38
3.3.1	The boundary integral equations of displacements	39
3.3.2	Steps of the subregion Boundary Element Method	41
3.3.3	The treatment of non-smooth boundaries	45

CHAPTER FOUR

4.	THE COUPLING OF THE FEM AND THE BEM	48
4.1	Introduction	48
4.2	Subregional FEM equations	50
4.3	The coupling of the FEM and the BEM equations	52
4.4	The discretisation of the interface	54
4.5	The problem of corners on the interface	55
4.5.1	Smooth interface	55
4.5.2	Interface corner on the external boundary	57
4.5.3	Interface corner on an internal interface	59

CHAPTER FIVE

5.	PLASTICITY PROBLEMS	63
5.1	Theoretical background	63
5.1.1	The yield condition	63
5.1.2	The flow rule	64
5.1.3	The hardening rules	65
5.2	Elastoplastic FEM formulation	67
5.2.1	Generalised equations of equilibrium	67
5.2.2	Linearisation of the non-linear equilibrium equations	69
5.2.3	Plasticity algorithm	71
5.3	Elastoplasticity with coupled FEM/BEM	74

CHAPTER SIX

6.	CONTACT ANALYSIS WITH GAP FINITE ELEMENTS	79
6.1	The node-to-node gap finite element	79
6.1.1	Frictionless contact analysis	79

6.1.2	Frictional contact analysis	80
6.2	The implementation of a gap element subregion into the coupled FEM/BEM package.	84
6.3	Generalised gap element algorithm	86
6.3.1	Incrementation	87
6.3.2	Algorithm for one main increment	88

CHAPTER SEVEN

7.	PROGRAMMING	97
7.1	Modular division	97
7.1.1	Module MAIN	97
7.1.2	Module FEM	98
7.1.3	Module BEM	98
7.1.4	Module DATA	99
7.1.5	Module ELEMENTS	99
7.1.6	Module SUBOP	100
7.1.7	Module GAP	100
7.1.8	Module INSTRESS and module INSTRAIN	101
7.2	Program structure	102
7.2.1	Main structure	102
7.2.2	Contact iteration loop	103

CHAPTER EIGHT

8.	RESULTS AND DISCUSSION	108
8.1	Patch test with FEM-, BEM- and/or gap-element-subregions	108
8.1.1	Patch test with FEM- and BEM-subregions	108
8.1.2	Patch test with FEM-, BEM- and gap element subregions	109
8.1.3	Patch test with FEM- and BEM-subregions and an interface corner	110
8.2	Elastoplastic case studies with FEM- and BEM-subregions	111
8.2.1	Pressurised cylinder	111
8.2.2	Rotating circular disc	112
8.2.3	Square plate with a central hole under biaxial tension	113

8.3	Elastic contact case studies with BEM- and gap element subregions	114
8.3.1	Frictional punch on a large rigid rectangular body	114
8.3.2	Frictionless contact of a cylinder and a large rectangular body.	116
8.4	Elastoplastic contact case studies with FEM-, BEM- and gap-element-subregions	117
8.4.1	Frictional elastoplastic punch on a large rectangular body	117
8.4.2	Elastoplastic frictionless contact of a cylinder and a large rectangular body	120
 CHAPTER NINE		
9.	CONCLUSIONS	150
REFERENCES AND BIBLIOGRAPHY		154
APPENDIX A	FUNDAMENTAL SOLUTION PARAMETERS	170
APPENDIX B	CORNER MODEL	171
APPENDIX C	DERIVATION OF THE ELEMENTAL TRANSFORMATION MATRIX	174
APPENDIX D	FINITE ELEMENT PLASTICITY PROCEDURES	177
APPENDIX E	GAP ELEMENT ROTATION	184
APPENDIX F	ASSEMBLY OF THE SUBREGION EQUATIONS INTO SOLVABLE STRUCTURAL EQUATIONS	187
APPENDIX G	PACKAGE USER'S GUIDE	189

LIST OF FIGURES

2.1	Typical steps of an FEM procedure or a BEM procedure.	32
2.2	Insertion of the coupling step into the typical course of steps for different methods of coupling.	33
2.3	A structure divided into an FEM subregion and a BEM subregion.	34
3.1	A domain discretised into two boundary elements subregions.	47
3.2	The modelling of a corner with two separate nodes.	47
4.1	A domain which is subdivided into a BEM subregion and a FEM subregion.	61
4.2	Interface for a case without a corner.	61
4.3	Corner modelling of an external interface corner.	62
4.4	Corner modelling of an internal interface corner.	62
5.1	Uniaxial stress-strain diagram.	78
5.2	Approximation of uniaxial plastic material behaviour with analysis using the elastic stiffness matrix and the elastoplastic stiffness matrix.	78
6.1	A node-to-node gap element between two nodes of two FEM- or BEM-subregions.	95
6.2	An artificial gap element subregion between 2 FEM- or BEM subregions.	95
6.3	The discretisation of the artificial gap element subregion into s artificial boundary elements.	96
7.1	Interrelation between the modules which is controlled by module MAIN.	105

7.2	Interrelation between the most important subroutines.	106
7.3	Iterative contact procedure.	107
8.1	Model of the patch test with one FEM subregion and one BEM subregion.	126
8.2	Model of the patch test with one FEM subregion, one BEM subregion and one gap element subregion.	126
8.3	Model of the patch test with an internal interface corner.	127
8.4	Model of the cylinder with coupled FEM/BEM.	128
8.5	Model of the cylinder with FEM.	128
8.6	Radial displacements of the pressurised cylinder.	129
8.7	Radial stress of the pressurised cylinder.	129
8.8	Hoop stress of the pressurised cylinder.	130
8.9	Radial displacement of the rotating disc.	130
8.10	Radial stress of the rotating disc.	131
8.11	Hoop stress of the rotating disc.	131
8.12	Plate with a small hole under biaxial tension.	132
8.13	FEM/BEM model of the plate with a hole.	132
8.14	FEM model of the plate with hole.	133
8.15	Radial displacement of the plate with a hole.	133
8.16	Radial stress of the plate with a hole.	134

8.17	Hoop stress of the plate with a hole.	134
8.18	A rectangular punch on a large rectangular body.	135
8.19	BEM model of the rectangular punch on a rectangular surface.	135
8.20	FEM model of the rectangular punch on a rectangular surface.	136
8.21	Relative tangential displacements of the contacting bodies of the elastic rectangular punch case along the contact area (at $y=0$ mm).	137
8.22	Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.0$.	137
8.23	Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.05$.	138
8.24	Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.2$.	138
8.25	Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.4$.	139
8.26	σ_y along the line of symmetry (at $x=0$ mm) of the surface body of the rectangular punch case for $\mu=0.0$.	139
8.27	A cylindrical body is pressed onto a large rectangular body.	140
8.28	BEM model for the contact of the cylinder and surface.	140
8.29	FEM model for the contact of the cylinder and surface.	141
8.30	Displacement in y-direction of the surface body along the contact area (at $y=0$ mm) for the elastic cylinder/surface case.	141
8.31	Normal contact pressure along the contact area (at $y=0$ mm) of the elastic cylinder/surface case.	142

8.32	σ_y along the axis of symmetry (at $x=0$ mm) of the surface body for the elastic cylinder/surface case.	142
8.33	FEMBEMGAP model of the elastoplastic punch on a surface.	143
8.34	Different subregions defined in the FEMBEMGAP model of the elastoplastic punch.	143
8.35	Displacement in y-direction along the contact area (at $y=0$ mm) of the surface body of the elastoplastic punch case.	144
8.36	Relative tangential displacement of the two contacting bodies along the contact area (at $y=0$ mm) of the elastoplastic punch case.	144
8.37	Normal contact pressure along the contact area (at $y=0$ mm) of the elastoplastic punch case.	145
8.38	Von Mises stress along the contact area (at $y=0$ mm) of the punching body of the elastoplastic punch case.	145
8.39	Von Mises stress along the right edge of the punching body (at $x=1$ mm) of the elastoplastic punch case.	146
8.40	Stress-strain curve of the material of the cylinder of the elastoplastic cylinder/surface case.	146
8.41	FEMBEMGAP model of the elastoplastic cylinder/surface case.	147
8.42	Different subregions defined in the FEMBEMGAP model of the cylinder/surface case.	147
8.43	Displacement in y-direction of the upper edge of the surface body of the elastoplastic cylinder surface case.	148
8.44	Normal contact pressure along the contact area (at $y=0$ mm) of the elastoplastic cylinder/surface case.	148

- | | | |
|------|--|-----|
| 8.45 | Von Mises stress along the contact area of the cylindrical body of the elastoplastic cylinder/surface case. | 149 |
| 8.46 | Von Mises stress along the axis of symmetry (at $x=0\text{mm}$) of the cylindrical body of the elastoplastic cylinder/surface case. | 149 |

LIST OF TABLES

8.1	Comparison of the FEM/BEM and theoretical displacement results at 15 points for the patch cases of sections 8.1.1 and 8.1.2.	122
8.2	Comparison of the FEM/BEM and theoretical stress results at 15 points for the patch cases of sections 8.1.1 and 8.1.2.	123
8.3	Comparison of the FEM/BEM and theoretical displacement results at 19 points for the patch case with corner of section 8.1.3.	124
8.4	Comparison of the FEM/BEM and theoretical stress results at 19 points for the patch case with corner of section 8.1.3.	125

LIST OF NOTATIONS

E	Young's modulus.
f_x, f_y, f_z	Body forces in x-, y- and z-direction respectively.
F_x, F_y, F_z	Point Forces in x-, y- and z-direction respectively.
F_n, F_t	Point forces in normal and tangential direction respectively.
h	Element thickness.
k_n, k_t	Gap element stiffness in normal and tangential direction respectively.
l, m	Directional cosines of the outward normal of the boundary.
$N_1, N_2, N_3, \dots$	Lagrangian shape functions with respect to node 1, 2, 3 respectively.
$Q^{(j)}$	Load ratio of the j th main increment.
r	Distance to the source point.
R_x, R_y, R_z	Unknown reaction forces in x-, y- and z-directions respectively.
T_x, T_y, T_z	Surface tractions in x-, y- and z-directions respectively.
u, v, w	Displacements in x-, y- and z-directions respectively.
u_i, v_i, w_i	Displacements in x-, y- and z-directions of the source point.
u_n, u_t	Displacements in normal and tangential directions.

x, y, z	Cartesian coordinates.
x_i, y_i, z_i	Coordinates of the source point.
α_i	Load ratio of the i th subincrement.
$\gamma_{xy}, \gamma_{xz}, \gamma_{yz}$	Shear strain components.
Γ	Boundary of a body.
$\epsilon_x, \epsilon_y, \epsilon_z$	Normal strain components.
μ	Coefficient of friction.
μ_{sl}	Coefficient of sliding friction.
μ_{st}	Coefficient of sticking friction.
μ_s	Shear modulus.
ν	Poisson's ratio.
$\sigma_x, \sigma_y, \sigma_z$	Normal stress components.
$\sigma_1, \sigma_2, \sigma_3$	Principal stresses.
$\tau_{xy}, \tau_{xz}, \tau_{yz}$	Shear stress components.
Ω	Domain of a body.

CHAPTER 1

INTRODUCTION

1.1 GENERAL INTRODUCTION

There has always been a considerable dispute amongst researchers whether the Finite Element Method (FEM) or the Boundary Element Method (BEM) is the most suitable method for Continuum Mechanics problems. In the FEM the domain of the analysed structure has to be discretised into finite domain elements, whilst in the BEM for many applications it is only necessary to discretise the boundary of the domain into boundary elements. Therefore, it generally appears to be more attractive to use the BEM, as the initial modelling effort is only a fraction of the modelling effort of the FEM.

There might be disadvantages to the BEM-formulation and its application which complicate its use on a commercial basis. But one should always consider that the BEM has some properties which are extremely powerful and interesting. The disadvantage of the often increased CPU-time of the BEM compared to the FEM (Bettess (1981)) is more apparent than real because modern high speed hardware is getting rapidly faster and cheaper. In contrast the labour time is getting more expensive, and with the BEM the labour time associated with modelling and general pre-processing can be reduced drastically.

Generally speaking there are advantages and disadvantages to both methods and each of them is more suitable for certain applications. The main issue should not be to find out which one of the two methods is generally better and then only use this method. It is much more important to recognise the strength of the FEM and the BEM within each specific application and then decide which method to use for that application.

Nevertheless there is also a very large number of engineering applications where the analysed domain could naturally be divided into a number of subdomains, each of which

would suggest the use of either the FEM or the BEM. One of those subdomains could for example have non-linear or inhomogeneous properties, so that the application of the FEM would be advantageous. Other subregions could behave in a linear and homogeneous manner or could be infinite, so that the application of the BEM would be favourable. Whenever such cases occur, it appears to be an ideal solution procedure to analyse each of the separate subregions by the most suitable method, rather than analysing the whole domain by one of the two methods. This can be realised by using coupled FEM/BEM schemes.

Such a coupling of the FEM and the BEM can be very beneficial in elastoplasticity problems. The pure BEM can not deliver the reliability and ease of use of the pure FEM in elastoplastic cases. Moreover elastoplastic BEM formulations rely on a discretisation of the plastic domain zones so that there is no great modelling advantage. Hence the modelling of plastic zones with the FEM and elastic zones with the BEM appears to be an ideal solution procedure. Such a coupled FEM/BEM scheme could provide significant reduction in model sizes compared to pure FEM especially for small plastic zones. A practical example for the efficiency of this method are elastoplastic contact cases. They appear to be particularly suited to coupled FEM/BEM analysis because plasticity often only occurs at small stress concentration zones near the contact area whilst the rest of the contacting structures behaves in an elastic manner.

1.2 OBJECTIVES

It was the aim of this work to establish a method for the coupling of the FEM and the BEM for elastoplastic problems which also allows for the definition of frictional contact conditions. In order to do this an existing elastic BEM package and an existing elastoplastic FEM package which were both coded in FORTRAN77 were to be modified. The main tasks of the research and programming may be summarised as follows:

- (i) A thorough literature review on the coupling of the FEM and the BEM which is a new research topic at Cranfield University. Moreover a survey on recent developments in contact and plasticity with the BEM.
- (ii) To establish a method for the coupling of the FEM and the BEM which should also allow for the definition of body forces.
- (iii) A method for modelling plastic subregions with FEM and elastic subregions with BEM. The effect of the plastic material behaviour in the FEM subregions on the elastic BEM subregions needs to be accounted for.
- (iv) To implement the definition of non-conforming as well as conforming contact conditions between any formation of FEM- and BEM-subregions.
- (v) To create an incremental iterative contact algorithm which changes the contact conditions during the application of the loads. The algorithm should be capable of dealing with friction and plasticity.
- (vi) Validation of the tasks (i)-(v) of the developed program against analytical solutions or the commercial finite element package ABAQUS if there are no analytical solutions available.
- (vii) Comparison of the developed program to pure FEM analysis for contact and plasticity case studies with respect to accuracy and modelling effort.

1.3 THESIS LAYOUT

Chapter 2 contains an extensive literature review on the coupling of the FEM and the BEM as well as a review on recent developments in plasticity and contact analysis with the BEM. Some conclusions on the literature review with respect to this work are summarised.

Chapter 3 outlines some relevant theory on two-dimensional elastic stress analysis problems and their solution by the FEM and the BEM. The boundary element subregion procedure is explained in detail because it was implemented in the developed package.

A method for the coupling of the FEM and the BEM is described in chapter 4 including measures which insure an accurate implementation.

The extension of the coupling procedure of the previous chapter to elastoplastic analysis is presented in chapter 5.

Chapter 6 explains how frictional contact conditions can be implemented in the presented coupled elastoplastic FEM/BEM by defining a gap element subregion. The algorithm which controls the contact conditions during the application of the load is also described in detail.

The programming and computational aspects of the package development are summarised in chapter 7. The interrelations of the main modules and of the main routines of the package are illustrated.

Several case studies have been run on the developed package and the results and discussions of these results are given in chapter 8.

Chapter 9 outlines the main conclusions. Additionally some recommendations for future work are made.

1.4 PUBLICATIONS

The following publications by the author may prove useful to the reader:

Landenberger, A., "Finite element versus boundary element for elastic contact problems using gap finite elements", MSc thesis, Cranfield University, Cranfield, G.B., 1995.

Landenberger, A. and El-Zafrany, A., "Coupling of gap finite elements and boundary elements for elastic contact problems", BEM18, pp. 373-383, Computational Mechanics Publications, Southampton, G.B., 1996.

Landenberger, A. and El-Zafrany, A., "The modelling of smooth and non-smooth finite-element/boundary-element interfaces", EUROBEM 98, Second European Boundary Element Symposium, May 1998, accepted for publication.

Landenberger, A. and El-Zafrany, A., "Boundary element analysis of elastic contact problems using gap finite elements", Computers & Structures, submitted for publication.

CHAPTER 2

LITERATURE REVIEW

There are three topics which are relevant to this work, namely the coupling of the FEM and the BEM, elastoplasticity and contact analysis.

The main emphasis of this literature review is the coupling of the FEM and the BEM for two dimensional stress analysis, since this topic has not been investigated at Cranfield University before and also because it is very much the main motivation for this work. Therefore an extensive review which summarises relevant developments and different methods is given in section 2.1.

The reviews on elastoplasticity and contact problems in sections 2.2 and 2.3 were guided by the findings of previous authors at Cranfield University. It was tried to explain principal methods briefly and to summarise recent relevant developments which were made mainly in the BEM field.

2.1 THE COUPLING OF THE FEM AND THE BEM

2.1.1 General aspects

A combined use of the FEM and the BEM requires some kind of coupling strategy, which enforces equilibrium and compatibility of the whole domain and mostly important along the common interface between the FE-subregion and the BE-subregion.

Zienkiewicz *et al.* (1977) were the first authors who suggested the coupling of the FEM and the BEM for stress analysis problems. The authors emphasised that a practical linking of the two methods should yield symmetric matrices and, moreover, should be capable of standard finite element assembly in conventional programs. First it was shown how to use boundary solutions as a special case of the generalised finite element procedure. Then it was demonstrated how an energy functional can be modified so that its stationary value gives the solution of the governing differential equations in both the finite element region and the boundary solution region.

A conference article by Osias *et al.* (1977) showed that the usefulness of the coupling of the two methods was also recognised by industrial researchers in an early stage. In this work for Pratt and Whitney, USA, constraint equations represented the interface conditions and the governing Boundary Integral Equations. These constraint equations and the finite element stiffness equations formulated the entire problem and a solution could be obtained. The practical application of their method was demonstrated with two case studies on the calculation of crack tip field stress intensity factors.

The continuation of their work by Cruse and Wilson (1978) suggested to couple the FEM and the BEM by representing the interface BIE tractions in terms of finite element interpolation functions. The authors realised soon that this method was not capable of solving problems with significant load transfer across the common interface. Therefore a better scheme was developed by balancing finite element nodal forces and BIE tractions on an element by element basis. This was found to allow vitally improved solutions. The authors highlighted that a set of constraint equations incorporating corner problems may allow a consistent coupling of the FEM and the BIEM along smooth and non-smooth parts of the interface.

Another early application of the FEM-BIEM method was wave scattering . This can be found in Shaw and Falby (1977). The authors used the coupling when studying a scheme for a harbour basin.

All the work mentioned so far represents initial attempts of scientists to establish possibilities for the coupling of Finite Element Methods and Boundary Integral Equation Methods in elastostatics. A great deal of further work has been carried out on this topic in the last two decades and much more sophisticated coupling methods can be found in the more recent literature. The nature of all of this work can be distinguished into three different types:

A. Direct coupling

The equation systems of the FEM-subregions and BEM-subregions are formulated for each of the subregions separately. Then the equations of the FEM subregions have to be transformed into BEM format and all subregional BEM matrices can be assembled into one structural BEM system of equations. Alternatively the BEM-subregion equations can be transformed into FEM format and assembled together with the equations of the FEM-subregions into one structural FEM system of equations (Refer to section 2.1.2).

B. Coupling from first principles or variational coupling

Within this method the variational expressions of FEM-subregions and BEM-subregions are combined, which leads to one variational expression for the whole domain (Refer to section 2.1.3).

C. Iterative coupling

Within iterative coupling procedures initial trial values for the unknown interface forces and tractions are chosen and the results gained for the FEM and BEM subregions lead to a more appropriate choice of the initially chosen values for the next iteration. This procedure is repeated until compatibility and equilibrium conditions are satisfied exactly (Refer to section 2.1.4).

Although the three different methods of coupling may have many similarities, there is a main difference, which is the order of the steps. A typical FEM or BEM course of steps can be illustrated as shown in figure 2.1. For each of the described coupling methods the coupling procedure itself has to be inserted into this typical course of steps so that FEM and BEM can be joined to form one global solution as shown in figure 2.2. It can be seen that the placement of the coupling procedure is different for each method.

In all three types of coupling shown in figure 2.2, the coupling step itself is simply the application of conditions of equilibrium and compatibility to the interface of the FEM- and BEM-subregions. These conditions may be formulated as follows:

- (1) Equilibrium of the forces: The forces of the FEM- and BEM-subregion must be in equilibrium. Therefore forces and tractions of the FEM-subregion must be equal and opposite to forces and tractions of the BE-subregion along the interface.
- (2) Compatibility of displacements: The displacements of the FEM- and BEM-subregions must be fully compatible and continuous. Therefore the displacements along the interface boundary must be equal.

Each of the three coupling types leads to the mentioned advantages compared to a pure FEM or BEM analysis, although one of the methods might be advantageous for specific problems.

In the large amount of work carried out on this topic, the coupling procedures were applied to a great number of practical engineering problems.

One class of problems which were tackled are soil structure interaction problems. In Brebbia and Georgiou (1979) the coupling was used to analyse a finite element plate on a semi infinite soil foundation which was modelled with the BEM, and a similar procedure can be also found in Guarracino *et al.* (1992). Carter and Xiao (1994)

suggested the analysis of a tunnel which was modelled with the FEM and surrounded by an infinite soil media which was modelled with the BEM. Within all these applications the infinite or semi infinite soil was modelled with the BEM, whilst the finite structural part was modelled with the FEM, so that non-linearities of the structure could be considered. Such an application can also be found in Beer and Meek (1981), where the non-linear behaviour of a pillar in a mine opening was analysed.

Graf and Gebre-Giorgis (1986) reported about their experiences with coupled FEM-BEM in their company and demonstrated how they analysed a cylinder head of a tractor and a planet gear carrier with coupled methods. It was concluded that the BEM should be used for chunky parts of the structure and stress concentration problems, whilst the FEM should be used for plate like regions, cyclic symmetry or gap elements.

One area which always attracted special attention of coupling researchers are crack problems. In the early work of Cruse and Wilson (1977) and Osias *et al.* (1977), the BEM was used to analyse the area around the crack. This was done because the BEM can model the stress concentration at the crack tip more accurately than the FEM in elastic problems. The FEM was used to model the remaining sections of the structure, as with FEM the domain properties can be modelled more accurately. This allows for example the analysis of bi-material problems or other inhomogeneous problems. A different approach in crack problems was used by Lu *et al.* (1991) and similarly by Krishnamurthy *et al.* (1993). In their approaches the authors modelled the area around the crack with FEM because of its advantages in treating the geometric and material non-linearities around the crack. The homogeneous and linear remainder of the structure was modelled with the direct BEM. It is highlighted that this method was found to be especially valuable in moving crack fields, since here the non-linearities surrounding the crack could be included without extensive remeshing as with pure FEM analysis.

Beaudoin and Dawson (1995) applied a coupled FEM-BEM program to the analysis of the deformation of a plastic workpiece by an elastic tool. The elastic large tool was modelled with the BEM and the plastic workpiece with the FEM.

FEM-BEM coupling was applied to contact problems by Ezawa and Okamoto (1995). They used their coupling method for the three-dimensional analysis of the contact of a magnetic disk and a slider.

The importance of practical applications of coupled schemes can be seen in many reports and papers, and the examples presented here only represent a small fraction of the practical possibilities.

Altogether it can be said that the coupling of FEM and BEM in Continuum Mechanics is an essential and promising tool for the solution of a wide range of problems. Good general reviews on the coupling of FEM and BEM can also be found in Kelly *et al.* (1979), Banerjee and Butterfield (1981) and more recently Becker (1992).

2.1.2 Direct coupling

Consider a structure divided into two subregions as shown in figure 2.3. One subregion shall be modelled with FEM and the other subregion with BEM. The boundary of the FEM subregion may be defined as Γ_F and the boundary of the BEM subregion as Γ_B . Similarly the domains of the two subregions are defined as Ω_F and Ω_B and the common boundary of the two subregions may be defined as Γ_{FB} . For the FEM subregion a set of equations can be established as follows:

$$\mathbf{K}_F \boldsymbol{\delta}_F = \mathbf{F}_F \quad (2.1)$$

Where $\mathbf{K}_F$ is the subregional finite element stiffness matrix, $\boldsymbol{\delta}_F$ is the nodal subregional displacement vector and $\mathbf{F}_F$ is the nodal subregional force vector.

And for the boundary element subregion:

$$\mathbf{H}_B \boldsymbol{\delta}_B = \mathbf{G}_B \mathbf{T}_B \quad (2.2)$$

Where $\mathbf{H}_B$ and $\mathbf{G}_B$ are subregional boundary element matrices which are derived by applying fundamental solutions (EL-Zafrany(1993)). $\boldsymbol{\delta}_B$ is a nodal subregional displacement vector and $\mathbf{T}_B$ is a nodal subregional vector of tractions.

The term "direct coupling" means that the matrices and vectors as in equation (2.1) are defined for the finite element subregion and as in (2.2) for the boundary element subregion. Then both systems of equations are assembled into one global system of equations, which can be solved and therefore lead to a global solution of the problem.

A big difficulty within this procedure is that the subregional systems of equations as defined in equations (2.1) and (2.2) are not readily compatible to each other. Therefore it becomes necessary to make the subregional equation systems compatible before the assembly into one global system of equations can be carried out. Generally there are two different possibilities for reaching such compatibility:

- (i) Transformation of the subregional FEM system of equations into BEM format.
Then a set of equations with BEM format can be obtained for both subregions and an assembly into global BEM-type matrices is possible.
- (ii) Transformation of the subregional BEM system of equations into FEM format.
Then a set of equations with FEM format can be obtained for both subregions and an assembly into a global FEM-type stiffness matrix is possible.

Although both methods are valid approaches, most workers prefer to use method (ii) because this method leads to a global FEM-type stiffness matrix. Within method (ii) the subregional equations of the BEM subregion can be transformed into an equivalent FEM-type stiffness matrix which can be assembled into the global stiffness matrix as a special large finite element or alternatively as a FEM-subregion. Therefore an implementation of the equivalent BE-stiffness matrix into a standard finite element package is comparably convenient. This appears to be more lucrative as the FEM is more commonly used and also much more established than the BEM. However, it is

important to notice that a full inversion of the $\mathbf{G}_p$ -matrix is necessary in order to evaluate the equivalent BE-stiffness matrix.

In contrast method (i) offers the big advantage that there is no full matrix inversion necessary and it is therefore computationally cheaper. Hence it seems always advisable to use method (i) unless FEM format is much desired.

Brebbia and Georgiou (1979) were the first authors to present the direct FEM-BEM-coupling in elastostatics. In this paper, the implementation of FE- subregions into BE-packages as well as the implementation of BE-subregions into FE-packages were described. The compatibility conditions (1) and (2) of section 2.1.1 were derived by the authors by equating the external energy expended by forces to the external energy expended by tractions due to a virtual displacement. Problems arose for the case of the assembly of equivalent BE-stiffness matrices in FEM equations (method (ii)) as the equivalent BE-stiffness matrix is by nature asymmetric although a stiffness matrix should be symmetric from first principles. This asymmetry is induced by the approximations of the discretisation process and the assumed solution, and it was overcome by the authors by minimising the error of the asymmetric coefficients.

The first coupling of elastic BEM and elastoplastic FEM can be found in Beer and Meek (1981). In this paper the BEM was applied to elastic infinite parts of the analysed structure, whilst the elastoplastic FEM was applied to finite parts of the structure with elastoplastic material behaviour. A direct coupling procedure with a symmetric equivalent BE-stiffness matrix was used to join FE- and BE-matrices of the subregions. This method was successfully applied to a circular opening in viscoplastic soil and also to the study of the elastoplastic behaviour of a pillar in an underground mine. The authors showed that the coupling of elastoplastic FEM and infinite elastic BEM led to slightly more accurate results than the use of a comparable pure FE analysis. However, the solution procedure of the coupling method was found to be significantly slower than a comparable pure FE-analysis.

Hartmann (1981) highlighted that there are main differences in the fundamental beliefs of mathematics and mechanics as far as equivalent BEM-stiffness matrices are concerned. The mechanical engineer believes that any stiffness matrix has to be symmetric and has to fulfil equilibrium and rigid body displacement criteria. This belief clashes with the fact that the BEM-stiffness matrices, which are derived by mathematical Integral Equation Methods, do not fulfil all these requirements. The author showed how these differences occur and drew up ways of how to obtain "mechanically correct" BEM-stiffness matrices which fulfil all the standard stiffness matrix requirements.

However, Tullberg and Bolteus (1982) showed that the "mechanically correct" symmetrised BEM stiffness matrix delivers less accurate results than the original unsymmetric matrix. The authors compared the accuracy of six different types of symmetrised BEM-stiffness-matrices and the original unsymmetrised BEM-stiffness-matrix. The results obtained showed clearly that the accuracy of the unsymmetrised original equivalent stiffness matrix is superior to each of the symmetrised approaches. The authors also concluded that the CPU-time for all BEM-stiffness-matrix methods is much higher than for a comparable pure FE analysis because of the previously mentioned inversion of a full non-symmetric matrix.

Hisatake *et al.* (1983) applied direct coupling procedures to three dimensional elastostatic and viscoelastic problems. The authors demonstrated a special coupling method for structures with geometric symmetry, which can reduce the computational effort for these cases. They applied the coupling to the analysis of a sphere cavern surrounded by an infinite elastic structure, and in a second case study surrounded by an infinite viscoelastic medium. The results obtained are in very good agreement with theoretical results, whilst attempts to model the infinite medium with large finite element regions rather than with BEM could not deliver similar accuracy. They also concluded that the transformation of BE-equations into equivalent FE-equations is advantageous, whenever a large FE-matrix is coupled with a small BE-matrix. In contrast, FE-matrices should be transformed into equivalent BE-matrices whenever large BE-matrices are coupled with small FE-matrices.

A very efficient direct coupling in elastostatics can be found in Beer (1983a). In this new strategy the equation system of the BE subregion was solved for unit values of the unknown tractions of the interface which leads to a linear relationship of nodal displacements and nodal tractions of the interface nodes. Transforming this relationship into terms of nodal forces instead of nodal tractions led to the boundary conditions needed for the solution of the FE-region. This process is also known as static condensation as only the BE-equations defining the linear relationship between displacements and tractions of the common interface are needed to get a global solution. Therefore only the tractions of the interface have to be transformed into forces rather than all the tractions as in previous work. Once the solution for the FE-region is obtained, the FE-results of displacements of the common boundary can be substituted into the remaining BE-equations which leads to the evaluation of the remaining unknowns of the BEM subregion.

Based on Beer (1983a), Li *et al.* (1986) published an improved static condensation method, the so called "bi-condensation". "Bi-condensation" means that not only the BE-subregion is condensed as in Beer (1983a) but also the FE-subregion. Within this method all the equations defining unknown displacements along the interface are extracted from the FE- and BE- equations and assembled into one system of equations. In contrast to Beer (1983a), Li and his co-workers derived their equivalent stiffness matrices of the BE-region in the original asymmetric form. Numerical tests showed that the asymmetric stiffness matrix method delivered generally better results than the symmetric stiffness matrix. The solutions obtained by the symmetric stiffness method turned out to be very poor for discontinuous interface tractions, which may occur at edges and corners.

Graf and Gebre-Giorgis (1986) demonstrated in their paper two very practical methods of the direct coupling of BEM and FEM in elastostatics. In one of these methods they derive a BE-stiffness matrix by applying a unit displacement to a node on the FE-BE-interface, whilst all other displacements on the interface are set to zero. The application of this prescribed displacement vector to the Boundary element subregion leads to traction results on the interface. These traction results can be used to form parts of the

required BE-stiffness matrix. The repetition of this procedure for each displacement degree of freedom on the interface leads to the required BE-stiffness matrix. The big advantage of this method is that the BE-matrices do not have to be inverted to obtain the equivalent stiffness matrix, but it can also be seen that the BE-subregional equations have to be solved several times which is very time consuming.

A procedure which leads to a more accurate symmetric stiffness matrix of the coupled FE-BE structure can be seen in Gangming (1989). Here the symmetrisation of the global stiffness matrix was obtained by defining a set of collocation points on the boundary, which were different to the boundary element nodes. If each of these collocation points is defined as a source point the stiffness matrix will tend to be symmetric with an increasing number of collocation points. The presented method delivered better results than comparable symmetric stiffness matrix methods such as Zienkiewicz *et al.* (1977) and Brebbia and Georgiou (1979).

Cruse and Osias (1991) summarised from previous work that forcing BEM equations into a finite element format by symmetrisation leads to faulty results. Moreover it was concluded that it is not possible to obtain accurate results for non-smooth interfaces if BE-stiffness matrices are applied. For the case of a non-smooth interface the authors suggested to force the FEM equations into boundary element format. Here, interface conditions representing non-smooth interfaces can be defined accurately. These interface conditions can be derived by either traction matching or force matching, and accurate results for non-smooth interfaces could be obtained for both methods.

In Ma and Le (1992), the initially asymmetric stiffness matrix of the coupled FE-BE system was decomposed into the sum of an asymmetric matrix and a symmetric matrix. The solution was then obtained by an iterative procedure, in which the symmetric matrix was used as the stiffness matrix whilst the effect of the asymmetric part was also considered. The results obtained were better than the ones of previous coupling procedures with symmetric equivalent stiffness matrices, such as e.g. Zienkiewicz *et al.* (1977) and Brebbia and Georgiou (1979). It has to be remarked that this attempt needs at least three iterations, and hence the symmetric equations have to be solved three

times. It is therefore questionable if this method results in time savings compared to the still more accurate asymmetric stiffness matrix method.

One of the problem areas of the direct coupling is that the coupled global matrix is sparsely populated in parts associated with finite element contributions and that it is fully populated in parts associated with boundary element contributions. There were no solvers known which could accelerate the solution procedure of such matrices compared to a standard solver so that recently many authors developed special solvers for the direct coupling.

Remacle *et al.* (1993) for example developed a hybrid solver for coupled FEM-BEM matrices. This hybrid solver consisted of a direct solver which solves the symmetric sparse parts of the coupled matrix and of a conjugate gradient solver which solves the unsymmetric fully populated parts of the coupled matrix.

Dostal and Malik (1995) suggested to solve the equations arising from coupled systems by the conjugate gradient method. As mentioned before the assembly of equivalent dense stiffness matrices of BE-subregions and sparse stiffness matrices of FE-subregions leads to a global stiffness matrix which is sparse, but also has dense population in rows and columns associated with the BE-subregions. Therefore the application of frontal or banded solvers become difficult and inefficient. Within the iterative conjugate gradient procedure as presented by the authors, the iterations were reduced to the common interface of subregions by preconditioning of the matrix system. The authors showed that the application of the conjugate gradient method with preconditioning can lead to substantial savings in numerical operations and therefore processing time and it is superior to a banded solver.

Floren and Duddeck (1993) highlighted that the subregion BEM is very well suited for the direct coupling with FEM. The subregion BEM leads to global matrices with a distinct blocked structure which results from the decoupling of the BE-variables of the separate subregions. FE-subregions with BEM-formulation can also be defined as BE-subregions, so that a global system of equations can be established. The matrices of this

system of equations have sparsely populated structures in parts associated with the FE-subregions and blocked structures in parts associated with the BE-subregions. The authors developed a solver which takes the special properties of these matrices into account. The occurring non-zero blocks of the global matrices could be stored in Full Matrix Storage Mode, Compress Matrix Storage Mode or Profile in Skyline Storage Mode. It was shown that this technique leads to substantial savings in CPU time and storage.

2.1.3 Coupling from first principles (variational coupling)

This kind of the coupling of the FEM and the BEM is called "coupling from first principles" as here the two methods are coupled at the initial stage of the derivation of FEM and BEM. Starting from the equations of equilibrium, a weak variational form can be derived for the FEM subregion and another one for the BEM subregion (Lu *et al.* (1991)). It is possible to combine the weak forms of the FE- and BE-subregions by applying the interface conditions for equilibrium and compatibility, so that one global variational weak form is obtained for the whole domain. An appropriate choice of the involved weighting functions in the global variational form leads to a set of equations which can be solved by using boundary elements for the boundary integral terms and finite elements for the domain integral terms.

The main difficulty within variational coupling is to find continuous weighting functions which can eliminate domain integral terms from the boundary element subregion by collocation and additionally create the finite element set of equations in the finite element region.

In Belytschko *et al.* (1989) variational coupling was demonstrated for the Laplace equation, whilst in Belytschko and Lu (1991) and Lu *et al.* (1991) the same procedure was applied to elastostatics. The big advantage of the presented method compared to previously known coupling methods is that the problems resulting from discontinuous tractions of the boundary element solution can be avoided because the tractions along

the interface are eliminated within the process of forming the global variational form. Lu *et al.* (1991) showed that their method delivers better patch test results than the other so far known coupling procedures, and promising crack problem results were also presented.

A variational coupling similar to the one of Belytschko and his co-workers was developed by Krishnamurthy and Raju (1993) for two dimensional potential problems and Krishnamurthy *et al.* (1993) for two dimensional elasticity problems. Because there are different weighting functions used in FE- and BE- subregions and the continuity of the "overall" weighting functions along the common interface must be satisfied, a kind of transition of these functions has to be introduced. In contrast to Belytschko and his co-workers, who simply defined a ramp function, Krishnamurthy and his co-authors defined a transition region between the BE- and FE-subregions. This transition region limits the effect of the BE-fundamental solution to the BE- and transition regions, so that the equations of the FE-subregion can be solved independently.

Gosz and Moran (1993) proceeded a similar variational coupling in a displacement based finite element context. By using this method, stiffness contributions of the boundary elements can be added to an existing finite element package without making any changes to the main program.

2.1.4 Iterative coupling

The direct coupling as well as the variational coupling rely on applying the conditions of compatibility and equilibrium along the interface explicitly. This means that the tractions (forces) of the two subregions along the interface must be equal and opposite and additionally the displacements of both subregions along the interface must be equal. Therefore unknown tractions (forces) and displacements can be eliminated and one solvable system of equations can be obtained and solved.

In contrast to the direct coupling procedures, iterative coupling procedures do not rely on explicit application of the interface conditions before solving the equation system. In iterative coupling trial displacements (or trial tractions (forces)) are applied to the common nodes on the interface of both subregions. Then both subregional systems of equations can be formulated and solved separately according to the applied trial displacements for both the FE-subregion and the BE-subregion. The obtained solution of both subregions is the right one if the results of the unknown tractions (forces) along the interface are in agreement with the interface conditions of equilibrium and compatibility. If the obtained solution is not accurate another iteration has to be run with improved trial displacements. This procedure has to be repeated until an accurate solution is obtained. Similar procedures to that applied in the iterative coupling can be found in established FEM- and BEM-subregion procedures and are known as domain decomposition methods.

Gerstle *et al.* (1992) applied such an iterative method to obtain a solution for the coupled system. The authors then applied a conjugate gradient method which is aimed at minimising the deviations of the force results so that more accurate trial displacements can be applied in the next iteration step. The exact solution is obtained when the deviations of the force results of the FE- and BE-subregion vanish. It was highlighted that the presented method has the big advantage that standard FE- and BE-procedures can be applied to each of the subregions to obtain results for each of the iterations.

Chia-Ching Lin *et al.* (1996) introduced a very similar iterative coupling method to Gerstle *et al.* (1992). However, in contrast to Gerstle *et al.* (1992), Chia-Ching Lin and his co-workers applied relaxation on the applied displacement in order to obtain the displacement used in the next iteration step. The validity of their proposed method was verified with the example problem of a cantilever beam. The authors concluded that iterative coupling can provide the advantage of the solution of two separate matrices, each of which can be solved with an appropriate solver.

2.2 ELASTOPLASTIC BOUNDARY ELEMENT ANALYSIS

Elasto-plasticity represents a non-linear problem which is usually numerically solved with the FEM. However in recognition of the modelling advantage of the BEM, nowadays boundary element researchers have established possibilities of modelling elasto-plasticity by means of the BEM.

In most BEM-plasticity-schemes which are presented in the literature a purely elastic response is computed and the effects of inelastic strains are superimposed in an iterative procedure. Generally the evaluation of the correct internal strain and stress values within this iterative procedure is achieved by applying the inelastic strain as an initial strain (initial strain approach) or as an initial stress (initial stress approach).

Some work on this topic has already been carried out by previous authors at Cranfield University. Husain (1989) was the first one there to investigate boundary elements in elastoplastic problems. The author presented a thorough literature survey on elasto-plastic stress analysis and the BEM for plasticity. The BEM plasticity algorithm in this work was based on the initial stress approach. A major drawback of this as well as most other BEM plasticity algorithms seems to be that domain integrals have to be evaluated in order to apply the effect of the initial internal stress at the boundary nodes. These domain integrals can only be evaluated by discretising the domain into integration cells. Husain found during his work that a simple integration cell method based on averaged cell contributions could not deliver accurate results. Instead it was suggested to use a finite element type of discretisation for the domain integral evaluation which was then found to deliver better accuracy. The author concluded from his experience with the FEM and the BEM, that the FEM is still the more stable and suitable method for plasticity problems.

Based on Husain's work, BEM plasticity algorithms have been used by Al-Edani (1990) and Hinks (1991) for fracture mechanics. Both authors concluded that the BEM plasticity algorithm was not suitable for conditions involving large plastic deformations. Also the required CPU-time of the BEM-plasticity algorithm compared to the FEM-

plasticity algorithm was found to be longer for cases with big plastic zones due to the time required for the domain integral evaluations.

Guzelbey (1992) used an initial strain approach for the solution of elastoplastic contact problems. It was concluded that the developed initial strain algorithm converges for materials with a large degree of hardening much faster than the initial stress algorithms of previous authors at Cranfield. Hence a significant saving in CPU time could be achieved.

Since the literature survey of Husain (1989) a number of papers by international researchers have been published on BEM-plasticity although not as many as in the nineteen-seventies or -eighties. Significant developments were made by Okada and Atluri (1994) for finite and small strain elasto-plasticity and by Israil and Banerjee (1992) in the dynamic elasto-plasticity field.

As far as elastoplastic contact problems with the BEM are concerned, recent developments have been published by Karami (1991) and Huesmann and Kuhn (1995).

A method in which the inelastic strains are accounted for in a symmetric matrix system rather than the standard unsymmetric BE-matrices was presented in Fossa and Nappi (1991).

Within almost all published BEM-plasticity papers it is necessary to discretise the plastic zones of the analysed domain. Hence the advantage of the reduced modelling of BEM compared to FEM is weakened because domain discretisations are actually needed. Therefore some more recent papers use a coupling of plastic FEM and elastic BEM rather than discretising parts of the BEM domain. Such coupled schemes do not need any more modelling effort than a BEM-plasticity algorithm. At the same time they have all the advantages of FEM-plasticity such as good reliability and easy modelling of the non-linear plastic material properties inside the domain. Recent work on elastoplastic coupling of FEM and BEM has been carried out by Wearing and Burstow (1994) who coupled FEM to a special BE-formulation namely the Regular Indirect Discrete Method

(RIDM). The results of some case studies which were presented in this paper were accurate. A coupling of elastoplastic FEM and elastic BEM for dynamic problems can be found in Pavlatos and Beskos (1994).

2.3 CONTACT ANALYSIS

2.3.1 General aspects

A wide range of engineering structures involve the interaction between components which are brought together under external loads. This problem is of enormous practical importance because of mechanical failure, which generally takes place in the stress concentration zones at or near the contact area.

More than one hundred years ago in 1882, the German scientist Heinrich Hertz published his theories on elastic contact problems (Johnson (1987)). Hertz's theory was the grounding work for many other researchers on the subject and is still in common use today. Several researchers derived closed form solutions for specific problems, based on the application of integral equation methods to Hertz's approach (Shtaerman (1949), Galin(1953), Johnson (1987), Hills *et al.* (1993)). A drawback of the Hertz-theory is that it is grounded on the elastic half space theory and is therefore only applicable to bodies with large extensions in comparison to the contact width. It is also restricted to frictionless contact with both mating bodies having parabolic non-conforming profiles. These restrictions limit analytical contact analysis enormously although different kinds of non-Hertzian contact problems are discussed and solved by a number of authors such as Mushkelishvili (1953), Gladwell (1980), Johnson (1987) and most recently Hills *et al.* (1993).

Analytical techniques for the solution of contact problems are only available for some specific contact problems, so that the application of analytical techniques to other more general contact problems is often difficult or even impossible. However, more versatility is offered by numerical methods for continuum mechanics which can be

modified to allow for the definition of contact conditions. It was found that numerical methods are generally more suitable and less restrained than analytical methods to solve this class of problems. Numerical methods are also very well suited to combine contact conditions with other non-linearities, such as plasticity or geometrical non-linearities.

In recent years the FEM and the BEM established themselves as the most commonly used numerical methods for contact mechanics. Generally there are many different ways of applying contact conditions to the FEM and BEM although most of the algorithms suggested in the literature can be distinguished into three different types:

a. Transformation Matrix Method (Reduction of the size of the structural matrices)

In this approach the contact interface displacements and forces of the contacting surfaces are applied as a kind of boundary condition. Hence interface forces and displacements of the two contacting bodies can be condensed by direct elimination and therefore the structural matrix system is reduced in size. This approach is sometimes used in FEM algorithms (e.g. Sachdeva *et al.* (1981)) but represents the most commonly used method in BEM algorithms (e.g. Olukoko *et al.* (1993b) and Olukoko *et al.* (1993c)). It is so popular with the BEM because in the BEM surface tractions are treated as unknowns and therefore the interface equilibrium conditions can be applied directly. In contrast this procedure can get rather complicated when applied to the purely displacement based FEM.

b. Constraint Equation Method (Expansion of the size of the structural matrices)

This method involves the definition of additional equations which define the contact conditions and therefore the size of the structural matrix system increases. These additional equations are usually derived by using Lagrange Multipliers. Although this is the most complicated of the three methods it is often used for large sliding contact and contact between non-aligned meshes and applications can mainly be found in the FEM (e.g. Bathe and Chaudhary (1985)).

c. Penalty Method (Size of the structural matrices remain unchanged)

Within this method penalty functionals are defined which can be added to the original functional to satisfy the contact constraints. In this way the final structural matrices do not change in size since no additional unknowns are introduced as with the Constraint Equation Method. A simplified penalty approach represents the so called gap element, which is an extra element defined across the contact interface. The gap element is the most commonly used contact method in the FEM because of its ease of use and its reliability. The gap element approach is especially relevant to this work and shall be reviewed in more detail in section 2.3.2.

There is a very large amount of literature on the FEM and the BEM in contact problems and Guzelbey (1992), a previous author at Cranfield University, presents a thorough literature review on this topic starting from the first publications in the early 1970s and concluding with more recent publications up to the year 1992. A comparison of the FEM and the BEM in Guzelbey's work showed that for linear elastic contact analysis the BEM requires less computational and human effort, i.e. is superior to the FEM. However, for contact analysis with other non-linearities present, such as elasto-plastic contact analysis, the FEM was found to be superior to the BEM since the BEM algorithm indicates convergence problems as well as large CPU times.

Since Guzelbey's literature review many papers have been published although it seems as if FEM-contact has almost reached saturation so that only very specialised applications are investigated whilst the original contribution to FEM contact algorithms is rather little. In contrast the BEM contact research is younger and many researchers recognised the advantages which BEM can offer in elastic contact problems and therefore some interesting developments were made in this area.

The authors of Olukoko *et al.* (1993b) and Olukoko *et al.* (1993c) presented a new iterative BE contact algorithm which is not based on node-to-node contact and can therefore allow an independent meshing of the contacting bodies. Element shape functions were used to distribute the contact geometry, displacements and tractions from

the node of one contacting body to the mating element of the contacting body. The independent meshing can lead to a significant reduction in the effort associated with the meshing and remeshing of the contact area.

A very important part within the solution of contact problems is the applied contact algorithm which evaluates the size of the load increments and models the changes in contact and frictional conditions. The efforts of Simunovic and Saigal (Simunovic and Saigal (1992), Simunovic and Saigal (1995a) and Simunovic and Saigal (1995b)) were mainly concerned with the application of linear and quadratic programming techniques to BE contact algorithms, whilst some other authors used mathematical programming techniques (Kong *et al.* (1993), Gakwaya *et al.* (1992) and Zhu (1995)). All these techniques are mainly known from mathematical optimisation theory and their recent application to contact problems can supply a solution without the conventional iterative trial and error method. Therefore convergence can be reached easier, and Zhu (1995) even claimed that, if there is no unloading, only one incremental step is needed for a solution.

A versatile incremental iterative algorithm which is "stable and robust" is the main focus of Man and his co authors (Man and Aliabadi (1993) and Man *et al.* (1993)). Similarly Paris and his co workers (Paris *et al.* (1992), Garrido *et al.* (1994) and Paris *et al.* (1995)) developed a new iterative BE contact procedure for different contact situations. Chen and Chen (1992) showed an iterative BE method which is based on a new transformation matrix which allows a mismatching of the contacting nodes. A BE solution procedure for three dimensional frictional contact which allows the use of parallel processors is suggested in Sipic and Rabinovic (1993) and contact problems involving three dimensional infinite BE subregions were investigated by Liu and Farris (1993).

Practical examples of the application of BE contact algorithms can be found for example in Olukoku *et al.* (1993a) who demonstrate three classical benchmark examples for frictional and non-frictional Hertzian contact using FEM and BEM. Their publication can be very useful for the validation of newly created FE- or BE-contact programs. A

BE contact approach was also used by Park *et al.* (1995) for analysing spherical inclusions and cylindrical fibres embedded in infinite dissimilar matrices. Simunovic and Saigal (1995c) used a BE algorithm to optimise the shape of contacting surfaces such that the peak contact traction is minimised.

There were two recent publications in the elastoplastic BE contact field which both seemed to supply converging solutions (Karami (1991) and Huesmann and Kuhn (1995)). The contact algorithms of both papers were based on iterative load scaling techniques and the results presented confirmed the correctness of these approaches. However, both algorithms relied on a discretisation of the yielding parts of the domain in order to evaluate the domain integrals of the elastoplastic BE formulation. Hence the modelling advantage of an elastoplastic BEM analysis compared to an elastoplastic FEM analysis is more apparent than real.

Very recently a coupled FEM and BEM scheme for contact analysis was suggested in Ezawa and Okamoto (1995). The contact formulation which was used in this paper was based on a penalty method. FEM and BEM were coupled with a direct coupling method which was based on an equivalent BE stiffness matrix. The efficiency of a coupled FEM-BEM contact analysis was demonstrated with the case study of the contact between a magnetic disk and a slider. The slider and a thin oil film on the disk were modelled with the FEM whilst the rather large bulky disk was modelled with the BEM. Coupled FEM-BEM contact analysis was also used by Beaudoin and Dawson (1995) for analysing the deformation of a plastic workpiece by an elastic tool. The viscoplastic workpiece was modelled with the FEM and the elastic tool with the BEM and a simple contact algorithm was used to model the interface.

2.3.2 Gap elements

The gap element is the most commonly used contact method in FEM and belongs to the penalty method category. Extra interface elements (gap elements) are defined across the contact area and the penalty parameters of each of these gap elements are simply stiffnesses which are added to the structural system of equations. Within two-

dimensional analysis gap elements can be either two-dimensional (area element) or one-dimensional (line element). Generally gap element algorithms are iterative so that the present non-linearities can be overcome. Within each successful iteration gap element stiffnesses (penalty parameters) change, so that geometrical non-linearities (e.g. non-conforming contact) as well as friction can be simulated.

The first time the gap element appeared in the literature was when Stadler and Weiss (1979) published their article on two-dimensional gap finite elements which can model contact by changing their normal and tangential Young's moduli (gap moduli) according to the contact conditions. The numerical procedure which Stadler and his co-workers applied relied very much on an analogy with the strain invariance principle as employed for plasticity calculations.

A one-dimensional gap finite element is described by Mazurkiewicz and Ostachowicz (1983) and later by Hellen (1988). Mazurkiewicz and Ostachowicz described the iterative numerical algorithm applied to the gap closure/opening and friction as well as the basic principles of the one-dimensional gap element. It was concluded that, although one-dimensional gap elements are easy to implement, they are restricted because of the node-to-node contact so that for example large sliding contact can not be modelled. A different kind of one-dimensional gap element was introduced by Hitchings (1988), who defined the gap element to connect two Gauss points rather than two nodes.

A more general formulation of gap finite element analysis is given by Bai and Zhao (1988), who formulated two-dimensional and three-dimensional gap elements taking into account non-linearities caused by large deformation, friction and materials.

A recent investigation of Lee (1993) showed that in order to obtain lower analysis times for iterative gap element algorithms it becomes necessary to provide a direction by adaptive algorithms. Adaptive algorithms can help to compromise processes in gap element algorithms or to choose more suitable iteration paths. The author also concluded that the most crucial task in using gap elements is to determine suitable penalty stiffnesses, which dictates the accuracy and efficiency of the contact analysis.

Choi and Chung (1996) developed an elastoplastic gap element algorithm. A special gap element was combined with finite elements which use automated local mesh refinement for three dimensional elastoplastic stress analysis. The authors conclude that their method offers assured convergence and that the suggested gap element algorithm can be an attractive alternative to existing more complicated algorithms.

All of these papers on gap elements are based on the FEM. This can mainly be explained with the fact that the gap element has finite element type equations which are not directly compatible to BEM equations. Nevertheless a paper which is based on the BEM and uses a penalty method was published by Yamazaki and Mori (1989) (and similarly by Yamazaki *et al* (1994)) but the penalty parameters are introduced in the variational formulation so that this approach is not really related to the gap element approach.

2.4 CONCLUSIONS

It was found in the literature review that there is a large amount of publications in all the areas which are relevant to this work, namely the coupling of FEM and BEM, elastoplastic and contact analysis. However no thoroughly fitting work was found which uses a coupled FEM/BEM scheme for elastoplastic contact analysis. Nevertheless several useful findings with respect to the topic of this PhD work were made which can be summarised as follows:

- * Three different coupling methods can be found in the literature, namely the direct coupling, the variational coupling and the iterative coupling. The drawback of the variational coupling is that it can not easily be implemented into already existing FEM and/or BEM codes. In contrast the iterative coupling is the easiest to implement and also allows the use of parallel processing. However, the solution within iterative coupling is based on an iterative procedure which leads to an error in the results which depends on predefined accuracy measures. It is questionable whether the iterative coupling is suitable for non-linear problems,

because here the errors may increase within iterative procedures. Altogether the direct coupling offers the most appropriate method for the scope of this work and within this method BE-format offers the advantage compared to FE format that no additional matrix inversion is needed.

- * A lot of research showed that symmetrised BEM matrices always deliver less accurate results than the original naturally unsymmetric BEM matrices. The result deviations may be in an unacceptable range depending on the symmetrisation procedure used and the problem to be solved. It seems therefore wiser to use the unsymmetric form especially because of the non-linear character of this work.
- * Most of the literature on the coupling of the FEM and the BEM is focused on different coupling methods and computational procedures and symmetrisation of matrices. Not so much work can be found on the practical applications of coupled FEM/BEM schemes. Many practical problem areas which promise advantages with coupled FEM/BEM are not investigated at all or not sufficiently. Two of those areas are contact problems and elastoplasticity.
- * It can be concluded from reviewing many BEM plasticity publications that the FEM is superior to the BEM in elastoplastic problems as far as ease of implementation, computational procedures and CPU time are concerned. The general modelling advantage of the BEM loses some of its significance in elastoplasticity, since most BEM algorithms presented in the literature rely on a discretisation of the plastic domain zones. A coupling of plastic FEM and elastic BEM could be an ideal solution method for partly plastic problems since it does not lead to more modelling effort than a pure BEM scheme.
- * Only one relevant paper which uses coupling of the plastic FEM and the elastic BEM could be found in the literature (Wearing and Burstow (1994)). However, it is based on a non-conforming BEM formulation with external source points. An extension of this work to the more generally used and advantageous

conforming BEM formulation is appropriate and necessary.

- * Gap elements provide a very simple and easy-to-implement contact solution method which nevertheless can solve a wide range of contact problems. Although they have FEM format and were so far only implemented in FEM algorithms, an application to BEM analysis or coupled FEM/BEM analysis could simplify previous algorithms.
- * There are no publications with coupled FEM/BEM applied to elastoplastic contact problems yet. An original contribution on this topic may underline the usefulness of the coupling of the FEM and the BEM.

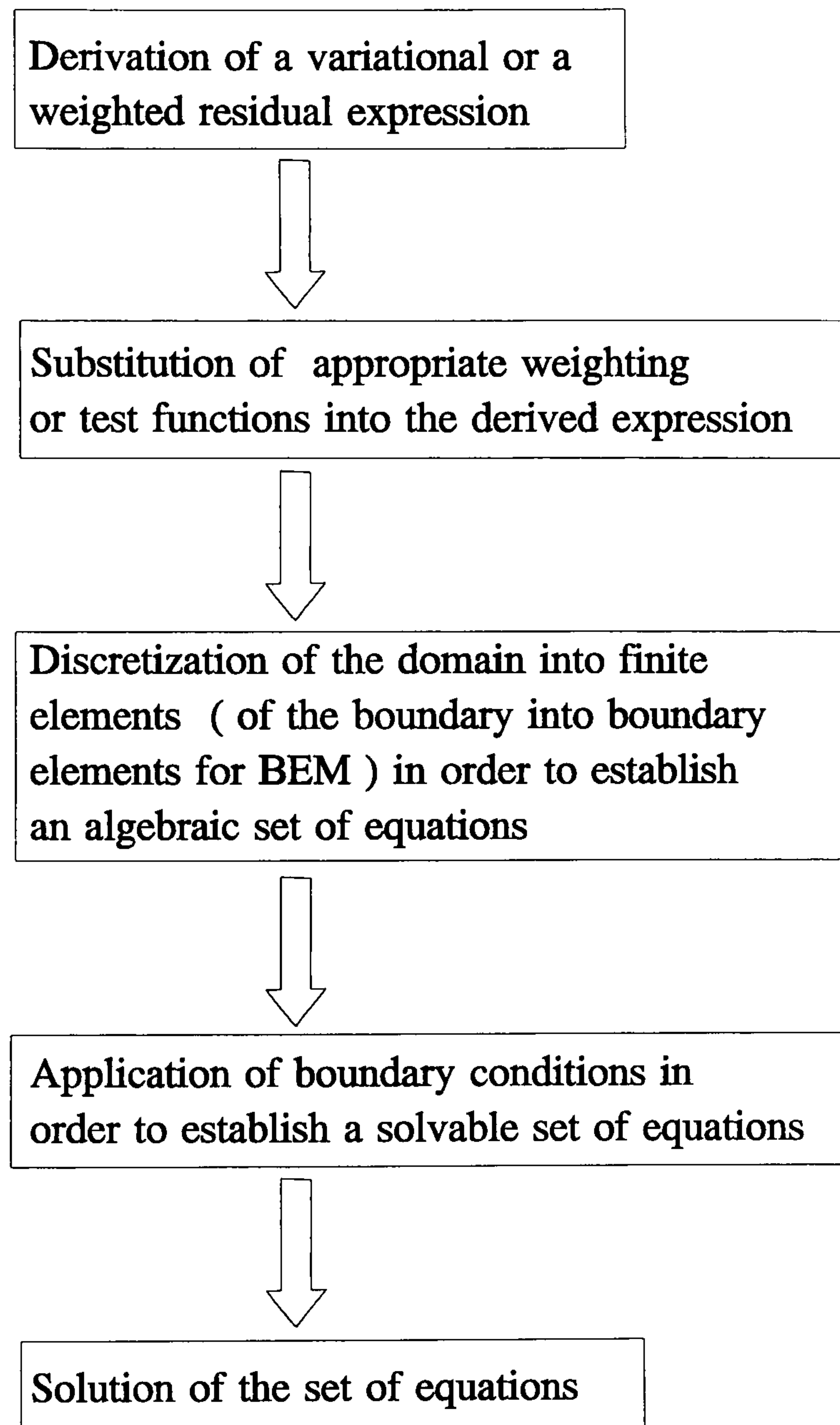


Figure 2.1 Typical steps of an FEM procedure or a BEM procedure.

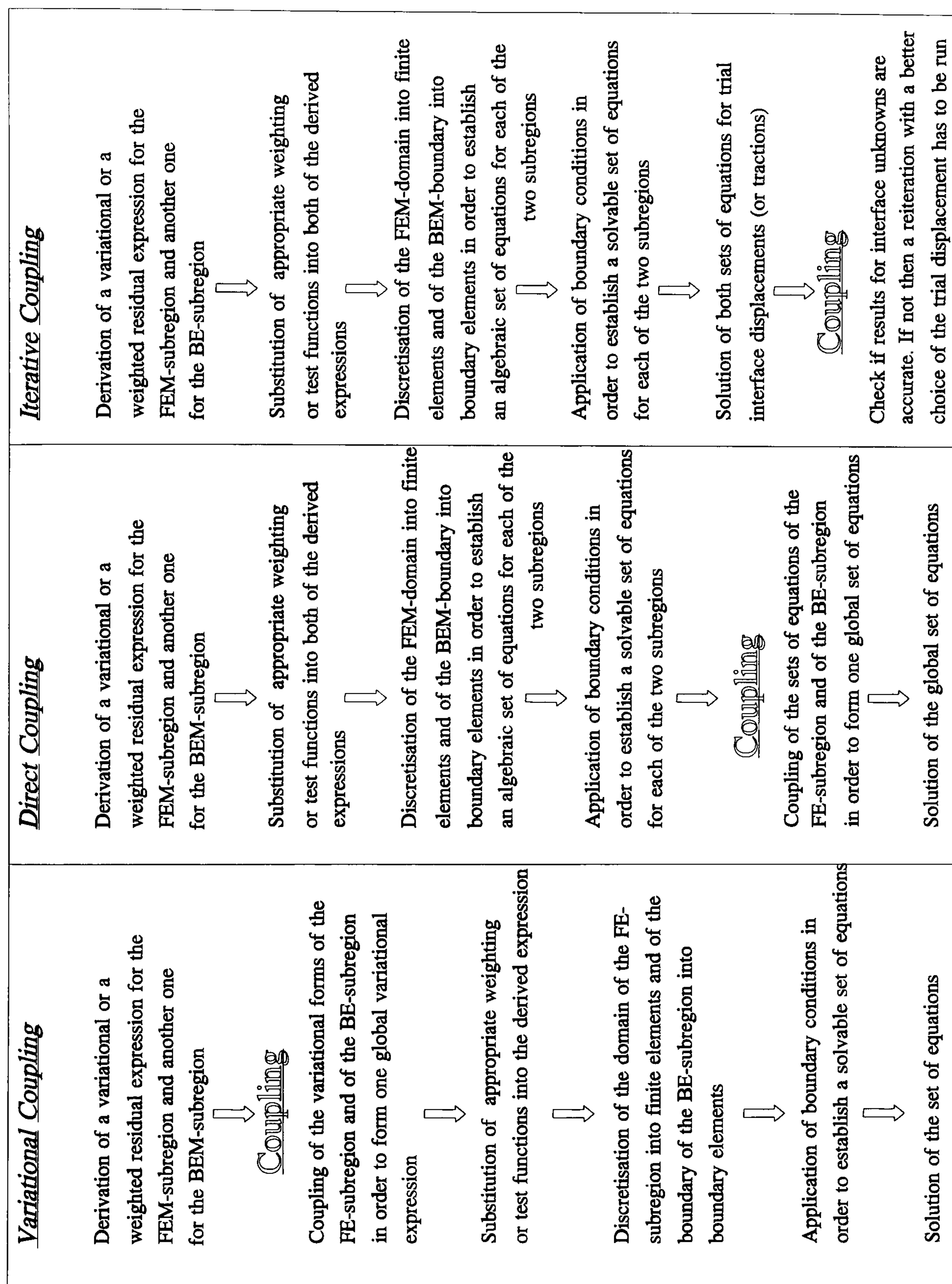


Figure 2.2 Insertion of the coupling step into the typical course of steps for different methods of coupling.

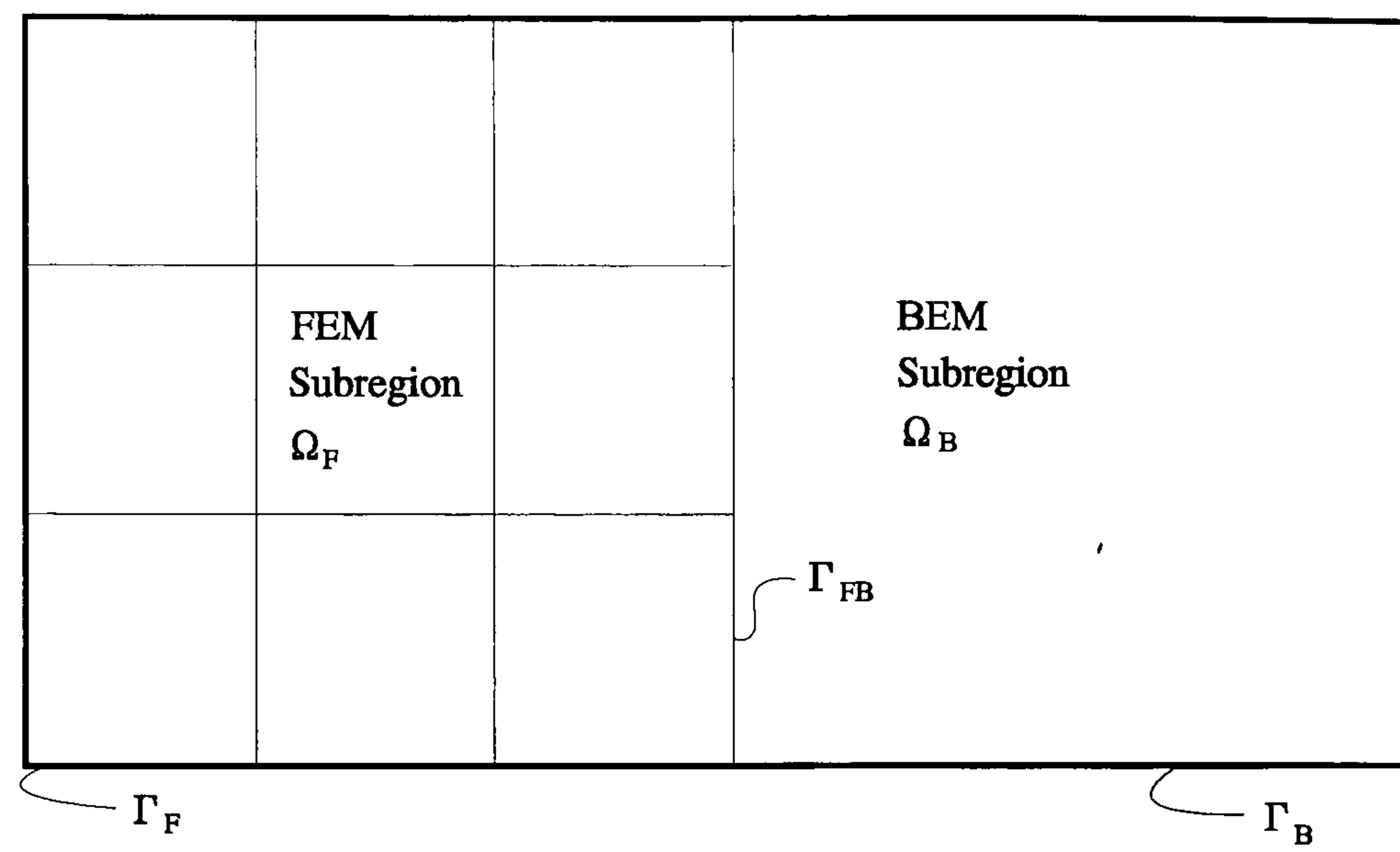


Figure 2.3 A structure divided into an FEM subregion and a BEM subregion.

CHAPTER 3

ELASTICITY PROBLEMS

In this chapter the basic governing equations and finite and boundary element formulations for two dimensional isotropic elastic stress analysis problems are summarised.

In the boundary element section (section 3.3) the boundary element subregion procedure is explained in detail, since it was implemented in the coupled FEM/BEM program. The coupling of the FEM subregion and the BEM subregion (as presented in section 6.2) relies on a similar scheme which makes use of the subregional BEM equations presented here.

The solution of the corner problem of the conforming isoparametric BEM formulation is also summarised in this chapter (section 3.3.3), because it will reoccur in a similar way on FEM-BEM interfaces (section 6.2).

3.1 GOVERNING EQUATIONS

For two dimensional stress analysis problems under plane stress or plane strain conditions, 8 basic independent parameters are to be determined: u , v , ϵ_x , ϵ_y , γ_{xy} , σ_x , σ_y , τ_{xy} . They are governed by equations which shall be summarised in this section.

The components of Cauchy's strain tensor for small strain problems may be defined by the following strain displacement relationship:

$$\begin{aligned}\epsilon_x &= \frac{\partial u}{\partial x}, & \epsilon_y &= \frac{\partial v}{\partial y} \\ \gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\end{aligned}\tag{3.1}$$

Another set of equations can be derived by using the equilibrium of the internal forces and the internal moments induced by the stress components acting on an infinitesimal element. From the internal force equilibrium it can be deduced that:

$$\begin{aligned}\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + f_x &= 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + f_y &= 0\end{aligned}\tag{3.2}$$

And the internal moment equilibrium yields:

$$\tau_{xy} = \tau_{yx}\tag{3.3}$$

Similarly by considering the external equilibrium of the boundary forces and the stress components on an infinitesimal plane:

$$\begin{aligned}T_x &= l\sigma_x + m\tau_{xy} \\ T_y &= l\tau_{xy} + m\sigma_y\end{aligned}\tag{3.4}$$

where l and m are directional cosines of the surface normal as defined in El-Zafrany (1993).

For isotropically elastic materials the stress components are related to the strain components by Hooke's law as follows:

$$\boldsymbol{\sigma} = \mathbf{D} \boldsymbol{\epsilon}\tag{3.5}$$

where

$$\mathbf{D} = \frac{2\mu_s}{1-2p} \begin{bmatrix} 1-p & p & 0 \\ p & 1-p & 0 \\ 0 & 0 & \frac{1-2p}{2} \end{bmatrix} \quad (3.6)$$

$$\boldsymbol{\sigma} = \{ \sigma_x \quad \sigma_y \quad \tau_{xy} \} \quad (3.7)$$

$$\boldsymbol{\epsilon} = \{ \epsilon_x \quad \epsilon_y \quad \gamma_{xy} \} \quad (3.8)$$

and in equation (3.6) $p = \nu$ for plane-strain problems and $p = \frac{\nu}{(1+\nu)}$ for plane-stress problems.

3.2 FINITE ELEMENT THEORY

The backbone of the FEM is formed by domain integral equations which describe the governing problem. A solution of these equations can then be found by piecewise discretising the whole domain into small segments which are also known as finite elements (El-Zafrany(1998a)).

Finite element equations may be formulated in matrix form as follows:

$$\mathbf{K}_e \boldsymbol{\delta}_e = \mathbf{F}_e \quad (3.9)$$

with

$$\mathbf{K}_e = \int_{\text{element}} \int h \mathbf{B}^t \mathbf{D} \mathbf{B} dx dy \quad (3.10)$$

$$\boldsymbol{\delta}_e = \{ u_1 \quad v_1 \quad u_2 \quad v_2 \quad \dots \quad u_n \quad v_n \} \quad (3.11)$$

$$\mathbf{F}_e = \{ F_{x_1} \quad F_{y_1} \quad F_{x_2} \quad F_{y_2} \quad \dots \quad F_{x_n} \quad F_{y_n} \} \quad (3.12)$$

where δ_e and F_e are the elemental displacement and force vector respectively which contain components of the elemental nodes 1 to n .

The integration over the element domain in equation (3.10) for the elemental stiffness matrix K_e can be numerically evaluated by means of a Gauss quadrature scheme, which can be applied after a transformation of the x - y coordinate system into an intrinsic coordinate system.

The B -matrix in equation (3.10) is a function of derivatives of the element Lagrangian shape functions. It defines the elemental displacement-strain-relationship ($\epsilon = B \delta_e$).

Equations (3.9) of each of the finite elements defined in the domain can then be assembled into one structural system of equations in order to obtain a global solution. This leads to the following system of equations in matrix form:

$$K \delta = F + R \quad (3.13)$$

where F is a global vector of externally applied forces, while R is a global vector of unknown reaction forces. A solvable set of equations can be established by applying the predefined boundary conditions to equation (3.13), which is shown for the purpose of an FEM subregion which is to be coupled to a BEM subregion in section 4.2.

3.3 BOUNDARY ELEMENT THEORY

It is the basic idea of the BEM to transform the governing differential equations as presented in section 3.1 into equations in terms of boundary integrals. Therefore the dimensionality of the given problem is reduced by one dimension. Discretisation of the boundary of the problem domain into boundary elements in a way similar to the domain discretisation in FEM is then used so that numerical methods can be employed.

3.3.1 The boundary integral equations of displacements

The boundary integral equations of displacements are defined with respect to a source point (x_i, y_i) and may be formulated as follows:

$$C_i u_i + \oint_{\Gamma} (F_{11} u + F_{21} v) d\Gamma = \oint_{\Gamma} (G_{11} T_x + G_{21} T_y) d\Gamma + U(x_i, y_i) \quad (3.14)$$

$$C_i v_i + \oint_{\Gamma} (F_{12} u + F_{22} v) d\Gamma = \oint_{\Gamma} (G_{12} T_x + G_{22} T_y) d\Gamma + V(x_i, y_i) \quad (3.15)$$

The BEM formulation presented here is used throughout this work and is based on isoparametric boundary elements. An elemental displacement vector and an elemental traction vector for such a n -noded isoparametric boundary element may be defined as follows:

$$\boldsymbol{\delta}_e = \{ u_1 \quad v_1 \quad u_2 \quad v_2 \quad \dots \quad u_n \quad v_n \} \quad (3.16)$$

$$\mathbf{T}_e = \{ T_{x_1} \quad T_{y_1} \quad T_{x_2} \quad T_{y_2} \quad \dots \quad T_{x_n} \quad T_{y_n} \} \quad (3.17)$$

The Boundary Integral Equations can be rewritten in terms of these elemental vectors with respect to a source point (x_i, y_i) as follows (El-Zafrany (1993)):

$$C_i \mathbf{q}_i + \sum_{e=1}^{n_e} \mathbf{h}^{(e)}(x_i, y_i) \boldsymbol{\delta}_e = \sum_{e=1}^{n_e} \mathbf{g}^{(e)}(x_i, y_i) \mathbf{T}_e + \mathbf{Q}_i \quad (3.18)$$

with

$$\mathbf{q}_i = \{ u(x_i, y_i) \quad v(x_i, y_i) \} \quad (3.19)$$

$$\mathbf{Q}_i = \{ U(x_i, y_i) \quad V(x_i, y_i) \} \quad (3.20)$$

Where C_i is a shape factor which depends on the location of the source point (e.g. $C_i=1$ for a source point inside the analysed domain) and the element matrices $\mathbf{g}^{(e)}$ and $\mathbf{h}^{(e)}$ are defined as follows:

$$g_{a,2(j-1)+b}^{(e)}(x_i, y_i) = \int_{\Gamma_e} G_{ba}(x - x_i, y - y_i) N_j(\xi) d\Gamma \quad (3.21)$$

$$h_{a,2(j-1)+b}^{(e)}(x_i, y_i) = \int_{\Gamma_e} F_{ba}(x - x_i, y - y_i) N_j(\xi) d\Gamma \quad (3.22)$$

where $a = 1, 2$, $b = 1, 2$ and $j = 1, 2, \dots, n$. $N_j(\xi)$ is a Lagrangian shape function with respect to the intrinsic coordinate ξ .

$U(x_i, y_i)$ and $V(x_i, y_i)$ in equation (3.20) contain domain integrals in terms of the body forces f_x and f_y and may be written as follows:

$$U(x_i, y_i) = \iint_{\Omega} (G_{11} f_x + G_{21} f_y) dx dy \quad (3.23)$$

$$V(x_i, y_i) = \iint_{\Omega} (G_{12} f_x + G_{22} f_y) dx dy \quad (3.24)$$

The body forces f_x and f_y in equations (3.23) and (3.24) can be generated by any type of domain loading such as for example translational or centrifugal acceleration. The evaluation of the domain integrals in equations (3.23) and (3.24) needs to be undertaken for the general case by means of a discretisation of the domain (Hussain(1989)). However these terms are cancelled out if there are no body forces and it is also possible to derive expressions in terms of boundary integrals only for very simple cases of domain loading such as constant or linear inertial loading.

Notice that the terms G_{11} , G_{12} , G_{21} , G_{22} , F_{11} , F_{12} , F_{21} and F_{22} used in equations (3.14)-(3.15) as well as (3.21)-(3.24) are fundamental solution parameters as listed in Appendix A.

3.3.2 Steps of the subregion Boundary Element Method

Boundary element analysis of complex shapes can be simplified by dividing the domain of the problem into a suitable number of subregions. This is essential for the analysis of cracked components and contacting bodies. A desired "side effect" of using BE subregions is that the solution variables of the subregions are uncoupled and therefore the structural matrix is not fully populated any longer. In the two-dimensional boundary element subregion procedure the following steps become necessary:

Step 1: Discretisation of the boundary

Considering the whole boundary, the element nodes lead to a total, global number of nodes m . Therefore the following global nodal vectors can be defined:

$$\boldsymbol{\delta} = \{ u_1 \quad v_1 \quad u_2 \quad v_2 \quad \dots \quad u_m \quad v_m \} \quad (3.25)$$

$$\boldsymbol{T} = \{ T_{x_1} \quad T_{y_1} \quad T_{x_2} \quad T_{y_2} \quad \dots \quad T_{x_m} \quad T_{y_m} \} \quad (3.26)$$

If the problem domain is modelled in terms of two different subregions α and β as shown in figure 3.1, the displacement and traction vectors can be partitioned into the following vectors:

$$\boldsymbol{\delta}_\alpha = \{ \boldsymbol{\delta}_{\alpha\alpha} \quad \vdots \quad \boldsymbol{\delta}_{\alpha\beta} \} \quad (3.27)$$

$$\boldsymbol{T}_\alpha = \{ \boldsymbol{T}_{\alpha\alpha} \quad \vdots \quad \boldsymbol{T}_{\alpha\beta} \} \quad (3.28)$$

$$\boldsymbol{\delta}_\beta = \{ \boldsymbol{\delta}_{\beta\beta} \quad \vdots \quad \boldsymbol{\delta}_{\beta\alpha} \} \quad (3.29)$$

$$\boldsymbol{T}_\beta = \{ \boldsymbol{T}_{\beta\beta} \quad \vdots \quad \boldsymbol{T}_{\beta\alpha} \} \quad (3.30)$$

where $\boldsymbol{\delta}_{\alpha\alpha}$ is the vector of the nodal boundary displacements for the independent nodes on subregion α (located on $\Gamma_{\alpha\alpha}$ in figure 3.1), and $\boldsymbol{\delta}_{\alpha\beta}$ contains the nodal boundary

displacements of subregion α for nodes common with subregion β (located on $\Gamma_{\alpha\beta}$ in figure 3.1). Independent nodes are all the nodes which are not on the common boundary between the two subregions. Similarly $\delta_{\beta\beta}$ represents the nodal boundary displacements for the independent nodes on subregion β (located on $\Gamma_{\beta\beta}$) and $\delta_{\beta\alpha}$ has the nodal boundary displacements of subregion β for nodes common with subregion α (located on $\Gamma_{\beta\alpha}$). The subscripts of the traction vectors have the same meaning as the subscripts of the displacement vectors.

Step 2: Assembly of the matrix equation for the whole boundary of each subregion

By using each boundary node of each of the defined boundary elements as a source point in the boundary integral equations of equation (3.18), a global solvable set of equations can be established by assembling the element equations according to a topology array. If this is done separately for each of the two subregions, the resulting sets of equations can be expressed in the following matrix equations for subregions α and β respectively:

$$\mathbf{H}_{\alpha} \delta_{\alpha} = \mathbf{G}_{\alpha} \mathbf{T}_{\alpha} + \mathbf{F}_{\alpha} \quad (3.31)$$

$$\mathbf{H}_{\beta} \delta_{\beta} = \mathbf{G}_{\beta} \mathbf{T}_{\beta} + \mathbf{F}_{\beta} \quad (3.32)$$

where $\mathbf{G}$ and $\mathbf{H}$ are structural matrices containing the assembled elemental $\mathbf{g}^{(e)}$ and $\mathbf{h}^{(e)}$ matrices respectively. $\mathbf{F}$ represents an assembly of the body force terms $\mathbf{Q}_i$ as defined in equation (3.20) for each of the source points.

Partitioning δ_{α} and $\mathbf{T}_{\alpha}$ as in equations (3.27) and (3.28) leads to:

$$\begin{bmatrix} \mathbf{H}_{\alpha\alpha} & \vdots & \mathbf{H}_{\alpha\beta} \end{bmatrix} \begin{bmatrix} \delta_{\alpha\alpha} \\ \vdots \\ \delta_{\alpha\beta} \end{bmatrix} = \begin{bmatrix} \mathbf{G}_{\alpha\alpha} & \vdots & \mathbf{G}_{\alpha\beta} \end{bmatrix} \begin{bmatrix} \mathbf{T}_{\alpha\alpha} \\ \vdots \\ \mathbf{T}_{\alpha\beta} \end{bmatrix} + \mathbf{F}_{\alpha} \quad (3.33)$$

which may be rewritten as

$$\mathbf{H}_{\alpha\alpha} \boldsymbol{\delta}_{\alpha\alpha} + \mathbf{H}_{\alpha\beta} \boldsymbol{\delta}_{\alpha\beta} = \mathbf{G}_{\alpha\alpha} \mathbf{T}_{\alpha\alpha} + \mathbf{G}_{\alpha\beta} \mathbf{T}_{\alpha\beta} + \mathbf{F}_{\alpha} \quad (3.34)$$

Similarly for subregion β :

$$\mathbf{H}_{\beta\beta} \boldsymbol{\delta}_{\beta\beta} + \mathbf{H}_{\beta\alpha} \boldsymbol{\delta}_{\beta\alpha} = \mathbf{G}_{\beta\beta} \mathbf{T}_{\beta\beta} + \mathbf{G}_{\beta\alpha} \mathbf{T}_{\beta\alpha} + \mathbf{F}_{\beta} \quad (3.35)$$

Step 3: Application of boundary conditions on independent nodes of each subregion

The pre-defined boundary conditions are applied on the independent nodes of each subregion. Hence, traction and displacement vectors for the independent nodes on subregion α can be partitioned as follows:

$$\boldsymbol{\delta}_{\alpha\alpha} = \{ \boldsymbol{\delta}_{\alpha\alpha}^x \quad : \quad \boldsymbol{\delta}_{\alpha\alpha}^p \} \quad (3.36)$$

$$\mathbf{T}_{\alpha\alpha} = \{ \mathbf{T}_{\alpha\alpha}^x \quad : \quad \mathbf{T}_{\alpha\alpha}^p \} \quad (3.37)$$

where the superscript x stands for a vector of unknowns, whilst a vector with the superscript p consists of prescribed values according to the given boundary conditions. Then equation (3.34) may be rewritten such that:

$$\mathbf{A}_{\alpha\alpha} \mathbf{x}_{\alpha\alpha} + \mathbf{H}_{\alpha\beta} \boldsymbol{\delta}_{\alpha\beta} = \mathbf{B}_{\alpha\alpha} \mathbf{p}_{\alpha\alpha} + \mathbf{G}_{\alpha\beta} \mathbf{T}_{\alpha\beta} + \mathbf{F}_{\alpha} \quad (3.38)$$

where

$$\mathbf{x}_{\alpha\alpha} = \{ \boldsymbol{\delta}_{\alpha\alpha}^x \quad : \quad \mathbf{T}_{\alpha\alpha}^x \} \quad (3.39)$$

$$\mathbf{p}_{\alpha\alpha} = \{ \mathbf{T}_{\alpha\alpha}^p \quad : \quad \boldsymbol{\delta}_{\alpha\alpha}^p \} \quad (3.40)$$

Hence:

$$\mathbf{A}_{\alpha\alpha} \mathbf{x}_{\alpha\alpha} + \mathbf{H}_{\alpha\beta} \boldsymbol{\delta}_{\alpha\beta} - \mathbf{G}_{\alpha\beta} \mathbf{T}_{\alpha\beta} = \mathbf{y}_{\alpha} + \mathbf{F}_{\alpha} \quad (3.41)$$

where

$$\mathbf{y}_{\alpha} = \mathbf{B}_{\alpha\alpha} \mathbf{p}_{\alpha\alpha} \quad (3.42)$$

Similarly for subregion β :

$$\mathbf{A}_{\beta\beta} \mathbf{x}_{\beta\beta} + \mathbf{H}_{\beta\alpha} \boldsymbol{\delta}_{\beta\alpha} - \mathbf{G}_{\beta\alpha} \mathbf{T}_{\beta\alpha} = \mathbf{y}_{\beta} + \mathbf{F}_{\beta} \quad (3.43)$$

Step 4: Assembly of the subregion equations by application of boundary conditions on nodes of the common boundary

In order to solve equations (3.41) and (3.43) it is necessary to apply boundary conditions on the nodes of the common boundary. For the case with no relative movement between the nodes on the common boundary of the subregions, equilibrium and compatibility conditions are as follows:

$$\boldsymbol{\delta}_{\alpha\beta} = \boldsymbol{\delta}_{\beta\alpha} \quad (3.44)$$

$$\mathbf{T}_{\alpha\beta} = -\mathbf{T}_{\beta\alpha} \quad (3.45)$$

By applying these conditions, equations (3.41) and (3.43) can be assembled into the following solvable system of equations:

$$\begin{bmatrix} \mathbf{A}_{\alpha\alpha} & \mathbf{H}_{\alpha\beta} & -\mathbf{G}_{\alpha\beta} & \mathbf{0} \\ \mathbf{0} & \mathbf{H}_{\beta\alpha} & \mathbf{G}_{\beta\alpha} & \mathbf{A}_{\beta\beta} \end{bmatrix} \begin{bmatrix} \mathbf{x}_{\alpha\alpha} \\ \boldsymbol{\delta}_{\alpha\beta} \\ \mathbf{T}_{\alpha\beta} \\ \mathbf{x}_{\beta\beta} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\alpha} + \mathbf{y}_{\alpha} \\ \mathbf{F}_{\beta} + \mathbf{y}_{\beta} \end{bmatrix} \quad (3.46)$$

Step 5: Evaluation of results at boundary nodes

The equation system as in equation (3.46) can be solved by means of an ordinary technique such as the Gauss-elimination. Alternatively a special solver may be used which makes use of the blocked banded structure of the global matrix. Therefore the unknown displacements and tractions at all boundary nodes can be obtained.

3.3.3 The treatment of non-smooth boundaries

If there are corners on the boundary of the analysed body some major difficulties arise. The surface normals are undefined at such points which means that it is impossible to define distinct surface traction and moreover if a source point is located at the corner it is not possible to establish the directional cosines needed for the fundamental solution parameters listed in Appendix A.

A great deal of work published on corner problems uses non-conforming elements where source points and geometrical nodes do not coincide which allows a modelling of a corner between two such non-conforming elements without a source point being located at the corner. However such formulations can not supply the accuracy of the conforming isoparametric BE.

Previous work at Cranfield University (Hussain(1989)) suggests that it is possible to use the conventional isoparametric BEM for such corner problems even though a source point is defined at the corner location. This scheme relies on the modelling of a corner by two separate nodes on connected BEs as shown in figure 3.2. This corner model then leads to the following expression as shown in Appendix B:

$$\begin{aligned} C_{11}(x_i, y_i) u_i + C_{12}(x_i, y_i) v_i + \int_{\Gamma'} (F_{11} u + F_{21} v) d\Gamma \\ = \int_{\Gamma'} (G_{11} T_x + G_{21} T_y) d\Gamma + U(x_i, y_i) \end{aligned} \quad (3.47)$$

$$\begin{aligned} C_{21}(x_i, y_i) u_i + C_{22}(x_i, y_i) v_i + \int_{\Gamma'} (F_{12} u + F_{22} v) d\Gamma \\ = \int_{\Gamma'} (G_{12} T_x + G_{22} T_y) d\Gamma + V(x_i, y_i) \end{aligned} \quad (3.48)$$

where Γ' represents the boundary without corners which means that the boundary integrals are simply considered without the effect of the corners. Such effects of the

corners are taken into account by the C_{11} , C_{12} , C_{21} and C_{22} coefficients, which can be shown to be defined for problems with finite domains as follows (Appendix B):

$$C_{ab}(x_i, y_i) = - \int_{\Gamma'} F_{ba}(x-x_i, y-y_i) d\Gamma \quad (3.49)$$

Therefore whenever there is a corner on the boundary it is modelled as shown in figure 3.2. One distinct traction vector and one source point are defined at each of the two corner nodes.

It is important to notice that four equations can be established at a corner, which are equations (3.47) and (3.48) for each of the two source points. However, the displacements of the two corner nodes are equivalent and by using the same source point the established equations are similar. Therefore situations may occur, depending on the boundary conditions at such a double corner node, where there are more unknowns than equations available (Mitra and Ingber (1993)).

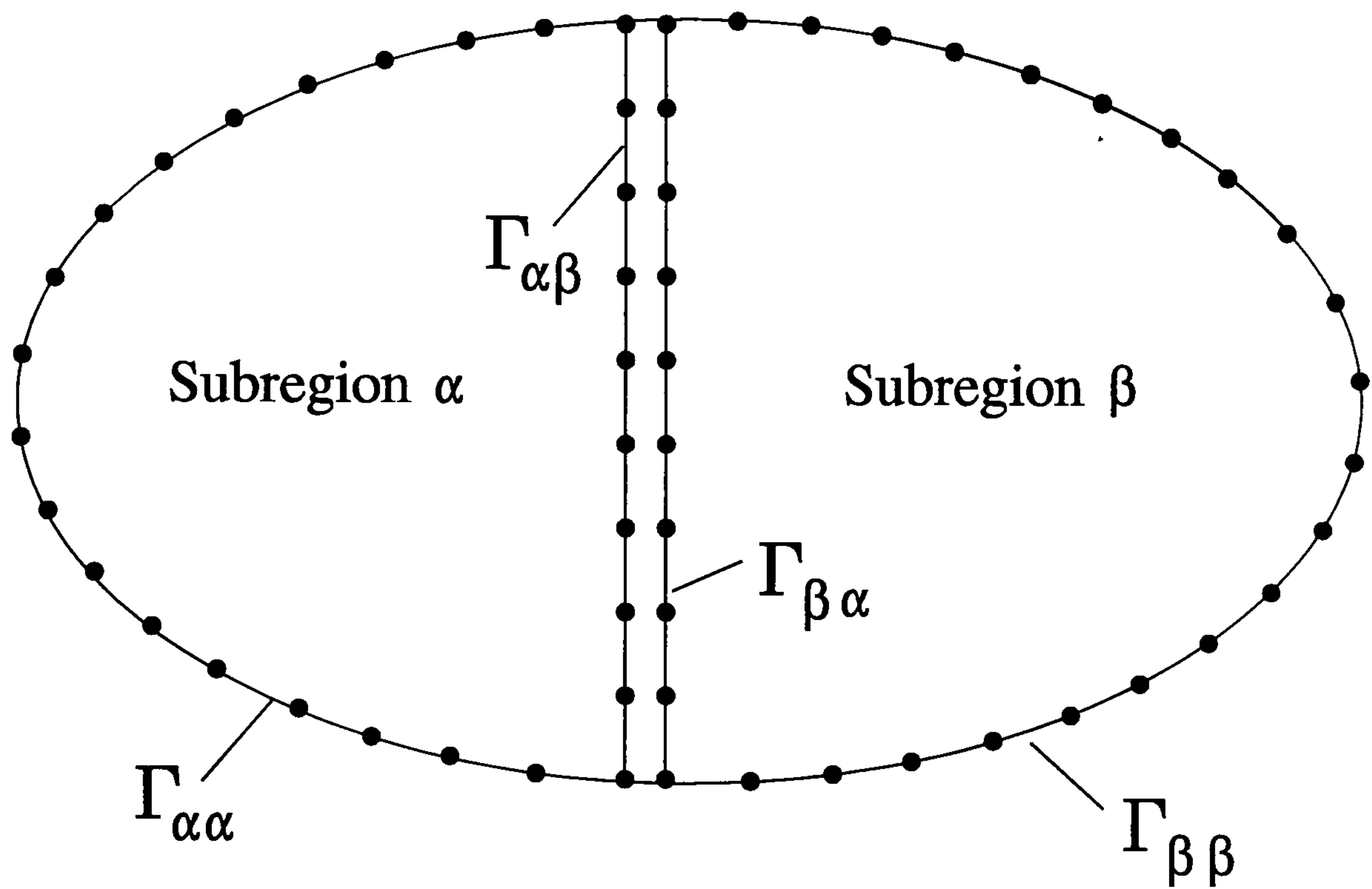


Figure 3.1 A domain discretised into two boundary element subregions.

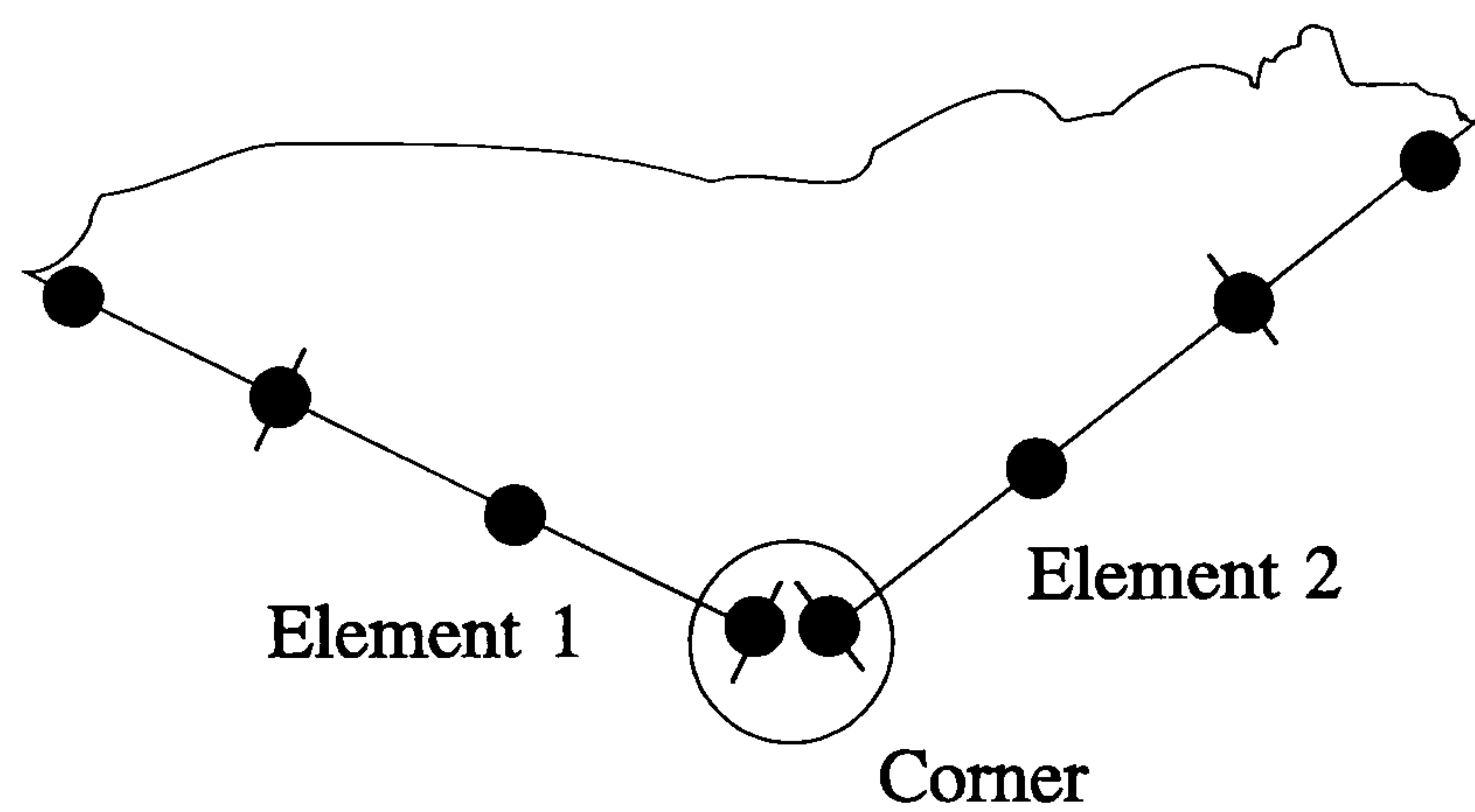


Figure 3.2 The modelling of a corner with two separate nodes.

CHAPTER 4

THE COUPLING OF THE FEM AND THE BEM

4.1 INTRODUCTION

Consider a domain Ω , which is subdivided into a BE-subdomain α and an FE-subdomain β as shown in figure 4.1. The following subregional set of equations can be established by assembling the boundary element equations according to a topology array (ref section 3.3.2):

$$\mathbf{H}_{\alpha} \boldsymbol{\delta}_{\alpha} = \mathbf{G}_{\alpha} \mathbf{T}_{\alpha} + \mathbf{F}_{\alpha} \quad (4.1)$$

Similarly for the finite element subregion, by assembling all finite element contributions into one subregional system of equations:

$$\mathbf{K}_{\beta} \boldsymbol{\delta}_{\beta} = \mathbf{F}_{\beta} + \mathbf{R}_{\beta} \quad (4.2)$$

It was already established in the literature review that the direct coupling is the most suitable coupling method for the scope of this work. Theoretically there are two possibilities of how to directly couple equations (4.1) and (4.2). The first one is to transform the BE-equation (4.1) into FEM format and the second one is to transform the FE-equation (4.2) into BEM format.

Although most of the work published on the coupling of FEM and BEM relies on the first method, the author decided to use the second method because it has advantages with respect to this work. For one the transformation of equation (4.1) into FEM format requires a full inversion of the $\mathbf{G}_{\alpha}$ -matrix in order to isolate the traction vector $\mathbf{T}_{\alpha}$ (if no body forces $\mathbf{F}_{\alpha}$ are present), which is not necessary for the transformation of the

FEM equation (4.2) into BEM format. Moreover it was important for this work to create a method for the coupling of the FEM and the BEM which allows the modelling of body forces in the FEM- as well as the BEM-subregion. Body forces lead to the $\mathbf{F}_\alpha$ -term in equation (4.1), which complicates the transformation of the BEM-equation (4.1) into FEM format, whilst it does not interfere with the chosen method of transformation.

The coupling method presented in this section only transforms the forces of the FE-nodes on the FEM/BEM-interface into tractions which saves computational effort and supplies better accuracy than a complete transformation of the forces of all the FE-nodes. The method is similar to the BE-subregion procedure presented in section 2.2 and it is therefore possible to use the BE-subregion procedure as well as the coupling procedure in the same package. Hence several BE-subregions and FE-subregions can be defined and coupled.

For a BEM subregion it has been already shown that:

$$\mathbf{A}_{\alpha\alpha} \mathbf{x}_{\alpha\alpha} + \mathbf{H}_{\alpha\beta} \boldsymbol{\delta}_{\alpha\beta} - \mathbf{G}_{\alpha\beta} \mathbf{T}_{\alpha\beta} = \mathbf{y}_\alpha + \mathbf{F}_\alpha \quad (3.41)$$

It will be shown in section 4.2 how to partition the subregional FEM equations such that the FEM/BEM-interface degrees of freedom are separated in a way similar to the subregional BE-equation (3.41). Then it will be demonstrated in section 4.3 how to transform the FEM-type equations of the interface degrees of freedom to BEM format and how to couple the resulting FEM-system of equations to equation (3.41). Section 4.4 explains how to discretise the FEM/BEM interface and in section 4.5 a method for the solution of the problem of corners on the FEM/BEM interface is presented.

4.2 SUBREGIONAL FEM EQUATIONS

As already explained in the introduction of this chapter, the FEM subregion β is defined by the following subregional matrix system of equations:

$$\mathbf{K}_\beta \boldsymbol{\delta}_\beta = \mathbf{F}_\beta + \mathbf{R}_\beta \quad (4.2)$$

Where $\mathbf{K}_\beta$ is the global stiffness matrix and $\boldsymbol{\delta}_\beta$ is the global displacement vector of the FE-subregion. $\mathbf{R}_\beta$ is the global reaction force vector which is zero everywhere apart from at degrees of freedom with prescribed displacements and at nodes on the FEM-BEM interface. $\mathbf{F}_\beta$ consists of externally applied nodal forces of known value due to point loads, surface tractions or body loads.

The displacement and the force vectors of equation (4.2) may be partitioned according to the degrees of freedom at which the displacement is prescribed (superscript p) and the degrees of freedom at which the displacement is unknown (superscript x), i.e.

$$\boldsymbol{\delta}_\beta = \{ \boldsymbol{\delta}_\beta^x \quad : \quad \boldsymbol{\delta}_\beta^p \} \quad (4.3)$$

$$\mathbf{F}_\beta = \{ \mathbf{F}_\beta^x \quad : \quad \mathbf{F}_\beta^p \} \quad (4.4)$$

$$\mathbf{R}_\beta = \{ \mathbf{R}_\beta^x \quad : \quad \mathbf{R}_\beta^p \} \quad (4.5)$$

Substituting equations (4.3) - (4.5) into equation (4.2) yields:

$$\begin{bmatrix} \mathbf{K}_\beta^{xx} & \mathbf{K}_\beta^{xp} \\ \mathbf{K}_\beta^{px} & \mathbf{K}_\beta^{pp} \end{bmatrix} \begin{bmatrix} \boldsymbol{\delta}_\beta^x \\ \boldsymbol{\delta}_\beta^p \end{bmatrix} = \begin{bmatrix} \mathbf{F}_\beta^x \\ \mathbf{F}_\beta^p \end{bmatrix} + \begin{bmatrix} \mathbf{R}_\beta^x \\ \mathbf{R}_\beta^p \end{bmatrix} \quad (4.6)$$

i.e.

$$\mathbf{K}_{\beta}^{xx} \boldsymbol{\delta}_{\beta}^x + \mathbf{K}_{\beta}^{xp} \boldsymbol{\delta}_{\beta}^p = \mathbf{F}_{\beta}^x + \mathbf{R}_{\beta}^x \quad (4.7)$$

$$\mathbf{K}_{\beta}^{px} \boldsymbol{\delta}_{\beta}^x + \mathbf{K}_{\beta}^{pp} \boldsymbol{\delta}_{\beta}^p = \mathbf{F}_{\beta}^p + \mathbf{R}_{\beta}^p \quad (4.8)$$

For the forthcoming solution it is sufficient to solve equation (4.7) for the unknown displacements $\boldsymbol{\delta}_{\beta}^x$ and then substitute the solution into equation (4.8) in order to evaluate the unknown $\mathbf{R}_{\beta}^p$. In order to simplify the notation, equation (4.7) may be rewritten in terms of a reduced force vector $\mathbf{F}_{\beta}^{*x}$ as follows:

$$\mathbf{K}_{\beta}^{xx} \boldsymbol{\delta}_{\beta}^x = \mathbf{F}_{\beta}^{*x} + \mathbf{R}_{\beta}^x \quad (4.9)$$

where:

$$\mathbf{F}_{\beta}^{*x} = \mathbf{F}_{\beta}^x - \mathbf{K}_{\beta}^{xp} \boldsymbol{\delta}_{\beta}^p \quad (4.10)$$

The unknown displacement vector, the force vector and the reaction vector of equation (4.9) may then be partitioned according to the division into the defined subregions such that:

$$\boldsymbol{\delta}_{\beta}^x = \{ \boldsymbol{\delta}_{\beta\beta}^x \quad \vdots \quad \boldsymbol{\delta}_{\beta\alpha}^x \} \quad (4.11)$$

$$\mathbf{F}_{\beta}^{*x} = \{ \mathbf{F}_{\beta\beta}^{*x} \quad \vdots \quad \mathbf{F}_{\beta\alpha}^{*x} \} \quad (4.12)$$

$$\mathbf{R}_{\beta}^x = \{ \mathbf{R}_{\beta\beta}^x \quad \vdots \quad \mathbf{R}_{\beta\alpha}^x \} \quad (4.13)$$

where, similarly to the previous section, $\boldsymbol{\delta}_{\beta\alpha}$ is a nodal displacement vector of the interface nodes of subregion β , whilst $\boldsymbol{\delta}_{\beta\beta}^x$ consists of independent nodal displacement components of subregion β . The subscripts of the force and reaction vectors of equation (4.12) and equation (4.13) have the same meaning. Notice that the superscripts of the interface degrees of freedom in equations (4.11)-(4.13) were neglected since it is clear that displacements are always unknown on the interface between subregions.

By substituting equations (4.11)-(4.13) into equation (4.9) a partitioned set of equations can be defined, such that:

$$\begin{bmatrix} \mathbf{K}_{\beta\beta}^{xx} & \mathbf{K}_{\beta\alpha}^{xx} \\ \mathbf{K}_{\alpha\beta}^{xx} & \mathbf{K}_{\alpha\alpha}^{xx} \end{bmatrix} \begin{bmatrix} \boldsymbol{\delta}_{\beta\beta}^x \\ \boldsymbol{\delta}_{\beta\alpha} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\beta\beta}^{*x} \\ \mathbf{F}_{\beta\alpha} \end{bmatrix} + \begin{bmatrix} \mathbf{R}_{\beta\beta}^x \\ \mathbf{R}_{\beta\alpha} \end{bmatrix} \quad (4.14)$$

Notice that $\mathbf{R}_{\beta\beta}^x$ is a null vector, since it is defined at independent finite element degrees of freedom with unknown displacements, and at such degrees of freedom the resulting reaction forces are equal to zero. Therefore equation (4.14) may be expanded to form the following equations for the FE-subregion β :

$$\mathbf{K}_{\beta\beta}^{xx} \boldsymbol{\delta}_{\beta\beta}^x + \mathbf{K}_{\beta\alpha}^{xx} \boldsymbol{\delta}_{\beta\alpha} = \mathbf{F}_{\beta\beta}^{*x} \quad (4.15)$$

$$\mathbf{K}_{\alpha\beta}^{xx} \boldsymbol{\delta}_{\beta\beta}^x + \mathbf{K}_{\alpha\alpha}^{xx} \boldsymbol{\delta}_{\beta\alpha} = \mathbf{F}_{\beta\alpha} + \mathbf{R}_{\beta\alpha} \quad (4.16)$$

The unknown components in equations (4.15) and (4.16) are $\mathbf{R}_{\beta\alpha}$, $\boldsymbol{\delta}_{\beta\beta}^x$ and $\boldsymbol{\delta}_{\beta\alpha}$.

4.3 THE COUPLING OF THE FEM AND THE BEM EQUATIONS

The subregional BEM equation (3.41) and the subregional FEM equations (4.15) and (4.16) cannot be solved yet as there are too many unknowns. Therefore it becomes necessary to establish additional boundary conditions on the FEM/BEM-interface in order to obtain one solvable set of equations. For the case of a continuous interface these additional boundary conditions have to fulfil equilibrium and compatibility requirements. This means that the interface displacements of FEM and BEM subregions need to be equal:

$$\boldsymbol{\delta}_{\alpha\beta} = \boldsymbol{\delta}_{\beta\alpha} \quad (4.17)$$

and moreover the interface tractions need to be equal and opposite:

$$\mathbf{T}_{\alpha\beta} = -\mathbf{T}_{\beta\alpha} \quad (4.18)$$

Equation (4.17) can be implemented directly, but it is clear that equation (4.18) cannot be implemented directly since the FEM-equations are expressed in terms of the nodal interface reaction vector $\mathbf{R}_{\beta\alpha}$ which is not compatible to the desired nodal interface traction vector $\mathbf{T}_{\beta\alpha}$. Therefore, in order to be able to apply condition (4.18) it becomes necessary to transform $\mathbf{R}_{\beta\alpha}$ into an expression in terms of $\mathbf{T}_{\beta\alpha}$. Such a transformation may be expressed as follows:

$$\mathbf{R}_{\beta\alpha} = \mathbf{C} \mathbf{T}_{\beta\alpha} \quad (4.19)$$

where the evaluation of the transformation matrix $\mathbf{C}$ will be demonstrated in section 4.4.

Using equation (4.19), equation (4.16) may be rewritten as follows:

$$\mathbf{K}_{\alpha\beta}^{xx} \boldsymbol{\delta}_{\beta\beta}^x + \mathbf{K}_{\alpha\alpha}^{xx} \boldsymbol{\delta}_{\beta\alpha} - \mathbf{C} \mathbf{T}_{\beta\alpha} = \mathbf{F}_{\beta\alpha} \quad (4.20)$$

Equations (4.15) and (4.20) now represent the FEM subregion systems of equations. These equations can now be coupled to the subregional system of equations of the BEM subregion (equation (3.41)), by applying the interface conditions given by equations (4.17) and (4.18). This leads to the following matrix system of equations:

$$\begin{bmatrix} \mathbf{A}_{\alpha\alpha} & \mathbf{H}_{\alpha\beta} & -\mathbf{G}_{\alpha\beta} & \mathbf{0} \\ \mathbf{0} & \mathbf{K}_{\alpha\alpha}^{xx} & \mathbf{C} & \mathbf{K}_{\alpha\beta}^{xx} \\ \mathbf{0} & \mathbf{K}_{\beta\alpha}^{xx} & \mathbf{0} & \mathbf{K}_{\beta\beta}^{xx} \end{bmatrix} \begin{bmatrix} \mathbf{x}_{\alpha\alpha} \\ \boldsymbol{\delta}_{\alpha\beta} \\ \mathbf{T}_{\alpha\beta} \\ \boldsymbol{\delta}_{\beta\beta}^x \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\alpha} + \mathbf{y}_{\alpha} \\ \mathbf{F}_{\beta\alpha} \\ \mathbf{F}_{\beta\beta}^{*x} \end{bmatrix} \quad (4.21a)$$

$$\text{or:} \quad \mathbf{A} \quad \mathbf{X} = \mathbf{Y} \quad (4.21b)$$

Equation (4.21a) or (4.21b) represents a solvable set of equations. Once it is solved, the obtained subregional FEM results may be substituted into equation (4.8) in order to evaluate the reaction forces at nodes with prescribed displacements ($\mathbf{R}_\beta^p$). (Notice that there are zero terms in the structural matrix and moreover that the FEM stiffness matrix portions are symmetric. Therefore the use of a solver, which takes advantage of these properties could bring savings in CPU time.)

4.4 THE DISCRETISATION OF THE INTERFACE

Connecting finite and boundary element subregions, the interface must have an equal number of nodes on the FEM side and on the BEM side. The connecting finite element sides and boundary elements must have the same number of boundary nodes in order to satisfy equilibrium and compatibility conditions as accurately as possible. Notice that the maximum number of nodes of any finite element side or boundary element on the interface should be no more than 5, because increased numerical errors have been discovered by the author for cases with more than 5 nodes.

For one side of a finite element, nodal tractions which are work equivalent to the nodal forces acting on the element side can be derived as shown in Appendix C. This leads to the following expression for the side of a finite element:

$$\mathbf{F}_{(e)} = \mathbf{C}_{(e)} \mathbf{T}_{(e)} \quad (4.22)$$

For each finite element on the interface, equation (4.22) can be defined for the side of the finite element which is on the interface. Then all the elemental $\mathbf{C}_{(e)}$ -matrices on the interface can be assembled into an interface $\mathbf{C}$ -matrix according to a topology array which defines the required transformation:

$$\mathbf{R}_{\beta\alpha} = \mathbf{C} \mathbf{T}_{\beta\alpha} \quad (4.23)$$

4.5 THE PROBLEM OF CORNERS ON THE INTERFACE

As already shown in section 3.3.3, a corner on the boundary of a boundary element subregion has to be modelled with two boundary nodes which are located at the corner. The coupling method presented in the previous sections leads to too many unknowns if there is a corner on the FEM/BEM-interface because there are two BE nodes to be coupled to one FE node.

This kind of problem occurs for many coupling methods found in the literature. However, not many researchers have investigated solutions. Within most papers found in the literature the interfaces are modelled without corners, or if corners are present solution methods and the result deviations near corners are not presented. An explanation of some principal methods for the solution of corner problems is given in Cruse and Osias (1991) who found that their methods may lead to local deviations.

Principally there are two different types of interface corners which require different treatment. The first type of interface corner is located on the external boundary of the domain as shown in figure 4.3, whilst the second type is located inside the domain as shown in figure 4.4. In sections 4.5.2 and 4.5.3 new simple methods show how the equation system has to be modified to allow for a modelling of interface corners of the first and the second type respectively.

4.5.1 Smooth interface

The solution of cases with smooth interfaces can be undertaken according to the theory presented in sections 4.1-4.3. This shall be demonstrated from a more practical point of view in this section.

A smooth interface is illustrated in figure 4.2, where nodes F1-F5 are finite element nodes and nodes B1-B5 are boundary element nodes. If we only consider the degrees of freedom in y-direction, it is clear that for example for node pair F2-B2 the following

additional equations and unknowns are introduced to the system (similarly valid for the degrees of freedom in x-direction):

Equations: 1 FE-equation + 1 BE-equation =2 equations

Unknowns: $(v)_{B2}, (v)_{F2}, (T_y)_{B2}, (R_y)_{F2}$ =4 unknowns

2 equations and 4 unknowns are introduced, which means that there are 2 unknowns too many. Hence 2 more conditions need to be established in order to obtain a solvable system of equations. The first one is the compatibility of the displacements:

$$(v)_{B2} = (v)_{F2} \quad (4.24)$$

The second extra interface condition is the equilibrium of the interface forces. This condition can not be defined explicitly. Thus for each of the finite element sides on the interface a transformation of nodal forces into nodal tractions needs to be undertaken as described in Appendix C. This leads to the following extra interface condition:

$$(R_y)_{F2} = - \left[(C_{21})_{FE1} (T_y)_{B1} + (C_{22})_{FE1} (T_y)_{B2} + (C_{23})_{FE1} (T_y)_{B3} \right] \quad (4.25)$$

Where the $(C..)_{FE1}$ parameters are parameters from the $C_{(e)}$ matrix of the interface side FE1 (refer to Appendix C).

The system of equations is now solvable. Notice that at FE nodes which are located on two element sides such as for example node 3, the reaction force contributions of both elements need to be taken into account in equation (4.25).

4.5.2 Interface corner on the external boundary

An interface corner which is located on the external boundary can be seen in figure 4.3. There are two different cases how the external boundary conditions can be set, one is that the displacements are prescribed (figure 4.3a) and the other one is that the tractions are prescribed (figure 4.3b) along the boundary. The nodes of interest are denoted F1, F2, F3, F4 and F5 for the FEM subregion and B1, B2, B3, B4, B5 and B6 for the BE-subregion (figure 4.3).

Case a: Prescribed displacements along the external boundary:

In this case there is a prescribed displacement in y-direction along the external boundary as shown in figure 4.3a. The nodes of interest are only the corner nodes F3, B3 and B4 because it is known from section 4.5.1 that all other degrees of freedom lead to a solvable system of equations. By using the nodes F3, B3 and B4 the following additional equations and unknowns are introduced in y-direction:

Equations: 1 FE-equation (F3) + 2 BE-equations (B3 & B4) = 3 equations

Unknowns: $(v)_{B4}, (T_y)_{B3}, (T_y)_{B4}, (R_y)_{F3}$ = 4 unknowns

Therefore there is a situation with one unknown too many. The following additional boundary condition can be established in order to obtain a solvable system of equations:

$$(R_y)_{F3} = - \left[(C_{11})_{FE1} (T_y)_{B4} + (C_{12})_{FE1} (T_y)_{B5} + (C_{13})_{FE1} (T_y)_{B6} \right] + \left[(C_{31})_{BE1} (T_y)_{B1} + (C_{32})_{BE1} (T_y)_{B2} + (C_{33})_{BE1} (T_y)_{B3} \right] \quad (4.26)$$

In equation (4.26) the reaction force consists of two parts. The first part is the bracketed expression in the first line which consists of interface tractions of the boundary element BE2 which is connected to the finite element side FE1. This part represents the reaction force of node B4. It is the same as for a smooth interface and it ensures the equilibrium

of the forces along the interface, i.e. the forces (tractions) are equal and opposite.

The second part of the reaction force is the positive bracketed expression of the second line, which only needs to be introduced for this special case. It represents the reaction force of node B3. This force is equally acting at node B3 and node F3 and it is therefore positive in equation (4.26).

Important remark: Notice that it is generally not valid to use the additional interface boundary condition $(v)_{B4} = (v)_{B3}$ because the 2 BE-equations are defined at the same source point and are therefore similar. If such a condition would be used the remaining system of equations would become singular. However it would be possible to apply this condition if simultaneously one of the two BE-equations would be removed from the system. But then it would be a system with 3 remaining unknowns and 2 remaining equations and the additional interface boundary condition (4.26) would still have to be applied.

Case b: Prescribed tractions along the external boundary:

In this case known external tractions and forces are applied to the boundary as shown in figure 4.3b, which leads to the following additional equations and unknowns in y-direction at the corner nodes F3, B3 and B4:

Equations: 1 FE-equation (F3) + 2 BE-equations (B3 & B4) = 3 equations

Unknowns: $(v)_{F3}, (v)_{B3}, (v)_{B4}, (T_y)_{B4}, (R_y)_{F3}$ = 5 unknowns

Where $(R_y)_{F3}$ is the reaction force at node F3 which results from the coupling of the nodes F3 and B4. Now there is a situation with two unknowns too many and the following additional conditions can be applied:

$$(v)_{B4} = (v)_{F3} \quad (\text{alternatively } (v)_{B3} = (v)_{F3}) \quad (4.27)$$

$$(R_y)_{F3} = - \left[(C_{11})_{FE1} (T_y)_{B4} + (C_{12})_{FE1} (T_y)_{B5} + (C_{13})_{FE1} (T_y)_{B6} \right] \quad (4.28)$$

Which leads to a solvable system of equations.

4.5.3 Interface corner on an internal interface

If the interface corner is on an FEM/BEM-interface which is inside the domain as shown in figure 4.4 then the following equations and unknowns are introduced to the system in y-direction (the same is valid for the x-direction):

Equations: 1 FE-equation (F3) + 2 BE-equations (B3 & B4) = 3 equations

Unknowns: $(v)_{F3}, (v)_{B3}, (v)_{B4}, (T_y)_{B3}, (T_y)_{B4}, (R_y)_{F3}$ = 6 unknowns

Hence there are three unknowns too many. Two of them can be eliminated by using the following additional interface conditions:

$$(v)_{B4} = (v)_{F3} \quad (\text{alternatively } (v)_{B3} = (v)_{F3}) \quad (4.29)$$

$$(R_y)_{F3} = - \left[(C_{31})_{FE1} (T_y)_{B6} + (C_{32})_{FE1} (T_y)_{B5} + (C_{33})_{FE1} (T_y)_{B4} \right] - \left[(C_{11})_{FE2} (T_y)_{B3} + (C_{12})_{FE2} (T_y)_{B2} + (C_{13})_{FE2} (T_y)_{B1} \right] \quad (4.30)$$

Hence now there are 3 equations with 4 unknowns remaining and there are no possible additional interface conditions to be used.

A solution for this problem was found by studying methods for the solution of corner problems in pure BEM analysis. If a double corner node is used in pure BEM environment there are certain combinations of boundary conditions at the corner nodes which lead to more unknowns than equations as described by Mitra and Ingber (1993). Mitra and Ingber suggested to use an additional external source point near to the corner

location which brings an additional equation and the system of equations can be solved.

A similar approach was used in this work for coupled FEM/BEM analysis although the effect of the additionally introduced equation of the external source point on the result accuracy was not known in advance.

An additional external source point needs to be defined outside the boundary element subregion near to the interface corner. Such an external source point leads to the following additional boundary integral equations:

$$\sum_{e=1}^{n_e} h^{(e)}(x_i, y_i) \delta_e = \sum_{e=1}^{n_e} g^{(e)}(x_i, y_i) T_e + Q_i \quad (4.31)$$

which is equivalent to equation (3.18) with $C_i = 0$, which is valid at any point outside the boundary element subregion. Thus the finally obtained system of equations can be solved since there are 4 equations and 4 unknowns additionally introduced.

Numerical procedure suggested: In the procedure used in the developed program it is more convenient to use the additional condition $(v)_{B3} = (v)_{B4}$ and simultaneously delete one of the two BE equations at the corner (as explained in section 4.5.2). Then there are 3 equations (1FE, 1BE & 1BE- external) and 5 unknowns and two of the unknowns can be eliminated by applying the additional conditions (4.29) and (4.30). Thus there are 3 equations and 3 unknowns which means that the global system of equations does not increase in size which is very beneficial to the programming.

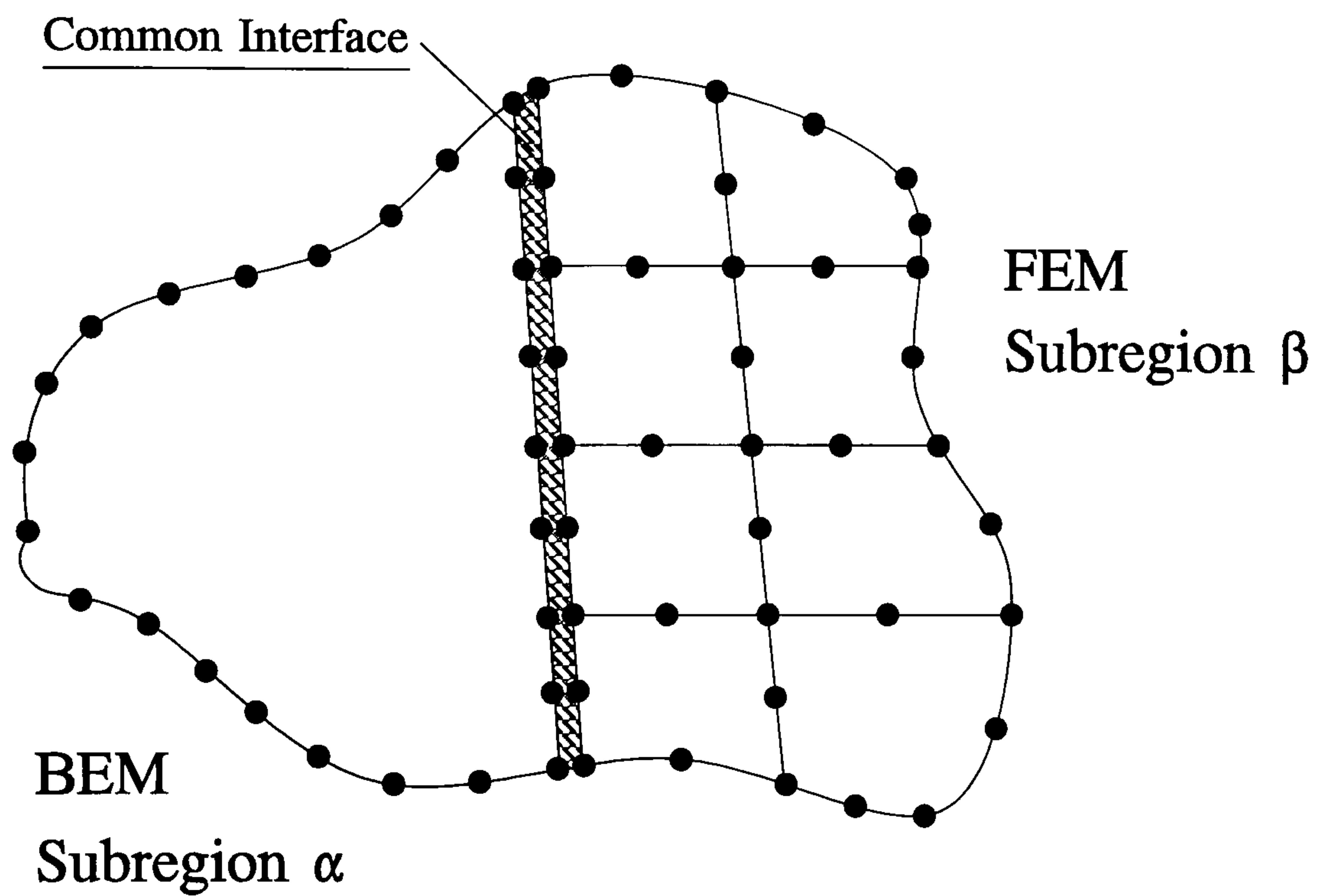


Figure 4.1 A domain which is subdivided into a BEM subregion and a FEM subregion.

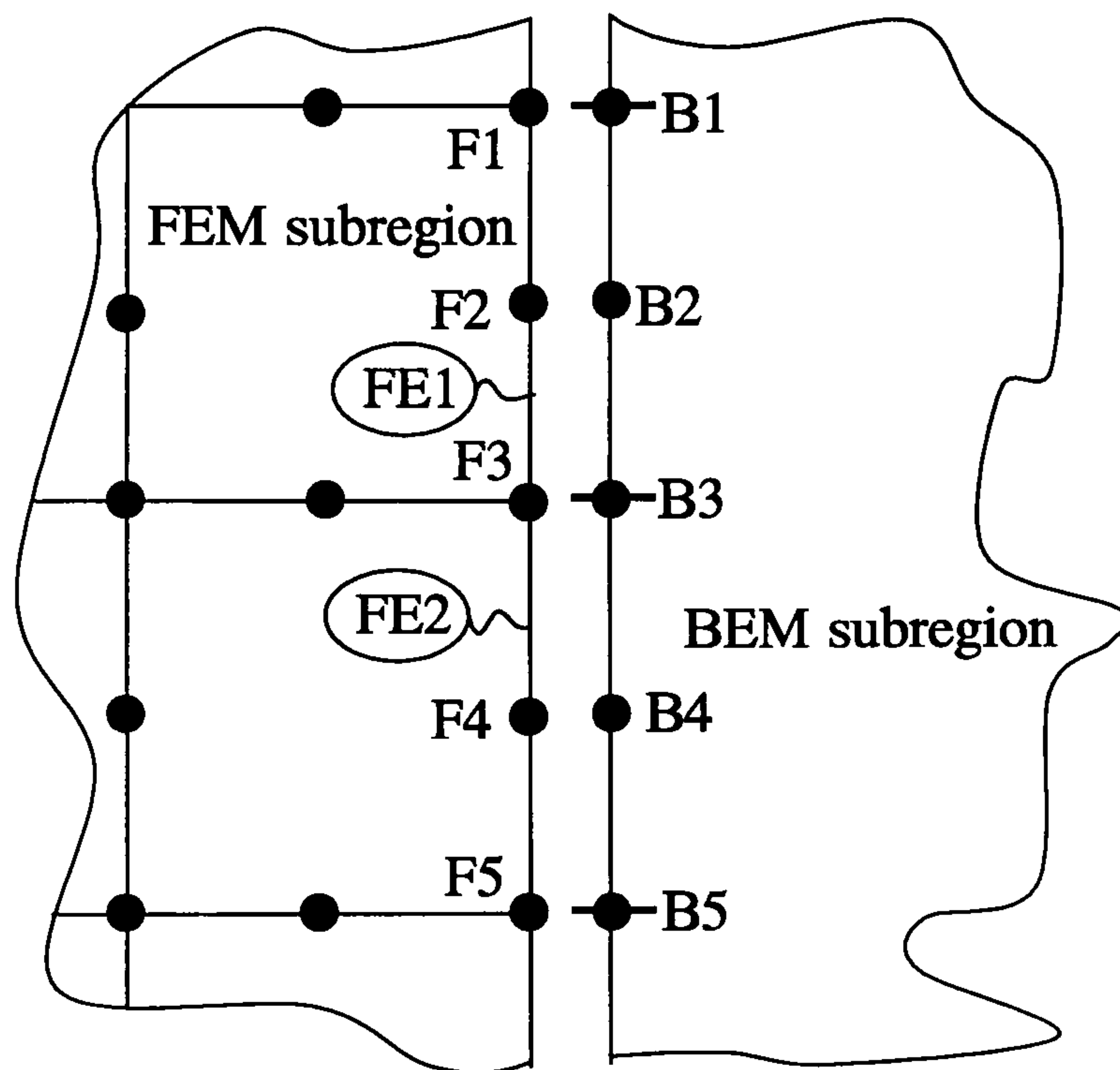


Figure 4.2 Interface for a case without a corner.

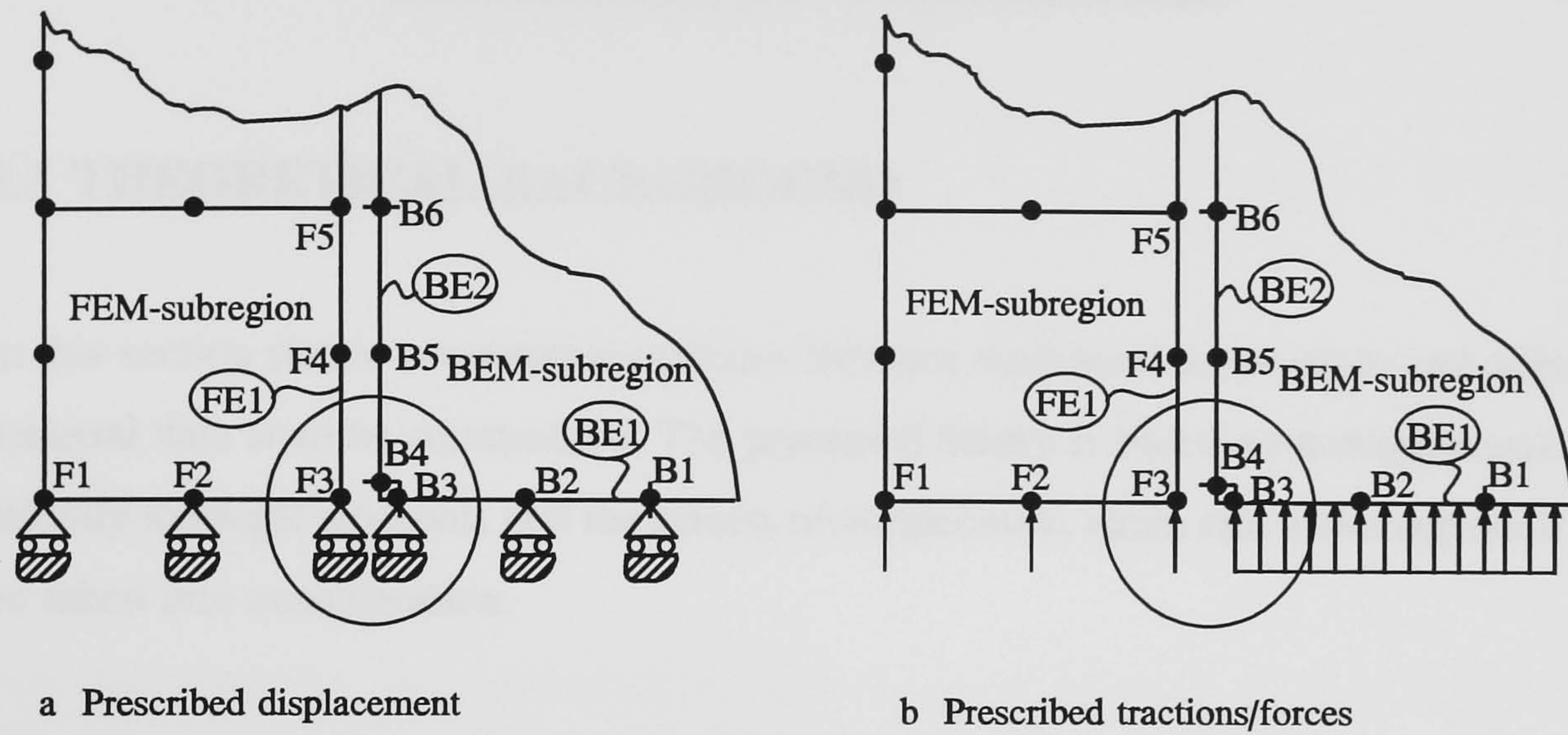


Figure 4.3 Corner modelling of an external interface corner.

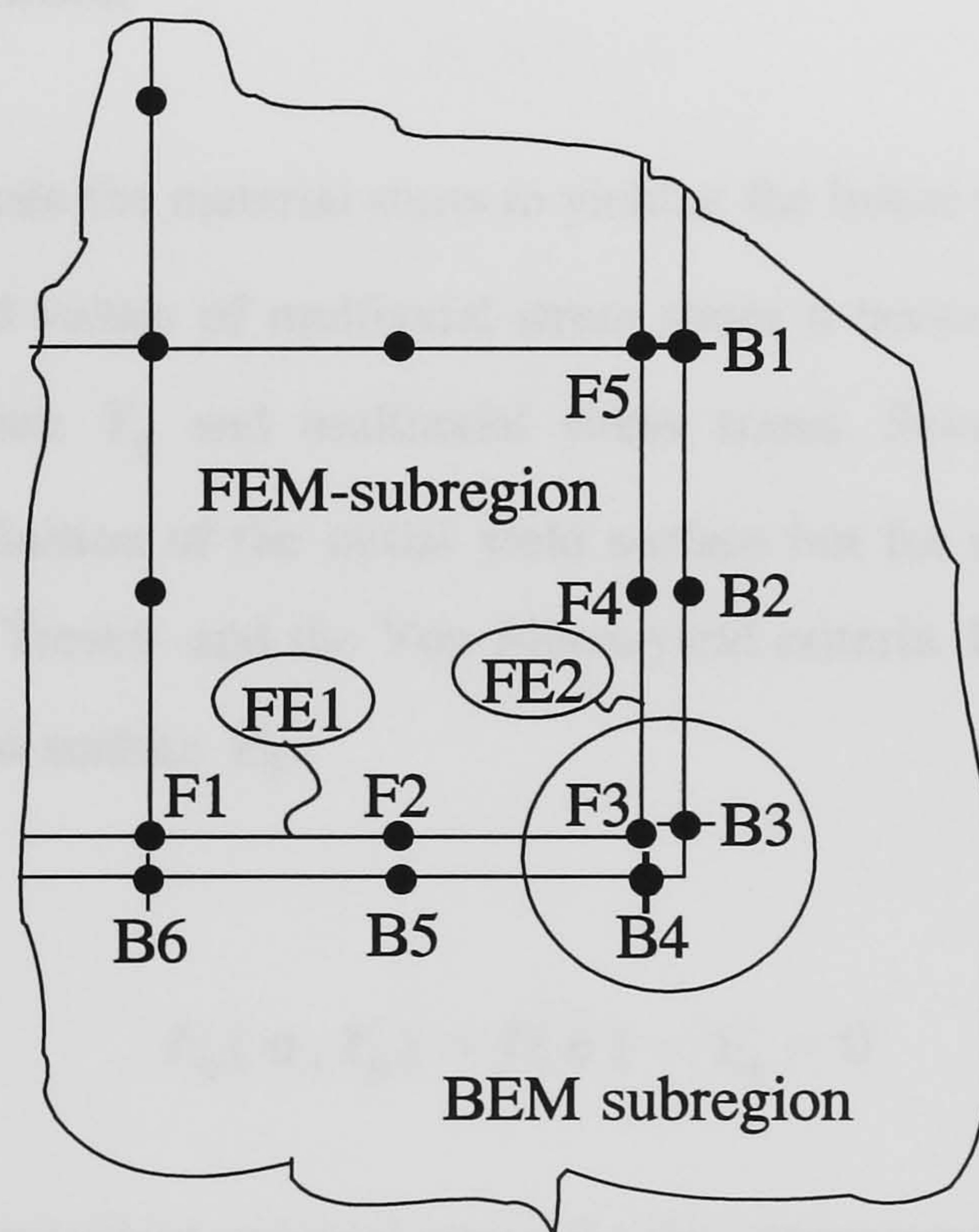


Figure 4.4 Corner modelling of an internal interface corner.

CHAPTER 5

PLASTICITY PROBLEMS

5.1 THEORETICAL BACKGROUND

In this section plastic constitutive relations between multiaxial stress states and uniaxial material data shall be summarised. The presented theory is based on homogeneous and initially isotropic materials and the effects of temperature, strain rate and creep shall not be taken into consideration.

There are three different rules and conditions which are necessary for the establishment of plastic constitutive relations, namely the initial yield condition, the flow rule and the hardening rule.

5.1.1 The yield condition

In a uniaxial stress state the material starts to yield at the initial yield stress Y_0 . In order to define initial yield values of multiaxial stress states it becomes necessary to define a relationship between Y_0 and multiaxial stress states. Several criteria have been proposed for the definition of the initial yield surface but for metal plasticity the two best verified are the Tresca- and the Von Mises-yield criteria. Both are defined by the following initial yield surface F_0 :

$$F_0(\boldsymbol{\sigma}, Y_0) = f(\boldsymbol{\sigma}) - Y_0 = 0 \quad (5.1)$$

where $f(\boldsymbol{\sigma})$ is the equivalent uniaxial stress for the assessment of multiaxial yielding. The following $f(\boldsymbol{\sigma})$ were suggested by Von Mises and Tresca:

$$f(\boldsymbol{\sigma})_{VON\ MISES} = \sqrt{3 J_2} \quad (5.2)$$

$$f(\boldsymbol{\sigma})_{TRESCA} = \sigma_1 - \sigma_3 \quad (5.3)$$

Where σ_1 and σ_3 are principal stresses and J_2 is a stress invariant (El-Zafrany (1998b)).

5.1.2 The flow rule

Once initial yielding has been exceeded, the initial yield surface F_0 no longer represents the actual multiaxial stress state. The more appropriate instantaneous yield surface F is defined. It takes the flow and hardening behaviour of the material into account and can be expressed as follows:

$$F(\boldsymbol{\sigma}, \mathbf{k}) = 0 \quad (5.4)$$

Where $\mathbf{k}$ is the hardening vector which is generally a function of the plastic strain $\boldsymbol{\epsilon}_p$ and a history parameter κ .

In order to explicitly define the instantaneous yield surface F a relationship between the plastic strain increment $d\boldsymbol{\epsilon}_p$ and the yield surface F needs to be established. This relationship is also known as flow rule and it is defined by the following equation:

$$d\boldsymbol{\epsilon}_p = d\lambda \frac{\partial F}{\partial \boldsymbol{\sigma}} = d\lambda \mathbf{a} \quad (5.5)$$

where

$$\mathbf{a} = \frac{\partial F}{\partial \boldsymbol{\sigma}} \equiv \frac{\partial f(\boldsymbol{\sigma})}{\partial \boldsymbol{\sigma}} \quad (5.6)$$

By starting from equation (5.5) an elastoplastic stress-strain matrix $\mathbf{D}_{ep}$ which defines the relationship between the multiaxial stress increment and the multiaxial strain increment can be derived. This leads to the following equation:

$$d\boldsymbol{\sigma} = \mathbf{D}_{ep} d\boldsymbol{\epsilon} \quad (5.7)$$

where:

$$\mathbf{D}_{ep} = \left(\mathbf{D} - \frac{\mathbf{D} \mathbf{a} \mathbf{a}^t \mathbf{D}}{A + \mathbf{a}^t \mathbf{D} \mathbf{a}} \right) \quad (5.8)$$

Equation (5.7) is of great practical importance since it can be used in incremental numerical procedures.

Notice that the parameter A in equation (5.8) represents the hardening terms and has to be evaluated according to the hardening rule applied as shown in the next section.

5.1.3 The hardening rules

There are two different hardening rules which are relevant to this work, namely the isotropic hardening rule and the kinematic hardening rule.

The theory of *isotropic hardening* states that during plastic flow the instantaneous yield surface expands uniformly in the stress space around the origin maintaining the same size, centre and orientation as the initial yield surface F_0 . This leads to the following hardening parameter A :

$$A = \left(\frac{\partial F}{\partial Y} \right)^2 \frac{dY}{d\epsilon_p} \quad (5.9)$$

Where $d\overline{\epsilon}_p$ is the effective plastic strain increment which is a uniaxial measure of the multiaxial plastic strain increment.

For the special case of a simple yield surface, such as the one obtained by the Von Mises yield criterion or the Tresca yield criterion, equation (5.9) simplifies such that:

$$A = H' \left(= \frac{dY}{d\overline{\epsilon}_p} \right) \quad (5.10)$$

where the strain hardening modulus H' can be evaluated according to the uniaxial stress strain diagram shown in figure 5.1 as follows:

$$H' = \frac{E_t}{1 - \frac{E_t}{E}} \quad (5.11)$$

E_t is the tangential modulus and is defined as follows:

$$E_t = \frac{d\sigma}{d\epsilon} \quad (5.12)$$

The theory of *kinematic hardening* states that during plastic deformation the instantaneous yield surface translates as rigid body in stress space by the shift vector α (Prager (1955)). During this translation the yield surface maintains its size, shape and orientation. There are two different definitions of the increment of translation $d\alpha$:

(i) Prager's definition (Prager (1955)) :

$$d\alpha = c d\epsilon_p \quad (5.13)$$

(ii) Ziegler's definition (Ziegler (1959)) :

$$d \boldsymbol{\alpha} = d \mu (\boldsymbol{\sigma} - \boldsymbol{\alpha}) \quad (5.14)$$

Where c and $d \mu$ are parameters defining the hardening behaviour of the material. Subsequent derivations and definitions lead to the following hardening parameter A :

$$A = c \boldsymbol{a}^t \boldsymbol{a} \quad (5.15)$$

where

$$c = \frac{2}{3} H' \quad (5.16)$$

It is also possible to use a mixture of isotropic and kinematic hardening which is often the most realistic model of the hardening behaviour of the material. *Mixed hardening* leads to the following hardening parameter A in terms of the mixed hardening parameter M :

$$A = (1 - M) A_{kinematic} + M A_{isotropic} \quad (5.17)$$

5.2 ELASTOPLASTIC FEM FORMULATION

5.2.1 Generalised equations of equilibrium

If a three-dimensional component is subjected to a virtual displacement field $\Delta \vec{q}(x, y, z)$, then the resulting change in the strain energy of the component may be expressed in terms of FE-contributions as follows:

$$\Delta U = \sum_{e=1}^m \int \int \int_{\text{element } e} \Delta \boldsymbol{\epsilon}^t \boldsymbol{\sigma} dx dy dz \quad (5.18)$$

The elemental displacement vector $\Delta \delta_e$ can be interpolated over each finite element and a B -matrix can be derived such that:

$$\Delta \epsilon = B \Delta \delta_e \quad (5.19)$$

and hence equation (5.18) may be rewritten as follows:

$$\begin{aligned} \Delta U &= \sum_{e=1}^m \int \int \int_{\text{element } e} \Delta \delta_e^t B^t \sigma \, dx \, dy \, dz \\ &\equiv \Delta \delta^t \left[\sum_{e=1}^m \int \int \int_{\text{element } e} B^t \sigma \, dx \, dy \, dz \right] \end{aligned} \quad (5.20)$$

The work done by the actual applied forces can be expressed as follows:

$$\Delta W = \Delta \delta^t F \quad (5.21)$$

Using the principle of virtual work, then it can be deduced that:

$$\Delta \chi = \Delta U - \Delta W = 0 \quad (5.22)$$

$$\text{i.e.} \quad \Delta \delta^t \left[\sum_{e=1}^m \int \int \int_{\text{element } e} B^t \sigma \, dx \, dy \, dz - F \right] = 0 \quad (5.23)$$

and therefore:

$$\sum_{e=1}^m \int \int \int_{\text{element } e} B^t \sigma \, dx \, dy \, dz - F = \mathbf{0} \quad (5.24)$$

Equation (5.24) represents the generalised equation of equilibrium which is valid for linear and nonlinear conditions.

5.2.2 Linearisation of the non-linear equilibrium equations

The non-linear equations of equilibrium (5.24) can be linearised in an iterative procedure which shall be presented in this section.

1st Iteration:

Consider the approximate solutions $\delta_1, \epsilon_1, \sigma_1$ and B_1 . It is clear that the error introduced into the equilibrium equation (5.24) by this first approximation can be expressed in terms of the following initial residual force vector R_1 :

$$R_1 = F - \sum_{e=1}^m \int \int \int_{\text{element } e} B_1^t \sigma_1 dx dy dz \neq \mathbf{0} \quad (5.25)$$

Solutions which satisfy the equations of equilibrium exactly can be defined such that:

$$\delta = \delta_1 + (\Delta \delta)_1 \quad \Leftrightarrow \quad \delta_1 = \delta - (\Delta \delta)_1 \quad (5.26)$$

$$\epsilon = \epsilon_1 + (\Delta \epsilon)_1 \quad \Leftrightarrow \quad \epsilon_1 = \epsilon - (\Delta \epsilon)_1 \quad (5.27)$$

$$\sigma = \sigma_1 + (\Delta \sigma)_1 \quad \Leftrightarrow \quad \sigma_1 = \sigma - (\Delta \sigma)_1 \quad (5.28)$$

For infinitesimal strain conditions it can be proved that $(\Delta B)_1 = 0$ and hence:

$$B = B_1 + (\Delta B)_1 \quad \Rightarrow \quad B_1 = B \quad (5.29)$$

Substituting (5.28) and (5.29) into (5.25):

$$R_1 = \left[F - \sum_{e=1}^m \int \int \int_{\text{element } e} B^t \sigma dx dy dz \right] + \sum_{e=1}^m \int \int \int_{\text{element } e} B^t (\Delta \sigma)_1 dx dy dz \quad (5.30)$$

where in equation (5.30) the bracketed term is equal to zero according to equation (5.24), hence:

$$\sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\Delta \boldsymbol{\sigma})_1 dx dy dz - \mathbf{R}_1 = \mathbf{0} \quad (5.31)$$

which represents the equation of equilibrium for the second iteration.

nth Iteration:

The equation of equilibrium established in the previous equation is:

$$\sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\Delta \boldsymbol{\sigma})_{n-1} dx dy dz - \mathbf{R}_{n-1} = \mathbf{0} \quad (5.32)$$

Consider approximate solutions $\boldsymbol{\delta}_n, \boldsymbol{\epsilon}_n$ and $\boldsymbol{\sigma}_n$ of the equilibrium equation (5.32). The residual force vector $\mathbf{R}_n$ is equal to:

$$\mathbf{R}_n = \mathbf{R}_{n-1} - \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t \boldsymbol{\sigma}_n dx dy dz \neq \mathbf{0} \quad (5.33)$$

Solutions which satisfy the equation of equilibrium (5.32) exactly are:

$$(\Delta \boldsymbol{\delta})_{n-1} = \boldsymbol{\delta}_n + (\Delta \boldsymbol{\delta})_n \quad (5.34)$$

$$(\Delta \boldsymbol{\epsilon})_{n-1} = \boldsymbol{\epsilon}_n + (\Delta \boldsymbol{\epsilon})_n \quad (5.35)$$

$$(\Delta \boldsymbol{\sigma})_{n-1} = \boldsymbol{\sigma}_n + (\Delta \boldsymbol{\sigma})_n \quad (5.36)$$

and substitution of equation (5.36) into (5.32) yields:

$$\sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\Delta \boldsymbol{\sigma})_n dx dy dz - \mathbf{R}_n = \mathbf{0} \quad (5.37)$$

which represents the equation of equilibrium for the (n+1)th iteration.

End of iterations

By substituting the results of all previous iterations up to n , equations (5.34)-(5.36), into equations (5.26)-(5.28) the following results can be established:

$$\boldsymbol{\delta} = \boldsymbol{\delta}_1 + \boldsymbol{\delta}_2 + \boldsymbol{\delta}_3 + \dots + \boldsymbol{\delta}_n + (\Delta \boldsymbol{\delta})_n \quad (5.38)$$

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}_1 + \boldsymbol{\epsilon}_2 + \boldsymbol{\epsilon}_3 + \dots + \boldsymbol{\epsilon}_n + (\Delta \boldsymbol{\epsilon})_n \quad (5.39)$$

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_1 + \boldsymbol{\sigma}_2 + \boldsymbol{\sigma}_3 + \dots + \boldsymbol{\sigma}_n + (\Delta \boldsymbol{\sigma})_n \quad (5.40)$$

It is the aim of the iterative procedure to obtain converging results, which can be achieved in equations (5.38)-(5.40) if:

$$|\boldsymbol{\delta}_1| > |\boldsymbol{\delta}_2| > |\boldsymbol{\delta}_3| > \dots > |\boldsymbol{\delta}_n| \quad (5.41)$$

$$|\boldsymbol{\epsilon}_1| > |\boldsymbol{\epsilon}_2| > |\boldsymbol{\epsilon}_3| > \dots > |\boldsymbol{\epsilon}_n| \quad (5.42)$$

$$|\boldsymbol{\sigma}_1| > |\boldsymbol{\sigma}_2| > |\boldsymbol{\sigma}_3| > \dots > |\boldsymbol{\sigma}_n| \quad (5.43)$$

and it is clear that the iterations may be ended whenever $|\boldsymbol{\delta}_n|$ is acceptably small compared to $|\boldsymbol{\delta}|$ and similarly for the strains and stresses.

5.2.3 Plasticity algorithm

It is possible to apply the linearisation procedure of the equilibrium equation (5.24) which was presented in section 5.2.2 to plasticity problems. Within each iteration the necessary results and parameters can be evaluated as described in this section.

An approximate displacement vector $\boldsymbol{\delta}_n$ of the n -th iterative step for the solution of the equilibrium equations established in the previous iteration (equation (5.32)) can be

evaluated by means of the following finite element equation:

$$\mathbf{K} \boldsymbol{\delta}_n = \mathbf{R}_{n-1} \quad (5.44)$$

where $\mathbf{K}$ is the elastic stiffness matrix. Notice that it is also possible to use a global stiffness matrix based on the elastoplastic elemental stiffness matrix $(\mathbf{K}_{ep})_e$, which is defined as follows:

$$(\mathbf{K}_{ep})_e = \int \int \int_{\text{element } e} \mathbf{B}^t \mathbf{D}_{ep} \mathbf{B} \, dx \, dy \, dz \quad (5.45)$$

The hardening and flow parameters of the elastoplastic stress strain matrix $\mathbf{D}_{ep}$ can be evaluated according to the yield surface at the beginning of the iteration.

It is clear that equation (5.44) can never supply accurate results. This is shown for a uniaxial case in figure 5.2, where $\boldsymbol{\delta}_e$ is the displacement result obtained by using the elastic stiffness matrix, whilst $\boldsymbol{\delta}_{ep}$ is the result obtained by using the elastoplastic stiffness matrix.

Uncorrected stress and strain results $\boldsymbol{\sigma}'_n$ and $\boldsymbol{\epsilon}'_n$ may be evaluated as follows:

$$\boldsymbol{\epsilon}'_n = \mathbf{B} \boldsymbol{\delta}_n \quad (5.46)$$

$$\boldsymbol{\sigma}'_n = \mathbf{D} \boldsymbol{\epsilon}_n \quad (5.47)$$

where in equation (5.47) $\mathbf{D}_{ep}$ has to be used instead of $\mathbf{D}$ if equation (5.44) was based on the elastoplastic stiffness matrix.

The stress and strain results are denoted "uncorrected" because they do not accurately correspond to the actual stresses and strains.

It is possible to keep the strain unchanged and to do corrections such that an accurately corresponding stress is obtained by applying an initial stress in the next iteration. This procedure is known as *initial stress algorithm* and leads to the following residual force vector for the next iteration as shown in Appendix D:

$$\mathbf{R}_n^{(\sigma)} = \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\Delta \boldsymbol{\sigma})_n dx dy dz \quad (5.48)$$

Alternatively the stress can be kept unchanged and the strain can be corrected by applying an initial strain in the next iteration. This procedure is known as *initial strain algorithm* and leads to the following residual force vector as shown in Appendix D:

$$\mathbf{R}_n^{(\epsilon)} = \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\boldsymbol{\epsilon}_p)_n dx dy dz \quad (5.49)$$

One of these residual force vectors may be used in order to define the equilibrium equation (5.44) for the next iteration.

It is important to notice that convergence is the most important issue within plasticity iterations. Guzelbey (1992) realised that for a small degree of hardening of the material, the initial strain procedure delivers diverging results. Therefore the initial strain procedure can only be used for cases with $H' \gg 0$.

5.3 ELASTOPLASTICITY WITH COUPLED FEM/BEM

For elastoplastic analysis with coupled FEM/BEM, regions which are thought to develop plastic material behaviour are modelled with the FEM whilst the elastic remainder of the structure is modelled with the BEM.

The iterative plasticity procedure which is used is different to a standard FE-plasticity procedure since the effect of the plastic deformation on the BE-subregions needs to be taken into account. The following iterative plasticity procedure is suggested:

Step 1: Procedure initiation

As with any FEM or BEM analysis the domain has to be meshed first. It may prove useful to run a pure elastic BEM analysis first in order to get an initial idea where the plastic zones are expected. Then these plastic zones will be modelled with an appropriate FE-mesh in order to account for occurring high stress gradients. The elastic remainder of the structure is to be modelled with BEM and special attention has to be drawn to the FEM/BEM interface which has to be modelled as described in chapter 4.

One may divide the load and prescribed displacements, which are defined by the Y -term in equation (4.21b) into p main increments, such that:

$$Y^{(j)} = Q^{(j)} Y \quad (5.50)$$

$$\text{with} \quad \sum_{j=1}^p Q^{(j)} = 1 \quad (5.51)$$

For each of the p main increments the following steps are to be applied.

Step 2: Solve for an initial elastic analysis

The matrix equations of the coupled system (equation (4.21)), is to be solved with elastic stiffness matrix contributions in the first iteration step as follows:

$$\mathbf{A} \mathbf{X}_1 = \mathbf{Y} \quad (5.52)$$

Hence an initial elastic solution for the displacements and reaction forces and tractions is obtained.

Step 3: Yield check

The results are used to obtain the elastic uncorrected stresses $\boldsymbol{\sigma}'$ and uncorrected strains $\boldsymbol{\epsilon}'$ of the FEM-subregion as described in section 5.2.3. Then at each Gaussian point in the FEM-subregion, a proper yield criterion such as the Von Mises or Tresca criterion is employed in order to check whether the initial yield stress of the material has been exceeded at any point. If this is the case the initial elastic results were not accurate and therefore plasticity corrections become necessary as described in step 4. Otherwise, the obtained elastic results are accurate and results are updated for the next main load increment to be applied.

Step 4: Evaluate the necessary stress or strain corrections

If the results obtained are not accurate yet, we need to do corrections according to the amount of plasticity involved. In accordance to the finite element initial stress and initial strain procedures shown in Appendix D, a residual force vector (correction force vector) can be established by substituting the FEM stress or strain results in equations (5.48) for initial stress or equation (5.49) for initial strain. This leads to the correction force vector

of the previously obtained results in the FEM subregion $\mathbf{R}_{n-1}^{(\sigma)}$ (or $\mathbf{R}_{n-1}^{(\epsilon)}$ for initial strain).

Step 5: Solve the coupled FEM/BEM system according to the plasticity corrections

It is clear that the following equation needs to be solved within the n -th iteration in order to obtain results $\mathbf{X}_n$, which correspond to the evaluated correction force vector:

$$\mathbf{A} \mathbf{X}_n = \mathbf{Y}_{n-1} \quad (5.53)$$

where for initial stress:

$$\mathbf{Y}_{n-1} = \begin{bmatrix} \mathbf{0} \\ (\mathbf{R}_{\beta\alpha})_{n-1}^{(\sigma)} \\ (\mathbf{R}_{\beta\beta})_{n-1}^{(\sigma)} \end{bmatrix} \quad (5.54)$$

and for initial strain:

$$\mathbf{Y}_{n-1} = \begin{bmatrix} \mathbf{0} \\ (\mathbf{R}_{\beta\alpha})_{n-1}^{(\epsilon)} \\ (\mathbf{R}_{\beta\beta})_{n-1}^{(\epsilon)} \end{bmatrix} \quad (5.55)$$

The correction force vector of the FEM subregion $(\mathbf{R})_{n-1}^{(\sigma/\epsilon)}$ needs to be partitioned such that $(\mathbf{R}_{\beta\alpha})_{n-1}^{(\sigma/\epsilon)}$ represents FEM/BEM-interface components of the residual force vector and $(\mathbf{R}_{\beta\beta})_{n-1}^{(\sigma/\epsilon)}$ contains the independent components of the FEM subregion β .

Step 6: Accuracy check

A measure for the accuracy of a converging solution are the following 2 non-dimensional parameters:

$$e_X = \frac{\mathbf{X}_n^t \mathbf{X}_n}{\mathbf{X}^t \mathbf{X}} \quad (5.56)$$

$$e_Y = \frac{\mathbf{Y}_n^t \mathbf{Y}_n}{\mathbf{Y}^t \mathbf{Y}} \quad (5.57)$$

with:
$$\mathbf{X} \approx \mathbf{X}_1 + \mathbf{X}_2 + \mathbf{X}_3 + \dots + \mathbf{X}_n \quad (5.58)$$

Where e_X is a measure of the error in the general results whilst e_Y is a measure for the error in the residual forces. Acceptable accuracy is achieved if both parameters become less than a small number. If accuracy is achieved, the next main increment may be applied. If not, another stress or strain correction iteration has to be run starting from step 4.

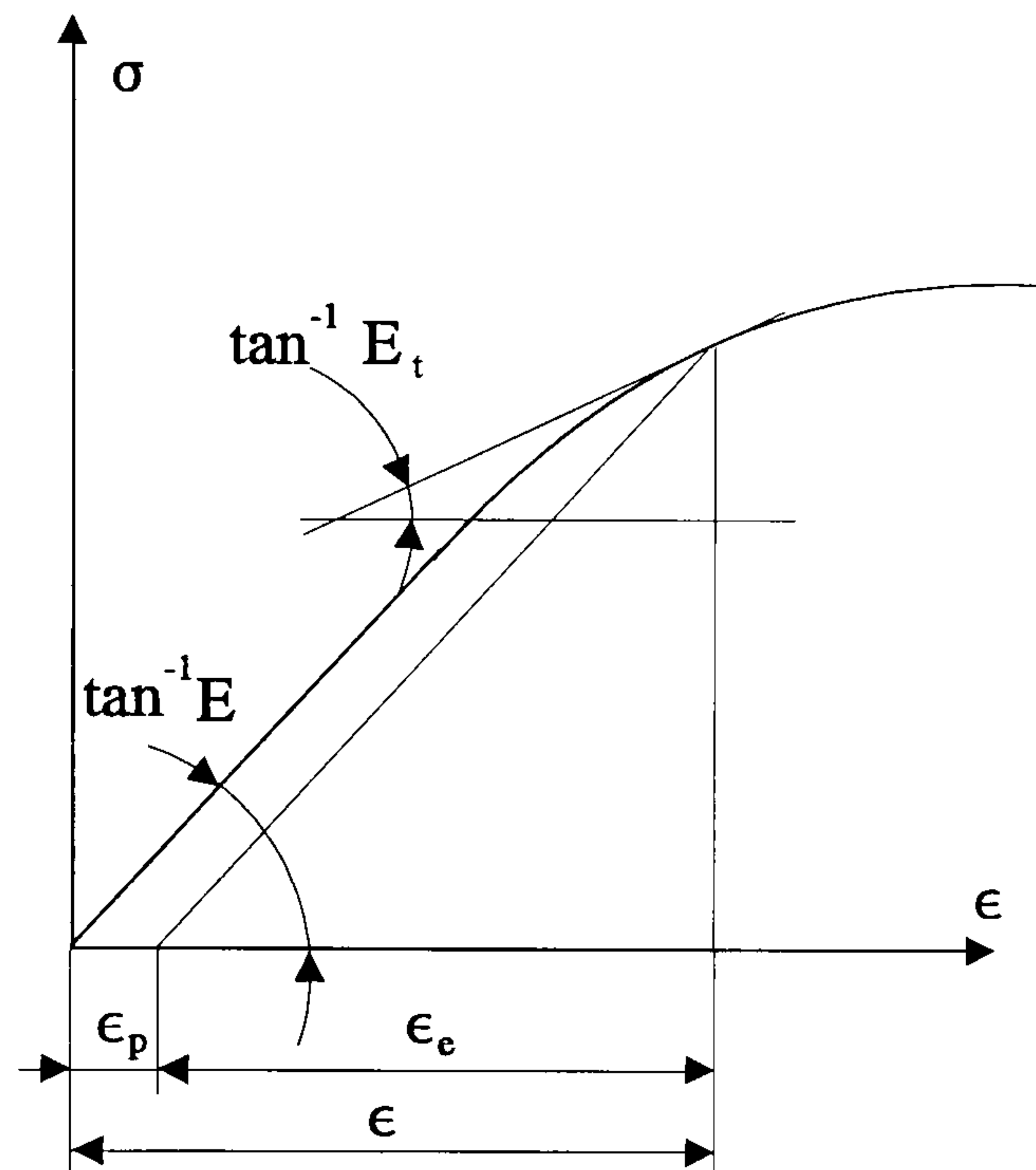


Figure 5.1 Uniaxial stress-strain-diagram.

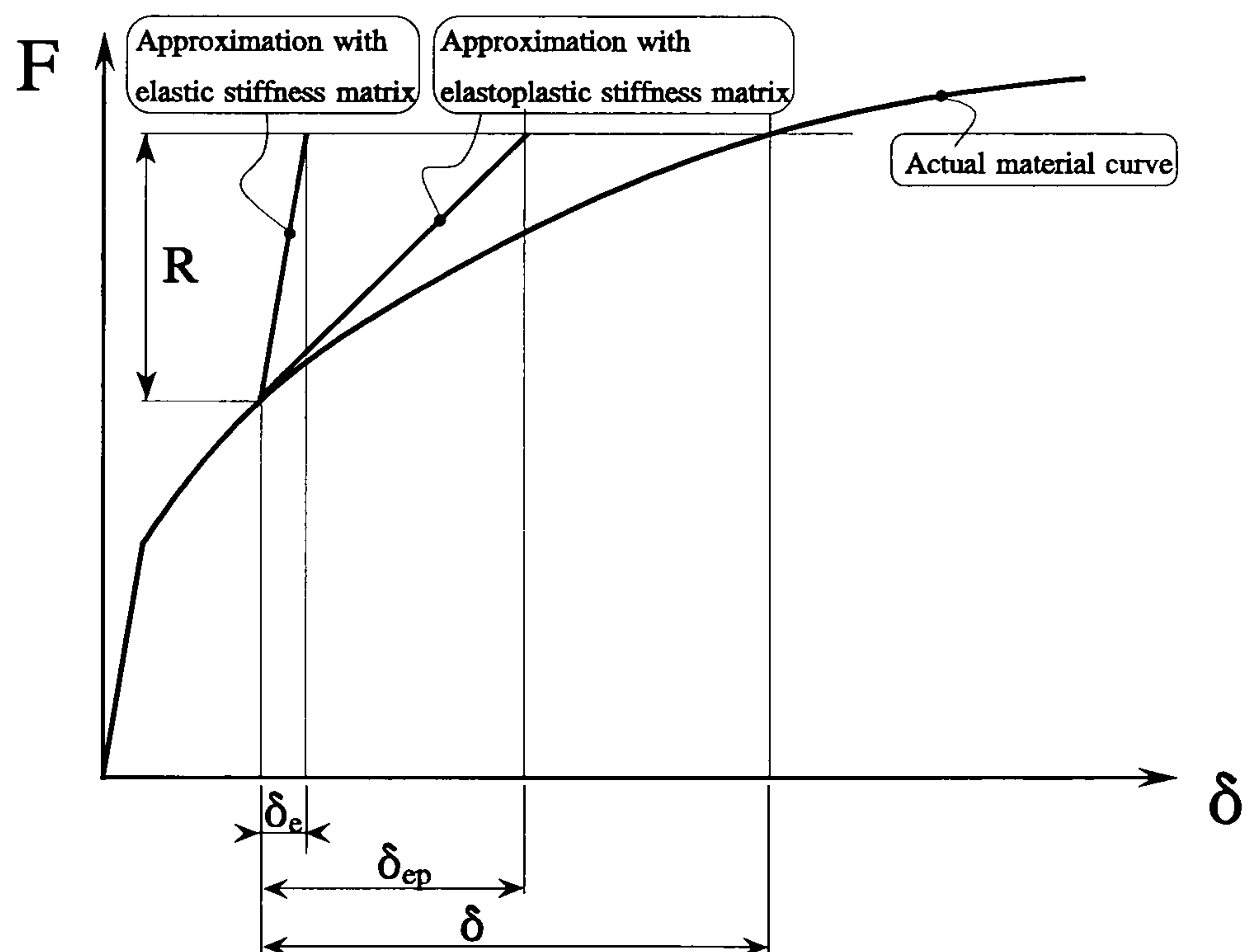


Figure 5.2 Approximation of uniaxial plastic material behaviour with analysis using the elastic stiffness matrix and the elastoplastic stiffness matrix.

CHAPTER 6

CONTACT ANALYSIS WITH GAP FINITE ELEMENTS

6.1 THE NODE-TO-NODE GAP FINITE ELEMENT

6.1.1 Frictionless contact analysis

Consider two boundaries of two Finite Element or Boundary Element meshes where node I and node J are in the contact zone as shown in figure 6.1. Normal and tangential directions can be defined according to the normal and tangential directions of node I . A gap element can be thought of as two independent linear springs which are orientated in the normal and tangential directions. These springs connect node I and node J and have stiffnesses k_n and k_t in the normal and tangential directions, respectively. Therefore, the gap element stiffness equations can be expressed in the following matrix form:

$$\mathbf{K}_{e_L} \boldsymbol{\delta}_{e_L} = \mathbf{F}_{e_L} \quad (6.1)$$

where the force and displacement vectors are defined as

$$\mathbf{F}_{e_L} = \left\{ (F_n)_I \quad (F_t)_I \quad (F_n)_J \quad (F_t)_J \right\} \quad (6.2)$$

$$\boldsymbol{\delta}_{e_L} = \left\{ (u_n)_I \quad (u_t)_I \quad (u_n)_J \quad (u_t)_J \right\} \quad (6.3)$$

and the gap element stiffness matrix will be:

$$\mathbf{K}_{e_L} = \begin{bmatrix} k_n & 0 & -k_n & 0 \\ 0 & k_t & 0 & -k_t \\ -k_n & 0 & k_n & 0 \\ 0 & -k_t & 0 & k_t \end{bmatrix} \quad (6.4)$$

Subscripts n and t indicate normal and tangential components and subscript L means that the matrices are defined with respect to a local coordinate system, i.e. the normal and tangential directions of node I .

The idea of the gap element is to change its stiffness according to the current contact condition (or gap status) of node I and node J . Considering a frictionless contact problem, there are two different contact conditions:

- (i) There is a gap in the normal direction between node I and node J . This case can be modelled by assigning very low stiffnesses k_n and k_t to the element.
- (ii) The gap is closed which means that node I and J are in contact. For this case a very high normal stiffness k_n and a very low tangential stiffness k_t are to be assigned.

6.1.2 Frictional contact analysis

For frictional contact analysis a closed gap element can be either in sliding condition or in sticking condition. The sticking condition can simply be modelled by assigning very high stiffnesses k_n and k_t to the gap element in order to restrain the relative normal and tangential movement. However, the modelling of a sliding gap element is much more difficult and one of the following two methods may be used:

Method 1: Change the tangential gap element stiffness in an iterative procedure

The tangential gap element stiffness of the gap element defines the sliding behaviour of the two connected nodes. If there is a high k_t defined, then the amount of relative sliding between the nodes will be small, and, vice versa a low k_t leads to large sliding.

It is possible to adjust the tangential stiffness in an iterative procedure such that the Coulomb friction law is satisfied:

$$F_t = \mu F_n \quad (6.5)$$

Within such an iterative procedure initially an arbitrary k_t is chosen and the resulting set of equations is solved. Then a more appropriate k_t can be found for the next iteration by using the following equation in terms of the obtained results F_n and Δu_t :

$$(k_t)_{next} = \left| \frac{(F_t)_{desired}}{\Delta u_t} \right| = \left| \frac{\mu F_n}{\Delta u_t} \right| \quad (6.6)$$

where:

$$\Delta u_t = (u_t)_I - (u_t)_J \quad (6.7)$$

and $F_n = (F_n)_I = - (F_n)_J$ is the normal force acting on node I of the gap element.

An accurate solution is obtained if k_t does not greatly change compared to k_t of the previous iteration, i.e. if equation (6.5) is satisfied for the current iteration.

Although this method promises an accurate linear approximation of the actual non-linear frictional behaviour, it was found that convergence is difficult to reach for actual contact cases. Whenever there is more than one gap element defined across the contact interface, changes in the tangential stiffnesses of some of these gap elements naturally also influence the sliding behaviour of the other gap elements. This influence is difficult if not even impossible to account for in the iterative procedure, and hence convergence can not often be achieved.

Method 2: Modify the gap element matrices

The Coulomb friction law defined by equation (6.5) may be taken into account by using the following gap element matrix in equation (6.1):

$$\mathbf{K}_{e_L} = \begin{bmatrix} k_n & 0 & -k_n & 0 \\ \mu k_n & k_t & -\mu k_n & -k_t \\ -k_n & 0 & k_n & 0 \\ -\mu k_n & -k_t & \mu k_n & k_t \end{bmatrix} \quad (6.8)$$

Writing the first two rows of equation (6.1) with stiffness matrix as defined in (6.8) as separate equations yields:

$$k_n ((u_n)_I - (u_n)_J) = (F_n)_I \quad (6.9)$$

$$\mu k_n ((u_n)_I - (u_n)_J) + k_t ((u_t)_I - (u_t)_J) = (F_t)_I \quad (6.10)$$

By choosing a negligibly small k_t and by using equation (6.9), equation (6.10) can be rewritten as follows:

$$\mu (F_n)_I = (F_t)_I \quad (6.11)$$

which defines the Coulomb friction criterion at node I . Similarly for node J :

$$\mu (F_n)_J = (F_t)_J \quad (6.12)$$

Hence it is clear that by choosing a very small coefficient k_t , the stiffness matrix of equation (6.8) can model Coulomb friction. Notice that the coefficient k_t cannot be equal to zero because then there is a zero value on the diagonal of the stiffness matrix, which makes the structural system of equations singular.

Equations (6.11) and (6.12) define the nodal tangential forces of node I and node J . They are equal and opposite as it would be expected. However, it is so far not secured that these frictional forces are applied in the direction opposite to the tangential movement of the contacting surfaces. From figure 6.1 it can be seen that there are two possibilities:

$$(i) (u_t)_I < (u_t)_J$$

In this case F_t at node I has to be positive and of course F_t at node J equal and opposite. Since F_n at node I is positive (for contacting surfaces) it can be seen from equation (6.11) that here $\mu \equiv +\mu_{sl}$ in equation (6.8) for sliding contact (where μ_{sl} is the coefficient of sliding friction).

$$(ii) (u_t)_I > (u_t)_J$$

In this case F_t at node I has to be negative and of course F_t at node J equal and opposite. Since F_n at node I is positive (for contacting surfaces), it can be seen from equation (6.11) that here $\mu \equiv -\mu_{sl}$ in equation (6.8) for sliding contact.

Hence in order to accurately account for the directions of the frictional forces within the current load increment, it is necessary to assume a direction first (and hence the corresponding μ) and then correct it in a reiteration if it was not accurate.

Notice that within the following sections on contact analysis and case studies method 2 was used exclusively because of the convergence problems of the first method.

6.2 THE IMPLEMENTATION OF A GAP ELEMENT SUBREGION INTO THE COUPLED FEM/BEM PACKAGE

Consider an artificial contact subregion consisting of r nodes and $r/2$ node-to-node gap elements as shown in figure 6.2. Each of the nodes corresponds to a node on the contacting surfaces. Such an artificial gap element subregion can be defined between two finite element subregions, two boundary element subregions or one finite element subregion and one boundary element subregion.

The elemental gap element equation of each of the gap elements in this subregion was already defined to be:

$$\mathbf{K}_{e_L} \boldsymbol{\delta}_{e_L} = \mathbf{F}_{e_L} \quad (6.1)$$

The structural matrices of the coupled FEM/BEM method presented in this work are based on x- and y-components and hence the above equation needs to be rotated as shown in Appendix E. This leads to the following gap element equation in terms of x- and y-components (with subscript G):

$$\mathbf{K}_{e_G} \boldsymbol{\delta}_{e_G} = \mathbf{F}_{e_G} \quad (6.13)$$

with:

$$\mathbf{K}_{e_G} = \boldsymbol{\Theta}^t \mathbf{K}_{e_L} \boldsymbol{\Theta} \quad (6.14)$$

Where $\boldsymbol{\Theta}$ is the rotation matrix as defined in Appendix E.

Each of the gap element equations can be rotated and assembled into subregional equations for the artificial gap element subregion γ such that:

$$\mathbf{K}_{\gamma} \boldsymbol{\delta}_{\gamma} = \mathbf{F}_{\gamma} \quad (6.15)$$

It is clear that equation (6.15) is not compatible to the BE-equations, and therefore a transformation of the nodal forces $\mathbf{F}_\gamma$ into nodal tractions is needed. The boundary of the artificial gap subregion can be discretised into s artificial boundary elements as shown in figure 6.3. The number of nodes of each of these artificial boundary elements should correspond to the number of nodes on the boundary element or finite element side it is connected to. Then the following transformation equation can be derived for each of these artificial boundary elements as shown in Appendix C:

$$\mathbf{F}_{(e)} = \mathbf{C}_{(e)} \mathbf{T}_{(e)} \quad (4.22)$$

By assembling the transformation equation (4.22) of each of the s artificial boundary elements into subregional equations:

$$\mathbf{F}_\gamma = \mathbf{C}_\gamma \mathbf{T}_\gamma \quad (6.16)$$

And by substituting (6.16) into (6.15) and defining $\mathbf{H}_\gamma = \mathbf{K}_\gamma$ and $\mathbf{G}_\gamma = \mathbf{C}_\gamma$:

$$\mathbf{H}_\gamma \boldsymbol{\delta}_\gamma = \mathbf{G}_\gamma \mathbf{T}_\gamma \quad (6.17)$$

Equation (6.17) represents the resulting set of equations for the artificial gap element subregion. It has BEM format and it can therefore be treated in the same way as any other BEM-subregion. Partitioning equation (6.17) according to the interfaces to the connected FEM-or BEM-subregions α and β as shown in figure 6.2 yields:

$$\mathbf{H}_{\gamma\alpha} \boldsymbol{\delta}_{\gamma\alpha} + \mathbf{H}_{\gamma\beta} \boldsymbol{\delta}_{\gamma\beta} = \mathbf{G}_{\gamma\alpha} \mathbf{T}_{\gamma\alpha} + \mathbf{G}_{\gamma\beta} \mathbf{T}_{\gamma\beta} \quad (6.18)$$

Moreover the following interface conditions can be defined for the satisfaction of equilibrium and compatibility requirements along the interface of the gap element subregion and the contacting subregions α and β :

$$\delta_{\gamma\alpha} = \delta_{\alpha\gamma} \quad (6.19)$$

$$\delta_{\gamma\beta} = \delta_{\beta\gamma} \quad (6.20)$$

$$\mathbf{T}_{\gamma\alpha} = -\mathbf{T}_{\alpha\gamma} \quad (6.21)$$

$$\mathbf{T}_{\gamma\beta} = -\mathbf{T}_{\beta\gamma} \quad (6.22)$$

Then it is clear that equation (6.18) can then be implemented into the structural system of equations by condensing interface degrees of freedom when applying the above interface conditions. This condensation can be undertaken for the coupling to an FEM subregion (α or β) in a way similar to the one presented in section 4.3 or for the coupling to a BEM subregion (α or β) similarly to section 3.3.2.

Hence, although all degrees of freedom of the artificial gap element subregion will be condensed, the stiffnesses of the gap elements and the coefficient of the transformation matrices will be assembled into the structural matrix and a solution can be obtained. An example of this procedure is given in Appendix F.

6.3 GENERALISED GAP ELEMENT ALGORITHM

In this section a contact algorithm shall be presented, which is capable of modelling frictional and frictionless elastoplastic or purely elastic contact with gap finite elements. It can be summarised from section 6.1 that for each gap element with stiffness matrix as in equation (6.8) there are three different contact conditions which may occur and which can be modelled as follows:

- (i) *Separation:* Choose very low k_n and k_t and choose $\mu = 0$ for the stiffness matrix in equation (6.8) in order to allow free relative movement of the connected nodes.

- (ii) *Sticking contact (only for frictional contact)*: Choose very high k_n and k_t and $\mu = 0$ in order to restrain the relative movement of the nodes in normal and tangential direction.
- (iii) *Sliding contact*: Choose very high k_n and very low k_t and use the coefficient of sliding friction $\mu = \mu_{sl}$ as described in method 2 of section 6.1.2 (use $\mu = 0$ for frictionless contact).

In most practical contact problems the gap elements do not stay in one of these three conditions and changes in the gap condition (gap status) throughout the application of the load are very common. For elastoplastic analysis the following incremental iterative procedure is suggested in order to account for such changes (Notice that the programming aspects of this procedure are summarised in section 7.2.2).

6.3.1 Incrementation

As for any non-linear problem the total load needs to be divided into increments. There are two kinds of increments used:

- (i) *Main increments*: As already described in section 5.3 for the plasticity procedure, the load and prescribed displacements, which are defined by the $\mathbf{Y}$ -term in equation (4.21b), can be divided into p main increments such that:

$$\mathbf{Y}^{(j)} = \mathbf{Q}^{(j)} \mathbf{Y} \quad (5.50)$$

with

$$\sum_{j=1}^p \mathbf{Q}^{(j)} = \mathbf{1} \quad (5.51)$$

- (ii) *Subincrements*: Subincrements are used to divide each main increment into smaller increments. The subincrements are adjusted automatically so that no gap elements change their gap status during the application of a successful increment as described in section 6.3.2. For the i -th subincrement of main increment number j :

$$(\mathbf{Y})_i^{(j)} = \alpha_i \mathbf{Y}^{(j)} \quad (6.23)$$

and notice also that:

$$\sum_{i=1}^q \alpha_i = 1 \quad (6.24)$$

where q is the total number of successfully applied subincrements within the main increment j .

6.3.2 Algorithm for one main increment

Step 0: Define the initial conditions

Whenever plastic deformation is expected then the zones which are expected to yield are to be modelled with finite elements and the elastic remainder of the structure with boundary element subregions.

Initially the gap status of all gap elements are to be defined. However, it is often not possible to evaluate whether a closed gap element is initially sticking or sliding and therefore an initial guess may have to be undertaken. Then a very small trial increment can be applied in order to check if the initial assumption was valid or needs to be corrected.

As already mentioned α_i represents the fraction of the load vector $\mathbf{Y}^{(j)}$ of the current main increment j . The following algorithm is based on an iterative procedure in which the subincrements α_i are applied and may not be successful because the gap status of a gap element changes within the subincrement, so that a reiteration with a smaller subincrement α_i may have to be undertaken without updating of the results. For the first subincrement iteration of a main increment start with:

$$\begin{aligned} \alpha_1 &= 1 \\ \alpha_T &= 0 \end{aligned}$$

Where α_T is the summation of all previous α_i for the current main increment.

Furthermore notice that all vectors denoted with subscript 0 contain total values before the application of the subincrement, such as F_0 , δ_0 , R_0 , T_0 , ϵ_0 and σ_0 .

Step 1: Evaluate gap element matrices

At the beginning of each subincrement gap element matrices, as defined by equation (6.8) have to be evaluated for each gap element according to its current gap status.

Step 2: Assemble gap element matrices into the global matrix

The gap element matrices are to be assembled into a subregional matrix and then into the structural matrices as shown in section 6.2.

Step 3: Solve the resulting system of equations

Boundary conditions can be applied so that the global system of equations can be reduced to a solvable system of equations and force and displacement results can be obtained.

Step 4: Plasticity corrections

Calculate the stress of the finite elements and if the yield stress at any of the Gaussian points of the finite elements is exceeded then an iterative plasticity correction procedure becomes necessary for the current subincrement as described in chapter 5. Therefore results which correspond to the plastic deformation can be obtained.

Step 5: Check the gap elements for changes in their gap status

The obtained results for the current subincrement α_i are denoted with subscript i such as $(\delta)_i$, $(T)_i$ and $(R)_i$.

It also becomes necessary to evaluate the following values for every gap element:

$(d_n)_i =$ Normal distance (gap) between the two nodes of the gap element after the application of the subincrement α_i .

$(d_n)_0 =$ Normal distance (gap) between the two nodes of the gap element before the application of the subincrement α_i .

$(F_n)_i^{(total)} =$ Total normal reaction force at node I of the gap element after the application of the subincrement α_i .

$(F_n)_0 =$ Total normal reaction force at node I of the gap element before the application of the subincrement α_i .

$(F_t)_i^{(total)} =$ Total tangential reaction force at node I of the gap element after the application of the subincrement α_i .

$(F_t)_0 =$ Total tangential reaction force at node I of the gap element before the application of the subincrement α_i .

For each possible change of the gap status of each of the gap elements a parameter $(\alpha_i)^{new}$ can be found such that the gap status would change at the end of the subincrement if $(\alpha_i)^{new}$ was applied instead of α_i . The following parameters $(\alpha_i)^{new}$ can be derived by using a linear relationship between the results of the already applied α_i and the results which would change the gap element status:

- (1) Change from separation to contact

$$(\alpha_i)_1^{new} = \left(\frac{(d_n)_0}{(d_n)_0 - (d_n)_i} \right) \alpha_i \quad (6.25)$$

- (2) Change from contact to separation

$$(\alpha_i)_2^{new} = \left(\frac{(F_n)_0}{(F_n)_0 - (F_n)_i^{(total)}} \right) \alpha_i \quad (6.26)$$

- (3) Change from sticking contact to sliding contact

$$(\alpha_i)_3^{new} = \left(\frac{\mu_{st} - \mu_0}{\mu_i - \mu_0} \right) \alpha_i \quad (6.27)$$

where $\mu_0 = \left| \frac{(F_t)_0}{(F_n)_0} \right|$, $\mu_i = \left| \frac{(F_t)_i^{(total)}}{(F_n)_i^{(total)}} \right|$ and μ_{st} is the coefficient of sticking friction.

- (4) Change from sliding contact to sticking contact

For this case it is not possible to define a parameter $(\alpha_i)^{new}$ which delivers converging results. However it is clear that a gap element changes from sliding contact to sticking contact if:

$$\frac{(\Delta u_t)_{i-1}}{(\Delta u_t)_i} < 1 \quad (6.28)$$

where: $(\Delta u_t)_i = ((u_t)_I)_i - ((u_t)_J)_i \quad (6.29)$

and $(\Delta u_t)_{i-1} = ((u_t)_I)_{i-1} - ((u_t)_J)_{i-1} \quad (6.30)$

It can also be seen that equation (6.28) expresses that if there is a change in the direction of the relative tangential displacement of a gap element from the last subincrement $i-1$ to the current subincrement i then the gap element must go through a sticking condition.

Therefore whenever equation (6.28) is satisfied for any gap element, change its gap status from sliding to sticking contact and reiterate with α_i from step 2 without updating the results.

For each of the gap elements the above $(\alpha_i)^{new}$ parameters need to be evaluated. It is then clear that the gap element with the smallest positive $(\alpha_i)^{new}$ would change its gap status first. This smallest positive $(\alpha_i)^{new}$ of all possible changes of all gap elements may be denoted $(\alpha_i)_{min}^{new}$ and the gap element at which it occurs may be defined as gap element S. Then the following three possibilities arise:

- (i) No changes in the gap element status of gap elements due to application of α_i .

This case occurs if $(\alpha_i)_{min}^{new} > \alpha_i$. This means that the α_i - iteration was successful and the results can be updated. Therefore:

* Go to step 6

- (ii) The gap element status of gap element S changes exactly at the end of the subincrement α_i .

This case occurs if $(\alpha_i)_{min}^{new} \approx \alpha_i$. This also means that the α_i - iteration was successful and results can be updated. Therefore:

* Change the gap status of gap element S.

* Go to step 6

- (iii) The gap condition changes during the application of subincrement α_i

This case occurs if $0 < (\alpha_i)_{\min}^{new} < \alpha_i$

This means that the applied α_i was too large because the gap status of gap element S would have changed before the end of the current subincrement α_i and the iteration has to be repeated with $(\alpha_i)_{\min}^{new}$ instead of α_i . Therefore:

- * Reiterate from step 1 with $\alpha_i \equiv (\alpha_i)_{\min}^{new}$.

Step 6: Evaluate stress and strain results and update all results

In this step the remaining unknown stress and strain results of the BEM subregion are to be evaluated. Then all total results can be updated according to the successfully applied α_i . Hence:

$$(\alpha_T)^{new} = \alpha_T + \alpha_i \quad (6.31)$$

$$(\delta_0)^{new} = \delta_0 + (\delta)_i \quad (6.32)$$

and similarly to equation (6.32) the results $(T)_0$, $(R)_0$, $(\sigma)_0$ and $(\epsilon)_0$ of the FEM and BEM subregions can be updated for the next subincrement to be applied.

Step 7: Check for the end of the main increment

There are two possibilities:

- (i) $(\alpha_T)^{new} = 1$

- * End of the main increment
* Apply next main increment and start from Step 0.

$$(ii) \quad (\alpha_T)^{new} < 1$$

* Apply next subincrement:

$$\alpha_{i+1} = 1 - (\alpha_T)^{new} \quad (6.33)$$

* Go to Step 1

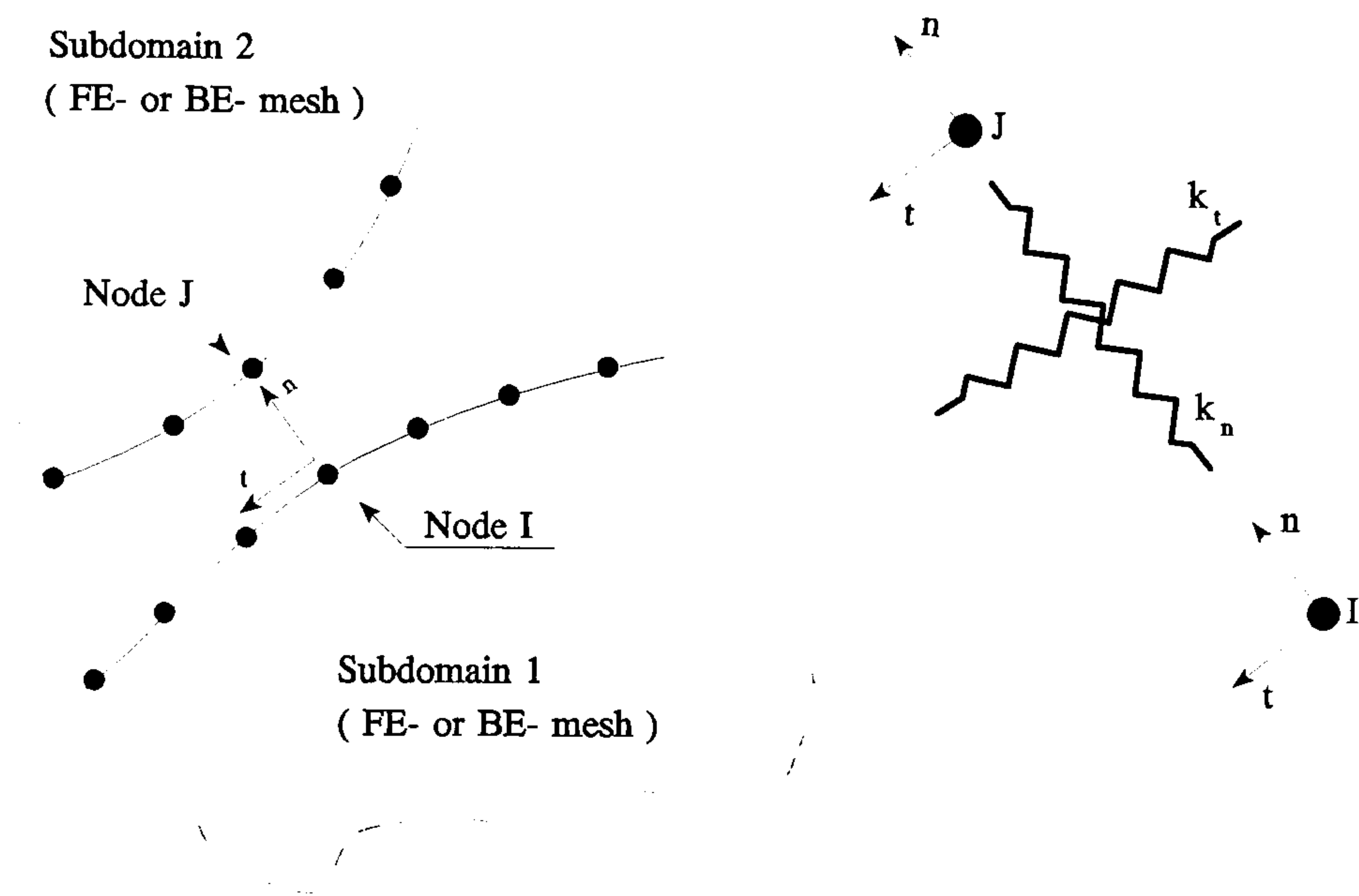


Figure 6.1 A node-to-node gap element between two nodes of two FEM- or BEM-subregions.

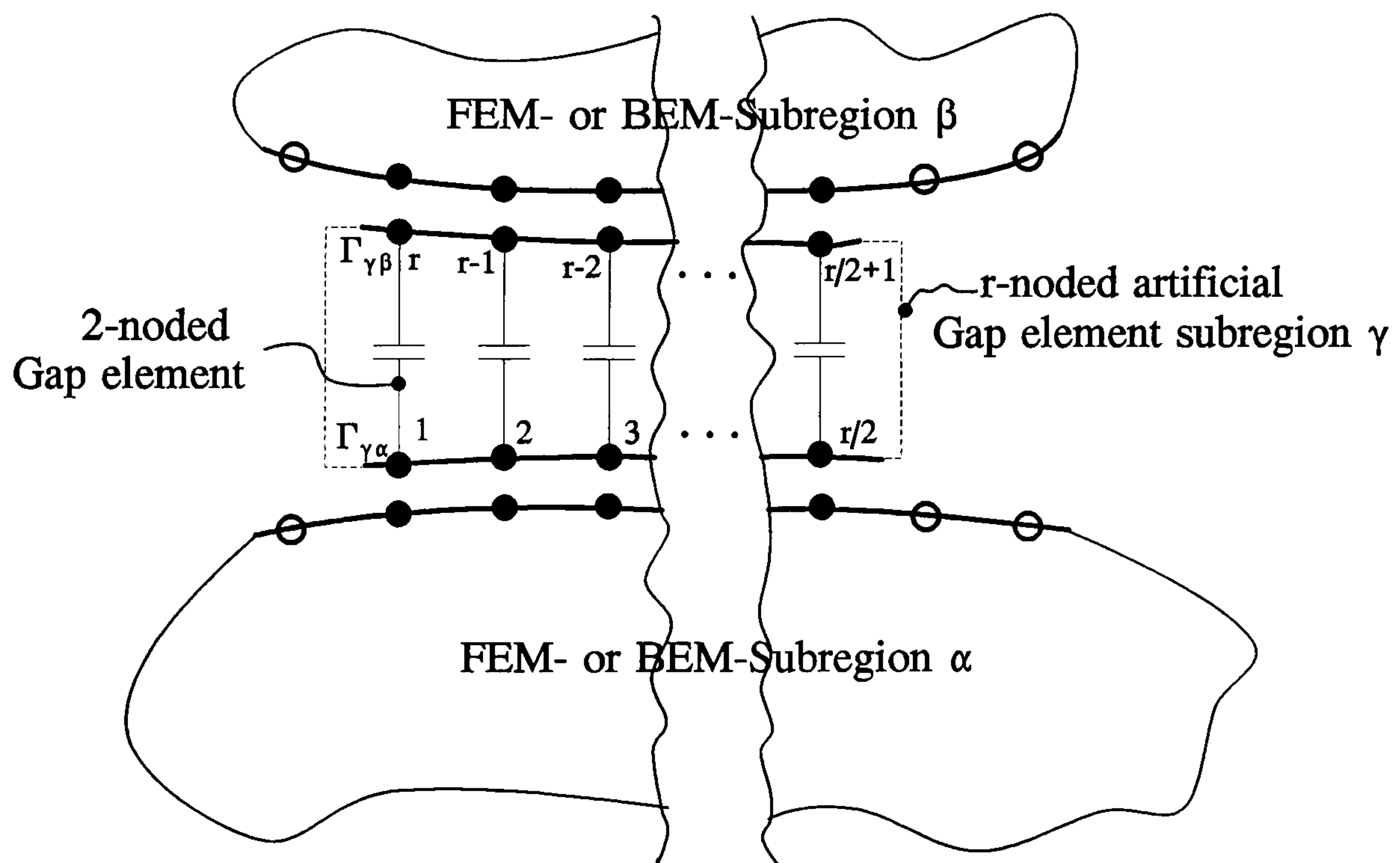


Figure 6.2 An artificial gap element subregion between 2 FEM- or BEM-subregions.

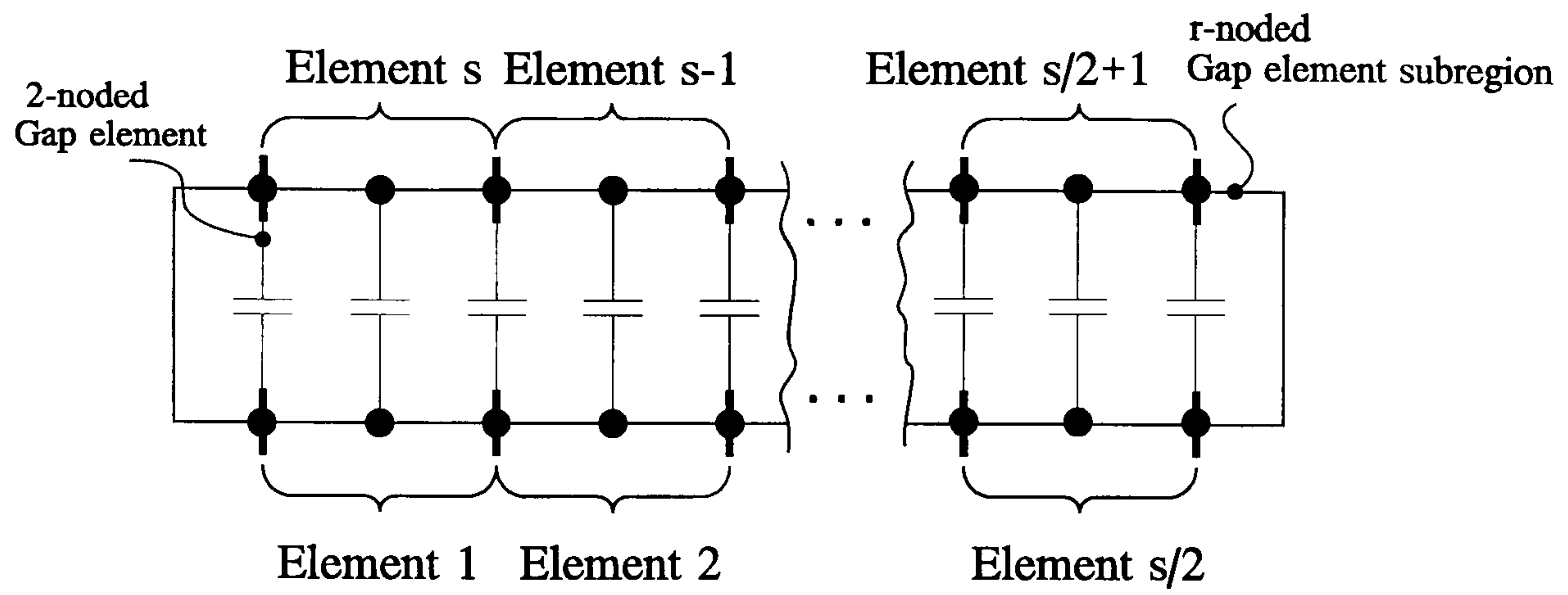


Figure 6.3 The discretisation of the artificial gap element subregion into s artificial boundary elements.

CHAPTER 7

PROGRAMMING

7.1 MODULAR DIVISION

A coupled FEM/BEM package capable of modelling contact and/or plasticity was created according to the theory and algorithms presented in chapters 4-6. The package was coded in FORTRAN 77 and based on existing FEM and BEM packages. These FEM and BEM packages were large programs already and many subroutines needed to be added so that the coupling-, contact- and plasticity-procedures could be included. Hence, in order to create a well structured package, it was divided into 8 modules. A summary of the main programming aspects of these modules is given in the following sections.

7.1.1 Module MAIN

This module includes the essential structure of the package as described in section 7.2 and needed to be newly created. It connects the other 7 modules to one another in a logical manner and also controls the incrementation and the iterative loops. The following subroutines were created:

- * **PROGRAM COUDIR** - This is the central main routine of the package which controls the subroutines and steers the incremental and iterative procedures and is described in more detail in section 7.2.
- * **SAVE OLD** - In this subroutine all total results and the results obtained by the last increment are saved before the application of the next increment.
- * **RESET OLD** - The total and incremental results obtained after an unsuccessful iteration are reset to the results before the application of the unsuccessful

increment whose were saved by SAVE OLD.

- * **COORDINATE UPDATE** - This subroutine updates the nodal coordinates according to the total displacements of previous increments and hence structural matrices are then evaluated with respect to the updated coordinates. This subroutine is optional and can be defined to be called by the user for cases with large displacements.

7.1.2 Module FEM

This module contains all FEM related subroutines, such as element stiffness matrix generation and assembly of the elemental matrices into the subregional FEM stiffness matrix as well as application of the independent boundary conditions. Moreover stress and strain evaluation at Gaussian as well as geometrical nodes.

Most of the subroutines of the given FEM package could be used without making great changes. This was possible by using node numbers from 1 to n in the n -noded FEM subregion. Once structural results are obtained, the FEM results can be condensed and printed. Notice that results for the interface reaction forces are not obtained explicitly with the chosen coupling method and they need to be calculated afterwards by using equation (4.19).

7.1.3 Module BEM

This module includes all BEM related subroutines, such as boundary element matrices and their assembly into subregional BEM matrices, as well as the application of independent boundary conditions, a stress and strain calculation at internal and boundary points.

It was possible to use many subroutines of the given BEM subregion package. Changes

needed to be made to the output routine BEMOUTPUT. Also for the case of an internal FEM/BEM interface corner only one corner point is used as a source point and the required additional equations are obtained by using an external point instead of the second corner node as additional source point as described in section 4.5.3.

7.1.4 Module DATA

The module DATA contains the data input routines of FEM, BEM, contact and material data. Two new subroutines BEMDATA and FEMDATA were created. An example of a data file with explanations can be found in Appendix G.

In BEMDATA all data associated with the BEM- and gap element-subregions are read. These include nodal coordinates, boundary conditions, boundary element definitions and material properties of the BEM subregion and gap element data and data associated with the contact iterations.

In FEMDATA the data associated with the FEM subregion and the coupling of FEM-, BEM- and/or gap element-subregions are read. They include nodal coordinates, boundary conditions and finite element definitions and the elastoplastic material data of the finite element subregion. It is also necessary to discretise the FEM/BEM interface as described in section 4.4 (or in section 6.2 for the case of an interface with a gap element subregion).

7.1.5 Module ELEMENTS

Consisting of elemental shape functions of finite elements and boundary elements which were extracted from previous FEM and BEM packages.

7.1.6 Module SUBOP

This module mainly consists of new subroutines which are involved in subregional operations. It is concerned with the assembly of the subregional equations into structural equations and then the solution of the structural set of equations. A brief description of the most important new routines shall be given here:

- * **ASSEMBLER** - For any combination of FEM-, BEM- and gap element subregions, the subregional equations of each of the defined subregions are assembled into one structural set of equations. Then the interface boundary conditions are applied in order to reduce the matrix to a solvable set of equations as presented in sections 3.3.2 and 4.3 and 6.2.
- * **ISOLVER** - Inverse matrix solver for the solution of the structural set of equations. An inverse matrix solver was chosen because the inverse of the matrix can be used for plasticity iterations so that the set of equations does not have to be solved for each iteration.
- * **SOUTPUT** - The results of each of the defined subregions are extracted from the structural results.

7.1.7 Module GAP

This is a new module which contains all subroutines needed for the evaluation of the subregional gap element equations. Moreover this module includes all contact algorithm subroutines which control the current contact status of the gap elements. The following new subroutines were created within this module:

- * **GAP1D** - Evaluation of the stiffness matrix of a node-to-node gap element as in equation (6.8) according to the current gap status of the gap element.

- * GAP STIFFNESS - Rotation of the elemental stiffness matrix obtained from GAP1D as in equation (6.14) and assembly into a subregional stiffness matrix of the gap element subregion according to a topology array as in equation (6.15).
- * CELMATRIX - Elemental transformation matrix as in equation (4.22).
- * CTMATRIX - Assembly of the elemental transformation matrices of each of the artificial boundary elements on the gap element subregion into subregional equations as in equation (6.16).
- * GAP - Renames the subregional stiffness and transformation matrices such that equation (6.17) is valid.
- * GAPCONDITION - Checks for changes in the gap status of any of the defined gap elements and evaluates all necessary contact iteration parameters as described in section 6.3.
- * SLIDECHECK - This subroutine controls whether the frictional forces of the gap elements were in the right direction as described in method 2 of section 6.1.2.
- * SLIDE STICK CHANGE CHECK - Checks whether there was a change from sliding contact to sticking contact.

7.1.8 Module INSTRESS and module INSTRAIN

These modules contain iterative plasticity corrections with the initial stress method in INSTRESS and the initial strain method in INSTRAIN. Notice that either INSTRESS or INSTRAIN can be implemented but not both at the same time. The extracted subregional results of the FEM subregion are used in order to obtain uncorrected stress/strain results. These uncorrected stress/strain results lead to a residual force vector which needs to be applied in the next iteration in terms of $\mathbf{Y}$ as described in detail in section 5.3.

7.2 PROGRAM STRUCTURE

7.2.1 Main structure

It was already mentioned that the main routine PROGRAM COUDIR of the module MAIN connects the other 7 modules in a logical manner, which may be schematically represented as shown in figure 7.1.

Figure 7.1 shows that for each main increment the subregional FEM-, BEM- and gap element-equations are to be evaluated by the modules FEM, BEM and GAP. Then the subregional equations are assembled into the A -matrix in SUBOP. It is possible to assemble and solve for any combination of FEM-, BEM- and gap element-subregions, which is also explained in Appendix F. Then the inverse of the A -matrix is calculated in SUBOP and a solution is also obtained in SUBOP.

It can be seen that there are three different incremental loops, which are the plasticity loop, the contact loop and the main increment loop. Within each contact iteration the plasticity procedure of INSTRESS or INSTRAIN has to be run until accurate results are obtained. Since a matrix inversion solver was used, it was possible to use the inverse of the structural A -matrix for each of the plasticity iterations and hence the plasticity loop rejoins at the solution step of SUBOP as shown in figure 7.1. The use of the inverse A -matrix for the contact loop and the main increment loop is not possible since the gap element stiffnesses might change within these loops and then the structural matrix needs to be reassembled. The computational aspects of the iterative contact loop are described in more detail in section 7.2.2.

The program structure in terms of the most important subroutines may be illustrated as shown in figure 7.2. The program is initiated by reading the data in BEMDATA for the BEM- and contact-data and in FEMDATA for the FEM-data.

In the subroutines ASSEMBLER, ISOLVER and SOUTPUT of the module SUBOP, the evaluation and coupling of the subregional equations is undertaken and the set of

structural equations is solved. In the subroutine ASSEMBLER the subregional equations of each of the defined subregions is evaluated by the subroutines shown in figure 7.2. Then a solvable structural set of equations is obtained by applying the interface boundary conditions which is then solved by ISOLVER. The subregional results are extracted in subroutine SOUTPUT.

Uncorrected stresses and strains are evaluated in the subroutines EXTERNAL, INTERNAL, STRESS and STRESS GAUSS.

If plasticity is involved the subroutines ASSFOR and CORRGeo evaluate the necessary stress corrections according to initial stress or initial strain procedures. The residual force is obtained in ASSFOR and then the accuracy of the results of the current iteration are checked in CONVERGENCE. If the results are not sufficiently accurate another plasticity iteration becomes necessary and the plasticity loop has to be run again with the new residual force vector by using the already evaluated inverse matrix of ISOLVER.

Then gap element checks are undertaken in the subroutines SLIDECHECK, SLIDE STICK CHANGE CHECK and GAPCONDITION as described in more detail in section 7.2.2.

Finally the results are stored and printed in the subroutines OUTPUT for BEM and contact results and FEMOUTPUT for the FEM results.

7.2.2 Contact iteration loop

Figure 7.3 shows the most important subroutines of the contact iteration loop and their logical connection. The logical structure follows the algorithm presented in section 6.3.

For each subincrement the total results need to be saved in SAVE OLD before the new incremental results are established. This is necessary because the total results are

continuously updated during the plasticity iterations. For the case of an unsuccessful subincrement, a reiteration without updating of the results is necessary which is possible by restoring the saved results in RESET OLD as indicated in figure 7.3.

Subroutine SLIDECHECK checks whether the directions of the frictional forces of the gap element were assumed accurately and if not a reiteration has to be run.

In subroutine SLIDE STICK CHECK CHANGE each of the gap elements is checked for a change from sliding condition to sticking condition. If there is a change of this condition of any of the gap elements a reiteration has to be run with updated gap status of the concerned gap element.

In GAPCONDITION the α -parameters of the gap elements are established in GET ALFA1, GET ALFA2 and GET ALFA3 and $\alpha_{\min}$ is to be found. Then a decision is made whether to reiterate or to apply the next subincrement as shown in figure 7.3.

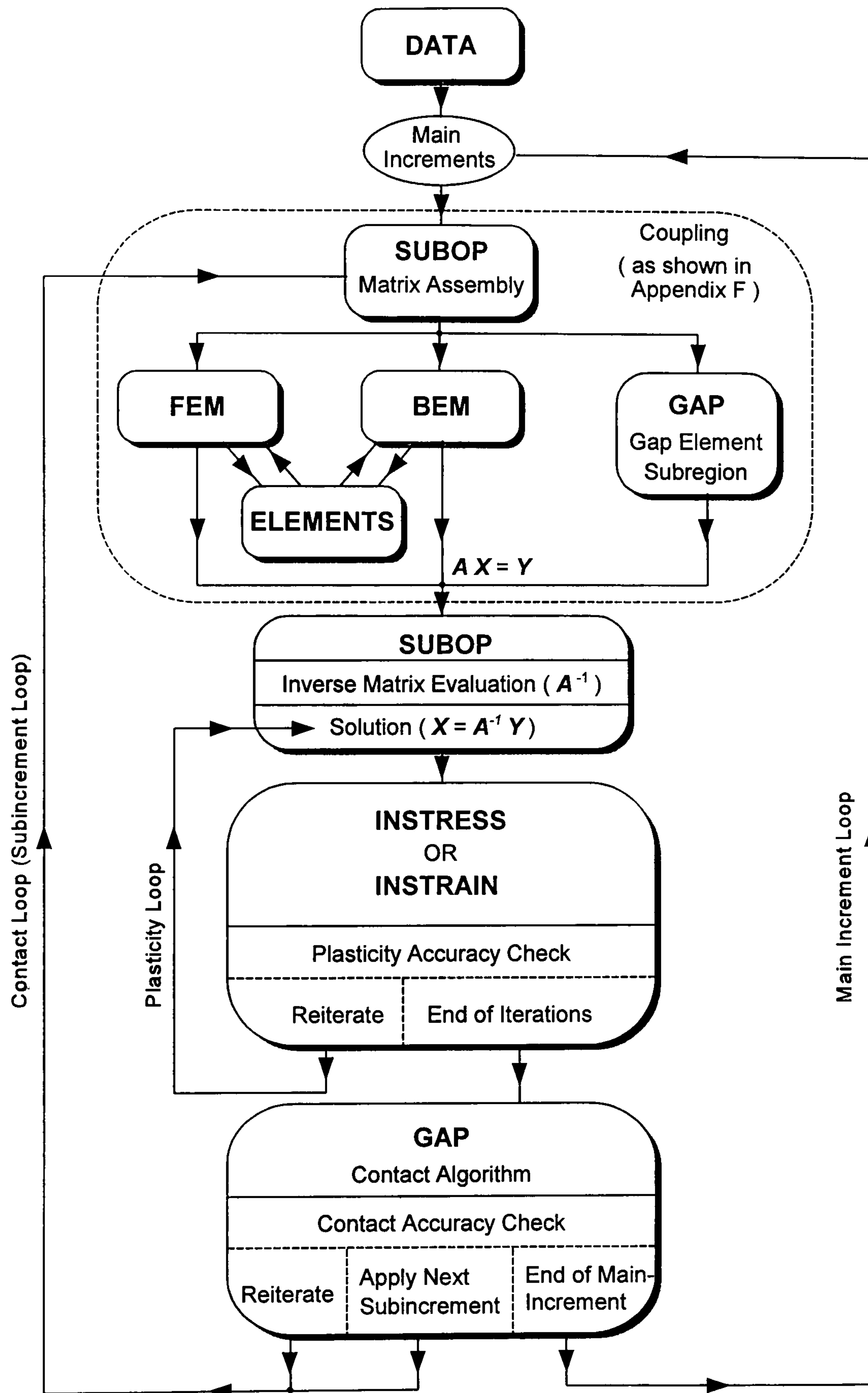


Figure 7.1 Interrelation between the modules which is controlled by module MAIN.

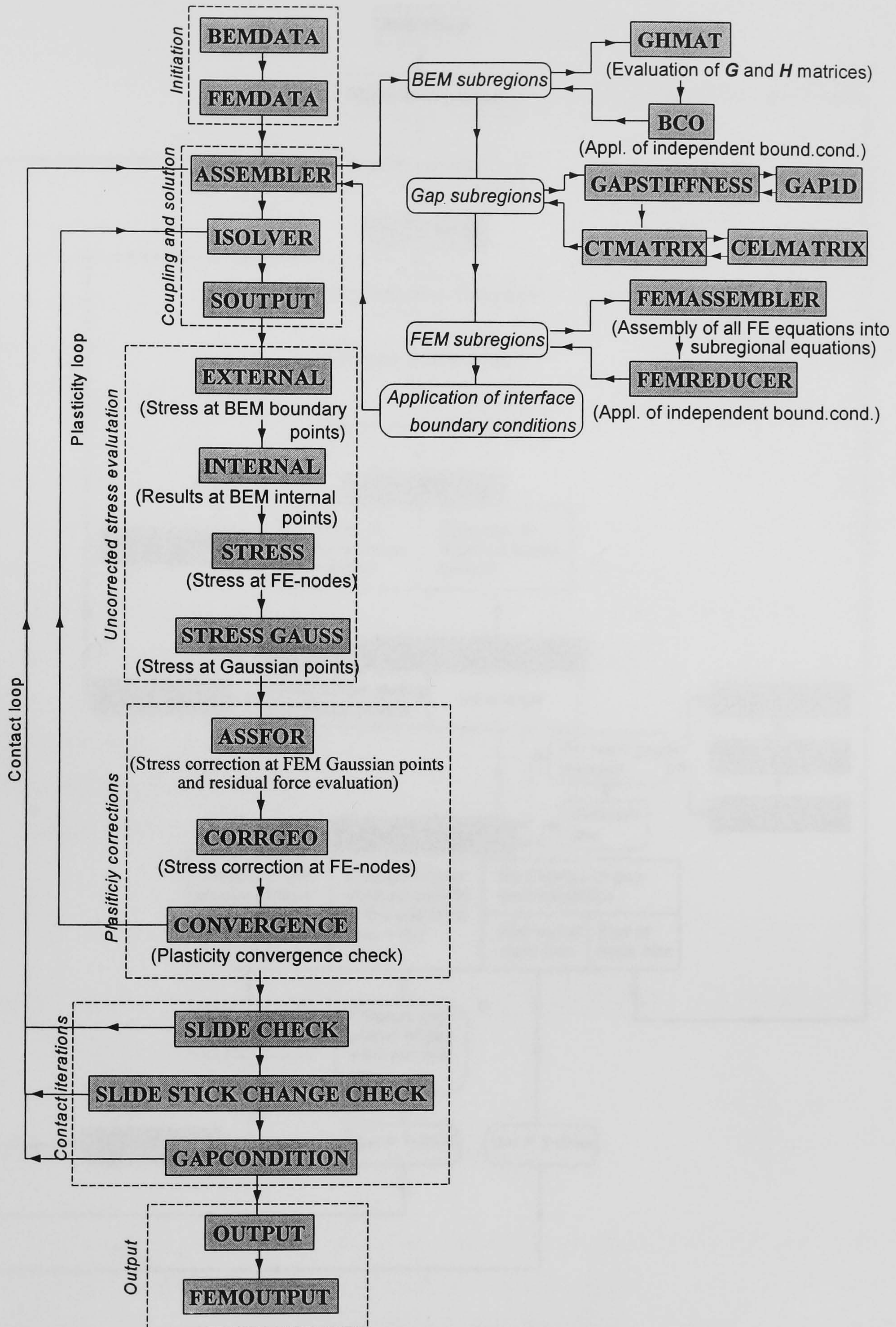


Figure 7.2 Interrelation between the most important subroutines.

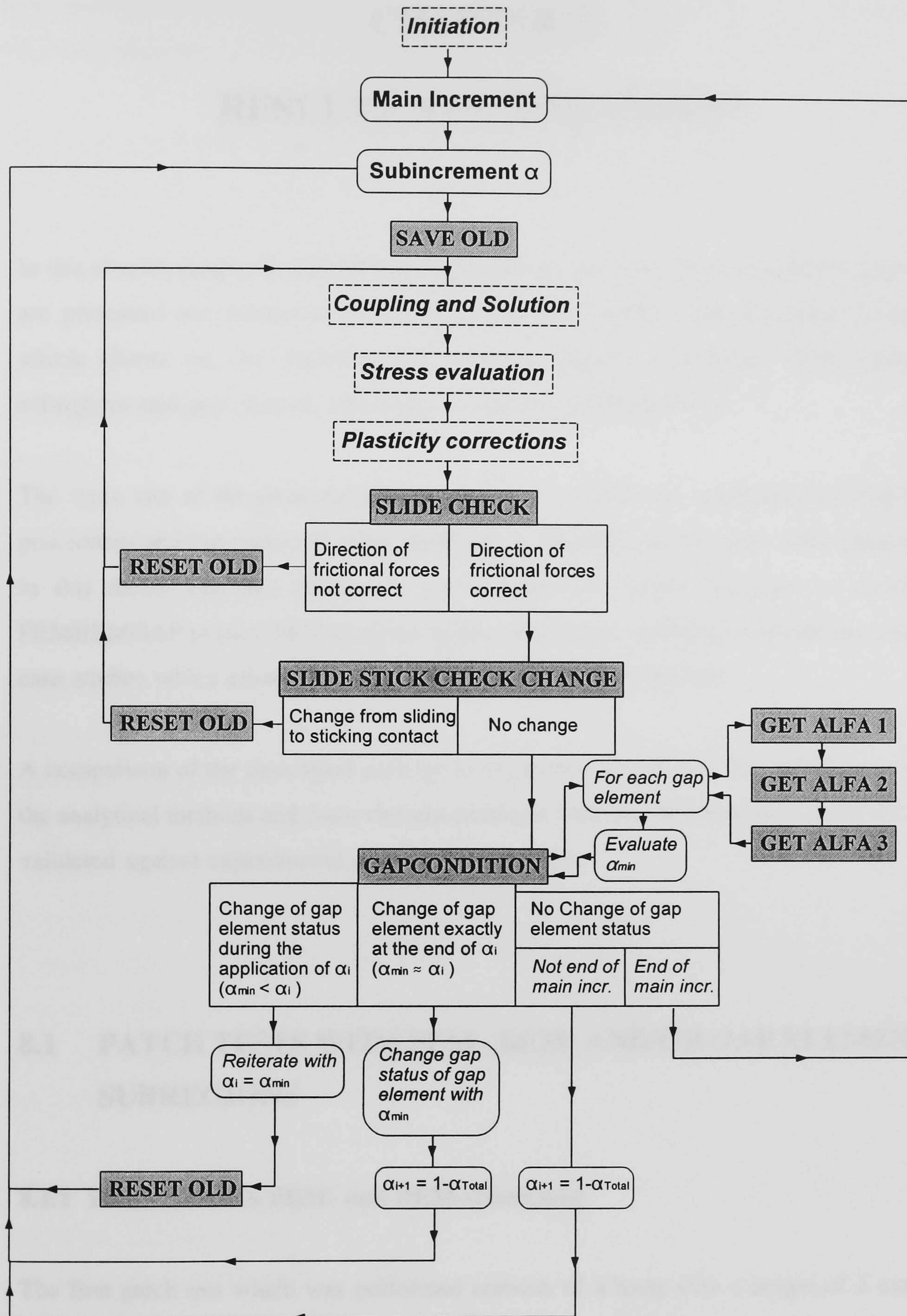


Figure 7.3 Iterative contact procedure.

CHAPTER 8

RESULTS AND DISCUSSION

In this chapter results for various case studies which were run on the developed package are presented and compared to analytical and FEM results. The developed package which allows for the definition of boundary element subregions, finite element subregions and gap element subregions is denoted FEMBEMGAP.

The main aim of the presented case studies is to validate the (elastoplastic) coupling procedures and the contact/friction algorithm of FEMBEMGAP which were presented in this thesis. For this purpose it was regarded as mostly important to compare FEMBEMGAP to pure FEM analyses and/or established analytical solutions for simple case studies which allow the validation of parts of FEMBEMGAP.

A comparison of the developed package to experimental data was not undertaken since the analytical methods and finite element packages used for the comparison were already validated against experimental data by other authors.

8.1 PATCH TESTS WITH FEM-, BEM- AND/OR GAP ELEMENT-SUBREGIONS

8.1.1 Patch test with FEM- and BEM-subregions

The first patch test which was performed consists of a body with a height of 2 mm, a length of 4 mm and a thickness of 1mm. It is loaded by a line load $T = 100$ N/mm along the right edge as shown in figure 8.1. The body with a Young's modulus $E=10^6$ N/mm² and a Poisson's ratio $\nu=0.3$ is subdivided into an FEM-subregion which consists

of 1 quadrilateral finite element and a BEM-subregion which consists of 4 parabolic boundary elements as shown in figure 8.1.

The displacement and stress results can be seen for the points 1 to 15 of figure 8.1 in comparison to the expected theoretical results in tables 8.1 and 8.2 respectively. There are no observed deviations apart from at the internal points number 14 and 15 (shown in figure 8.1). However these deviations are very small and are caused by deviations in the internal stress evaluation rather than by the coupling procedure. Therefore it can be concluded that the coupling procedure and the transformation of nodal forces into tractions at the interface nodes of the FEM-subregion are accurate and do not introduce any significant errors.

8.1.2 Patch test with FEM-, BEM- and gap element-subregions

In this case study the same model and material and loading data as in the previous case study were used and an artificial gap element subregion which consists of 3 gap finite elements was inserted between the FEM-subregion and the BEM-subregion (as shown in figure 8.2).

Very high normal and tangential stiffnesses of 10^{13} N/mm were assigned to the gap elements, which is a number significantly higher than the Young's modulus of the material $E=10^6$ N/mm². These high stiffnesses restrain the relative movement of the FEM- and BEM-subregion and the model is expected to deform as a solid body like in the previous case study.

The results obtained were exactly identical to the results of the case without gap element subregion and can therefore also be seen in tables 8.1 and 8.2. Hence it can be concluded that a gap element subregion can be inserted without noticeable loss of accuracy. This shows that the coupling procedure and the associated transformation of nodal forces to nodal tractions of the FEM- and gap element-subregions (as shown in figure F.1 of Appendix F) deliver very accurate results.

8.1.3 Patch test with FEM- and BEM-subregions and an interface corner

A square plate with a Young's modulus of 10^6 N/mm² and a Poisson's ratio of 0.3 is loaded by a line load $T = 100$ N/mm along its right edge whilst the left edge and the lower edge are restrained as shown in figure 8.3. The length of the sides of the square plate is 8 mm and its thickness is 1 mm. The domain is subdivided into a FEM-subregion with 65 nodes and 16 8-noded finite elements and a boundary element subregion with 54 nodes and 24 3-noded boundary elements as shown in figure 8.3.

There is an internal interface corner at point 7 of figure 8.3 which was modelled by using an external source point next to the corner location as described in section 4.5.3. There are external interface corners at points 3 and 11 which were modelled as suggested in case (a) of section 4.5.2.

A comparison of theoretical results and the results obtained by FEMBEMGAP for 19 points (as shown in figure 8.3) is displayed in tables 8.3 and 8.4. It can be seen that the errors of the results are all in an acceptable range with displacements under 0.4% error and stresses under 2.6% error. The displacements of the points 5-10 which are located near the internal interface corner deviate 0.005% - 0.31% whilst the displacements at the other points including the internal points of the BEM subregion have very accurate values. Similarly the error in the stresses is clearly larger near the internal interface corner with values between 0.05% and 2.6%. The results at the external interface corners at points 3 and 11 indicate very good accuracy.

Therefore it can be concluded that the results for this patch test are acceptably accurate and that there are slight result deviations near the internal interface corner. However the local deviations near the internal interface corner are in an acceptable range and do not affect the accuracy of the results in the other parts of the body.

8.2 ELASTOPLASTIC CASE STUDIES WITH FEM- AND BEM-SUBREGIONS

8.2.1 Pressurised cylinder

The first elastoplastic case study is a partly plastic internally pressurised cylinder. The cylinder has an internal radius of 100 mm, an external radius of 200 mm and an internal pressure of 12 N/mm². The yield stress of the material is defined to be 24 N/mm² and the material behaves perfectly plastic ($H'=0$) with a Young's modulus of 10⁵ N/mm² and a Poisson's ratio of $\nu=0.3$.

The FEMBEMGAP model is shown in figure 8.4, where the FEM-part models the region between the radii of 100 mm and 140 mm since yielding is expected in this area (and it can be analytically proved that the elastoplastic radius is at 120 mm). The FEM subregion consists of 117 nodes and 30 8-noded quadrilateral finite elements. The elastic remainder of the structure is modelled with a boundary element subregion with 46 nodes and 21 3-noded boundary elements.

The FEMBEMGAP package has been compared to a validated elastoplastic FEM-package of Guzelbey (1992) and to an analytical solution (Johnson and Mellor (1983)) for a plane strain case. The FEM mesh consists of 181 nodes and 50 elements as shown in figure 8.5. Notice that for FEMBEMGAP as well as for the FEM package an initial stress procedure was used in order to avoid convergence problems, which might occur with initial strain procedures for very small H' .

The radial displacement results can be seen in figure 8.6 and the obtained radial stress and hoop stress results are illustrated in figures 8.7 and 8.8 respectively. The radial displacement results of FEMBEMGAP deviate less than 0.3% from the analytical solution (where the analytical solution was only evaluated in the elastic section). The radial stress results deviate with less than 5% and the hoop stress results deviate with less than 2% error from the analytical results. These deviations are in an acceptable range. The pure FEM package delivers almost the same deviations throughout, although

the radial stress results of FEMBEMGAP in the elastic section are more accurate than the FEM results. This improved accuracy of the stresses in the elastic region can be explained with the fact that there are more nodes defined along the restrained edge of the FEM/BEM cylinder model where the results are evaluated and an additional improvement factor is that the stress evaluation within the BEM does not have the drawback of using additional approximations as within the FEM.

It is clear that FEMBEMGAP compares well to the analytical solution and can even supply slightly better accuracy than the pure FEM package, even though altogether less modelling effort was necessary in comparison to the pure FEM. Hence it can be concluded that the coupling procedure does not introduce any recognisable additional errors compared to the pure FEM package. This case study can also be regarded as a validation of the plasticity procedure for coupled FEM/BEM of section 5.3. since the plasticity iterations converged very well and the results in the plastic zones are very accurate.

8.2.2 Rotating circular disc

The second case study is a rotating circular disc with a rotational speed of $\omega = 335$ rad/s and a material density of $7.8 \cdot 10^6$ kg/m³. Material and geometry data are identical to the pressurised cylinder data except that for this case plane stress conditions were assumed.

The meshes of the previous case study were used since yielding did not exceed the FEM-region which was specified for the coupled FEM/BEM analysis and it is clear that the modelling effort with coupled FEM/BEM is lower than with the pure FEM.

Again a comparison between FEMBEMGAP, pure FEM and an analytical solution (Johnson and Mellor (1983)) was undertaken and the results can be seen in figures 8.9 - 8.11. The deviations of FEMBEMGAP from the analytical solution are smaller than 0.25% for the displacements, while radial and hoop stress deviate with less than 3% and 1% respectively. Similarly to the pressurised cylinder case study, the pure FEM shows

similar deviations to FEMBEMGAP and again FEMBEMGAP delivers slightly more accurate radial stress results for the same reasons as in the previous case study.

It can be concluded that for this case study the coupled FEM/BEM delivers very good results with similar accuracy to the pure FEM and that this case study is another proof of the validity of the elastoplastic coupling of FEM and BEM suggested in this work.

8.2.3 Square plate with a central hole under biaxial tension

A large square plate with a small central hole is loaded by a constant distributed load of 30 N/mm^2 along its edges as shown in figure 8.12. The side length of the square plate is 2000 mm and the thickness is 1mm. The hole has a radius of 100 mm. The yield stress of the material is 50 N/mm^2 with $H'=0$ and the Young's modulus is 10^5 N/mm^2 and Poisson's ratio is $\nu=0.3$.

An analytical solution for an infinite plate with the same loading conditions has been obtained (Savin (1961)) and was compared to an initial stress analysis with FEMBEMGAP and pure FEM. Because of the symmetry conditions of this case study it was possible to model only a quarter of the plate. The meshes used for coupled FEM/BEM and for pure FEM are shown in figures 8.13 and 8.14 respectively. Within FEMBEMGAP the plastic area around the hole was modelled with 117 nodes and 30 8-noded quadrilateral finite elements, whilst the remaining elastic sections were modelled with 85 nodes and 40 3-node isoparametric boundary elements. In contrast the much more costly pure FEM model consists of 437 nodes and 130 8-node finite elements.

The results of the region near the hole (up to a radial distance of 500 mm from the centre of the plate) are illustrated in figures 8.15-8.17. Although there was no analytical solution for the radial displacements found in the literature, it can be seen that FEMBEMGAP and FEM deliver almost identical results, which indicates an accurate solution. The results for the radial stress of FEMBEMGAP deviate by less than 2% from

the analytical solution and the results for the hoop stress by less than 1%. The pure FEM shows very similar deviations from the analytical solution.

Yet again the accuracy of the results of FEMBEMGAP validates the suggested elastoplastic coupling procedure. It can also be concluded that especially for this case study there is an enormous saving in modelling effort with coupled FEM/BEM compared to the pure FEM analysis and at the same time the accuracy is equally good. This proves that the suggested elastoplastic coupling of the FEM and the BEM is especially beneficial for problems with small plastic stress concentration zones.

8.3 ELASTIC CONTACT CASE STUDIES WITH BEM- AND GAP ELEMENT-SUBREGIONS

8.3.1 Frictional punch on a large rigid rectangular body

In this case study a rectangular body with a Young's Modulus of 1 N/mm² is pressed onto a comparably rigid surface body with a Young's modulus of 1000 N/mm² with both bodies having a Poisson's ratio of $\nu = 0.3$ as shown in figure 8.18. The thickness of both bodies is 1mm and the loading q on the upper side of the punching body is 0.1 N/mm². A plane strain analysis was performed on FEMBEMGAP and compared to ABAQUS since there are no analytical solutions available for this class of problem.

Because of the given symmetry of this problem it was possible to model only half of the bodies by restraining the line of symmetry in horizontal direction. The FEMBEMGAP model is a pure BE-mesh with 50 3-noded boundary elements as shown in figure 8.19 and with a gap element subregion defined across the contact interface. The ABAQUS FE-model can be seen in figure 8.20. It consists of 360 isoparametric 8-noded finite elements, and INTER3 contact elements are defined on the contact interface. It is clear that the modelling effort here is significantly larger than for the BE-mesh. Especially the refined mesh around the edge of the punch complicates the FE-model and leads to a large number of elements.

It can be seen that the elastic punching body gets compressed and the rigid surface body is not expected to deform greatly because of its rigidity. The punching body naturally deforms in the direction of the loading but also in the direction along the contact area. The effect of different coefficients of friction on contact pressure and the tangential sliding along the contact area is evaluated for both ABAQUS and FEMBEMGAP. Figure 8.21 shows the relative tangential displacement (=sliding) results along the interface for 4 different coefficients of friction. There is a frictionless case ($\mu=0.0$), a case where the complete interface is sliding ($\mu=0.05$), a partly sliding case ($\mu=0.2$) and a case with a fully sticking interface ($\mu=0.4$). In all four cases FEMBEMGAP and ABAQUS deliver very similar results with deviations smaller than 2%. For the partly sliding case with $\mu=0.2$ the stick/slip zones are almost identical which shows that the iterative procedure to evaluate changes in frictional conditions of the gap elements applied within FEMBEMGAP works accurately.

In figures 8.22-8.25 the contact pressure in the contact area is illustrated for different coefficients of friction. For the frictionless case ($\mu=0.0$ in figure 8.22) a contact pressure similar to the pressure applied at the top side of the elastic body is expected since the rigidity of the surface body turns this case into a kind of patch test. ABAQUS and FEMBEMGAP show the expected pressure distribution although ABAQUS shows a small stress concentration at the edge of the punch. The contact pressure distributions of the frictional cases ($\mu=0.05$, $\mu=0.2$ and $\mu=0.4$) also show very good agreement between ABAQUS and FEMBEMGAP.

Figure 8.26 shows the stress in y-direction of the frictionless case along the line of symmetry for FEMBEMGAP and ABAQUS. Both packages agree very well.

It can be concluded that both packages have almost identical results for each of the friction coefficients defined. It was the main purpose of this case study to validate the friction modelling capacity of the developed package with different friction conditions for a simple case study. The similarity of the results indicates that the friction modelling of FEMBEMGAP (as suggested in chapter 6) works as well as the ABAQUS friction algorithm does, with FEMBEMGAP offering the additional advantage of minimising the

modelling effort by using a pure boundary element mesh.

8.3.2 Frictionless contact of a cylinder and a large rectangular body

A cylinder is pressed onto a rectangular surface body by applying a prescribed displacement of $v = -1.2$ mm along its horizontal axis of symmetry. All dimensions are shown in figure 8.27 and both bodies have a Young's modulus of $E=10000$ N/mm² and a Poisson's ratio of $\nu = 0.3$ and a thickness of 1mm. The case study was run under plane stress conditions on FEMBEMGAP and ABAQUS with analysis models as shown in figure 8.28 and figure 8.29 respectively. The contacting surfaces are considered frictionless. It is possible to model only one half of the bodies by restraining the line of symmetry in x-direction. The FEM model used for ABAQUS consists of 112 8-noded isoparametric finite elements with INTER3 contact elements defined across the contact interface. The BEM model used for FEMBEMGAP consists of 50 quadratic boundary elements with a gap element subregion defined in the contact area between the cylinder-subregion and the rectangular body subregion.

The results for the displacement in y-direction of the top edge of the rectangular body (along contact area, at $y=0$) are shown in figure 8.30 for FEMBEMGAP and ABAQUS and agree very well with deviations between the packages smaller than 1%.

In Figure 8.31 the contact pressure results are plotted for FEMBEMGAP, ABAQUS and an analytical solution. The analytical solution which was used is a classical Hertz solution for a cylinder loaded by a point load on its central top point (Johnson(1987)). This point load can be evaluated by adding up the reaction forces in y-direction of the case study and is $P = 4000$ N. The contact pressure results in figure 8.31 of all three solutions show similar elliptical distributions although there are deviations between the solutions. There is a deviation of about 5% between the numerical solutions and the analytical solution for the contact pressure. This deviation can be explained with the fact that the Hertz analytical solution is based on large bodies with contact dimensions being small compared to the size of the bodies which represents an idealisation for this case

study. The contact pressure results of FEMBEMGAP and ABAQUS are very similar and slight deviations result from different algorithms used for the iterative gap opening and closure procedure. However, considering the comparably coarse modelling of the contact area, the results are surprisingly similar.

Figure 8.32 illustrates the stress in y-direction σ_y along the line of symmetry (at $x=0\text{mm}$) of the surface body. It shows that FEMBEMGAP and ABAQUS deliver almost identical results which again deviate from the analytical solution by as much as about 5 % in the region near the contact area and by up to 30 % in portions far away from the contact area. Again this is because the analytical solution is based on large bodies.

It can be concluded that there is very good agreement between FEMBEMGAP and ABAQUS although the numerical solutions and the analytical solution deviate. However this deviation does not indicate that the numerical solutions are inaccurate since it is clear that it is induced by the idealisation used within the analytical solution and shows the limitations of analytical solutions in contact problems. It can be seen that FEMBEMGAP delivers the same accuracy as ABAQUS with the extra advantage of using a pure BE mesh and the comparably simple gap element approach. It is also clear that the iterative procedure applied for the opening and closure of the gap elements as described in section 6.3 works very well for this case study.

8.4 ELASTOPLASTIC CONTACT CASE STUDIES WITH FEM-, BEM- AND GAP ELEMENT-SUBREGIONS

8.4.1 Frictional elastoplastic punch on a large rectangular body

A punching body is pressed onto a surface body under plane strain conditions as shown in figure 8.18. The Young's modulus of both bodies is 1 N/mm^2 with a Poisson's ratio of $\nu=0.3$ and the loading q is 0.1 N/mm^2 and both bodies are 1mm thick. The punching body develops perfectly plastic material behaviour ($H'=0$) at an initial yield stress of $Y_0=0.105 \text{ N/mm}^2$. An initial stress procedure with Von Mises yield criterion is chosen

for the plasticity analysis of FEMBEMGAP. The surface body behaves in a purely elastic manner and the coefficient of friction between the contacting bodies is defined to be $\mu=0.05$.

The case study was modelled by FEMBEMGAP with 2 BEM-subregions, one FEM-subregion and one gap element subregion as shown in figures 8.33 and 8.34. The FEM-subregion consists of 25 8-noded quadrilateral finite elements and is defined in the area of the punching body where yielding is expected. The remainder of the punching body which is expected to be elastic is modelled as the first BEM-subregion with 23 isoparametric boundary elements. The elastic surface body is defined as the second BEM-subregion with 30 isoparametric boundary elements. The contact area is modelled with a gap element subregion which consists of 21 gap elements. There is an internal FEM/BEM interface corner which has been modelled as described in section 4.5.3. The results gained by FEMBEMGAP were compared to the results obtained by an ABAQUS analysis with the same mesh as already used for the elastic case (figure 8.20). As far as modelling effort is concerned it is obvious that FEMBEMGAP is superior to ABAQUS with altogether 212 nodes and 88 finite and boundary elements for coupled FEM/BEM compared to 1233 nodes and 360 finite elements for the pure FEM mesh.

The displacement in y-direction of the surface body at its upper edge (along the contact area at $y=0$ mm) is displayed in figure 8.35. FEMBEMGAP and ABAQUS deviate from each other by less than 3% near the edge of the punch whilst everywhere else the deviations are smaller than 1%. The larger deviations near the edge of the punch result from a slight overlap of the edge of the punching body and the surface body in FEMBEMGAP which is caused by the node-to-node contact modelling of the gap elements.

The relative tangential displacement (sliding) along the contact interface is shown in figure 8.36. Although the distributions are similar, the FEMBEMGAP values are on average about 12% smaller than the ones obtained by ABAQUS. This large deviation also results from the previously mentioned edge overlap.

Figure 8.37 illustrates the contact pressure. The contact pressure modelling of FEMBEMGAP and ABAQUS produces very similar results with a maximum difference of 3 % in the area near the edge of the punch and agreement within 1% deviation in the area up to $x=0.7$ mm. This proves that the contact pressure transmission of FEMBEMGAP in this case study works well.

In figure 8.38 the Von Mises stress along the contact area is plotted. It can be seen that the FEMBEMGAP results are 5-7% lower than the ABAQUS results in the elastic area (up to $x=0.75$ mm). The yielding zone of ABAQUS is slightly larger than the yielding zone of FEMBEMGAP but both packages model the plastic material behaviour ($H'=0$) accurately. Again the differences can be explained with the overlap at the edge of the punch of FEMBEMGAP.

In figure 8.39 the Von Mises stress along the edge of the punching body (at $x=1$ mm) is illustrated. Both packages deliver very similar distributions with deviations being smaller than 5% and the already mentioned smaller yielding zone of FEMBEMGAP can be seen.

In this case study ABAQUS seems to produce more accurate results than FEMBEMGAP because it is able to model the corner at the edge of the punch more accurately. With the node-to-node contact of the used gap elements a slight overlap occurs at the edge of the punch which could not even be corrected with remeshing the contact area. This overlap leads to larger normal displacements and smaller relative tangential displacements in the contact area and a smaller stress concentration at the edge of the punch, which explains why the plastic zones evaluated by FEMBEMGAP are smaller than the corresponding ABAQUS ones. It can be concluded that the deviations obtained for this case study are due to the limitations in the contact modelling capability of the node to node contact algorithm in comparison to the surface contact algorithms of ABAQUS. However the gap element algorithm is much simpler than a surface contact algorithm and the deviations obtained in this case study are still in an acceptable range. Therefore the FEMBEMGAP model delivers acceptably accurate results with a significant reduction in modelling effort compared to the pure FEM solution. The

partitioning of the structure into FEM-, BEM- and gap element-subregions was found to represent a simple and clearly laid out approach to the complex nature of this problem. It can also be concluded that the internal corner modelling did not introduce any recognisable errors into the presented results.

8.4.2 Elastoplastic frictionless contact of a cylinder and a large rectangular body

Similarly to the already presented elastic cylinder contact case, a cylinder is pressed onto a rectangular surface body under plane stress conditions as shown in figure 8.27. The contacting bodies have the same loading conditions ($v=-1.2$ mm) and material data ($E=10000$ N/mm² and $\nu=0.3$) as in the elastic case except that the cylindrical body is defined to have an initial yield point at $Y_0=580$ N/mm² and constant hardening as shown in figure 8.40.

The FEMBEMGAP model consists of 2 BEM-subregions, 1 FEM-subregion and 1 gap element subregion as shown in figures 8.41 and 8.42. The FE-subregion is defined in the stress concentration area where plastic deformation is expected and consists of 16 8-noded quadrilateral finite elements. The first BE-subregion defines the elastic remainder of the cylindrical body and consists of 24 isoparametric boundary elements. The second BE-subregion is the elastic surface body and is modelled with 26 isoparametric boundary elements. The contact area is modelled with a gap element subregion consisting of 14 nodes and 7 gap elements as shown in figure 8.42. Figure 8.42 also shows that there is an internal corner on the FEM-BEM-interface which requires modelling as defined in section 4.5.3. The analysis results of FEMBEMGAP have been compared to ABAQUS results obtained with the same mesh as in the elastic cylinder case (112 elements, figure 8.29). It can be seen that the FEMBEMGAP model is superior since less elements and nodes are required. The Von Mises yield criterion was used for FEMBEMGAP as well as for ABAQUS and for the FEMBEMGAP package an initial strain procedure was chosen for the plasticity iterations since it is known to converge quicker than initial stress procedures for cases with constant strain hardening (Guzelbey (1992)).

The results of the displacements in y-direction of the upper edge of the rectangular

surface body (= along contact area) for FEMBEMGAP and ABAQUS are plotted in figure 8.43. FEMBEMGAP and ABAQUS deliver almost identical results with deviations smaller than 1%.

The results of the normal contact pressure along the contact area are illustrated in figure 8.44. Again FEMBEMGAP and ABAQUS show similar distributions with acceptable deviations which are smaller than 6%. Similarly to the elastic cylinder/surface case, these deviations result from differences in iterative procedures of the contact algorithms used by the packages. It is obvious that in the contact region between $x=0.0\text{mm}$ and $x=2.1\text{mm}$ the otherwise elliptical pressure distribution is disturbed because in this region the material is yielding.

The Von Mises stresses along the contact area of FEMBEMGAP and ABAQUS in figure 8.45 agree even better to one another than the contact pressure results and show that the yielding area with Von Mises stress values larger than 580 N/mm^2 indeed lies between $x=0.0\text{mm}$ and $x=2.1\text{mm}$.

Figure 8.46 illustrates the Von Mises stress distribution of the cylindrical body along the axis of symmetry (at $x=0\text{mm}$). Again FEMBEMGAP and ABAQUS deliver very similar results. The yielding zone with Von Mises stresses larger than the initial yield value is between $y=0.0\text{mm}$ and $y=3.5\text{mm}$. Similarly to the Von Mises stress distribution along the contact area the yield initiation point at $y=3.5\text{mm}$ can be clearly seen in the graphs since there is a change in the stress gradient at this point.

Considering the highly non-linear nature of this case study, the results of FEMBEMGAP and ABAQUS are in extremely good agreement which indicates that both packages deliver accurate results. It also proves again that the internal FEM/BEM interface corner modelling as suggested in section 4.5.3 works accurately. As far as modelling effort is concerned FEMBEMGAP is superior to the pure FEM analysis since it requires less nodes and elements. The modelling of the different parts of the structure and the contact conditions with subregions as shown in figure 8.42 was found to have benefits in data input and data processing.

Table 8.1 Comparison of the FEM/BEM and theoretical displacement results at 15 points for the patch cases of sections 8.1.1 and 8.1.2.

Point	u theoret. in 10^{-4} mm	u FEMBEM in 10^{-4} mm	Error in %	v theoret. in 10^{-4} mm	v FEMBEM in 10^{-4} mm	Error in %
1	0.0	0.0000	-	0.0	0.0000	-
2	1.0	1.0000	0.0000	0.0	0.0000	-
3	2.0	2.0000	0.0000	0.0	0.0000	-
4	2.0	2.0000	0.0000	-0.3	-0.3000	0.0000
5	2.0	2.0000	0.0000	-0.6	-0.6000	0.0000
6	1.0	1.0000	0.0000	-0.6	-0.6000	0.0000
7	0.0	0.0000	-	-0.6	-0.6000	0.0000
8	0.0	0.0000	-	-0.3	-0.3000	0.0000
9	3.0	3.0000	0.0000	0.0	0.0000	-
10	4.0	4.0000	0.0000	0.0	0.0000	-
11	4.0	4.0000	0.0000	-0.3	-0.3000	0.0000
12	4.0	4.0000	0.0000	-0.6	-0.6000	0.0000
13	3.0	3.0000	0.0000	-0.6	-0.6000	0.0000
14	3.0	3.0000	0.0000	-0.3	-0.3000	0.0000
15	3.0	3.0000	0.0000	-0.15	-0.1508	0.5333

Table 8.2 Comparison of the FEM/BEM and theoretical stress results at 15 points for the patch cases of sections 8.1.1 and 8.1.2.

Point	σ_x theoret. in N/mm ²	σ_x FEMBEM in N/mm ²	Error in %	σ_y theoret. in N/mm ²	σ_y FEMBEM in N/mm ²
1	100	100.0000	0.0000	0.0	0.0000
2	100	100.0000	0.0000	0.0	0.0000
3	100	100.0000	0.0000	0.0	0.0000
4	100	100.0000	0.0000	0.0	0.0000
5	100	100.0000	0.0000	0.0	0.0000
6	100	100.0000	0.0000	0.0	0.0000
7	100	100.0000	0.0000	0.0	0.0000
8	100	100.0000	0.0000	0.0	0.0000
9	100	100.0000	0.0000	0.0	0.0000
10	100	100.0000	0.0000	0.0	0.0000
11	100	100.0000	0.0000	0.0	0.0000
12	100	100.0000	0.0000	0.0	0.0000
13	100	100.0000	0.0000	0.0	0.0000
14	100	99.9890	0.0110	0.0	0.0109
15	100	99.2186	0.7814	0.0	0.8053

Table 8.3 Comparison of the FEM/BEM and theoretical displacement results at 19 points for the patch case with corner of section 8.1.3.

Point	u theoret. in 10^{-4} mm	u FEMBEM in 10^{-4} mm	Error in %	v theoret. in 10^{-4} mm	v FEMBEM in 10^{-4} mm	Error in %
1	0.0	0.0000	-	0.0	0.0000	-
2	2.0	2.0000	0.0000	-0.6	-0.6000	0.0000
3	0.0	0.0000	-	-1.2	-1.2000	0.0000
4	2.0	2.0001	0.0050	-1.2	-1.2000	0.0000
5	3.0	3.0008	0.0260	-1.2	-1.1974	0.2167
6	3.5	3.4957	0.1229	-1.2	-1.2038	0.3167
7	4.0	4.0041	0.1025	-1.2	-1.2031	0.2583
8	4.0	4.0058	0.1450	-1.05	-1.0469	0.2952
9	4.0	3.9965	0.0875	-0.9	-0.9004	0.0444
10	4.0	3.9998	0.0050	-0.6	-0.6001	0.0166
11	4.0	4.0000	0.0000	0.0	0.0000	-
12	0.0	0.0000	-	-2.4	-2.4000	0.0000
13	4.0	4.0000	0.0000	-2.4	-2.4000	0.0000
14	8.0	8.0000	0.0000	-2.4	-2.4000	0.0000
15	8.0	8.0000	0.0000	-1.2	-1.2000	0.0000
16	8.0	8.0000	0.0000	0.0	0.0000	0.0000
17	2.0	2.0000	0.0000	-1.8	-1.8000	0.0000
18	6.0	6.0000	0.0000	-1.8	-1.8000	0.0000
19	6.0	6.0000	0.0000	-0.6	-0.6002	0.0333

Table 8.4 Comparison of the FEM/BEM and theoretical stress results at 19 points for the patch case with corner of section 8.1.3.

Point	σ_x theoret. in N/mm ²	σ_x FEMBEM in N/mm ²	Error in %	σ_y theoret. in N/mm ²	σ_y FEMBEM in N/mm ²
1	100	100.0003	0.0003	0.0	0.0006
2	100	100.0104	0.0104	0.0	-0.0092
3	100	99.9974	0.0026	0.0	-0.0049
4	100	99.9435	0.0565	0.0	-0.1195
5	100	99.0987	0.9013	0.0	0.5579
6	100	99.99265	0.0074	0.0	-1.1137
7	100	102.5860	2.5860	0.0	-1.4594
8	100	101.6049	1.6049	0.0	0.2162
9	100	99.0596	0.9404	0.0	0.5248
10	100	100.1337	0.1337	0.0	0.0206
11	100	100.0150	0.0150	0.0	0.0002
12	100	99.9999	0.0001	0.0	0.0000
13	100	100.0002	0.0002	0.0	0.0000
14	100	100.0000	0.0000	0.0	0.0000
15	100	100.0000	0.0000	0.0	0.0010
16	100	100.0000	0.0000	0.0	-0.1358
17	100	100.0013	0.0013	0.0	0.0015
18	100	99.9983	0.0017	0.0	0.0014
19	100	100.0005	0.0005	0.0	0.0012

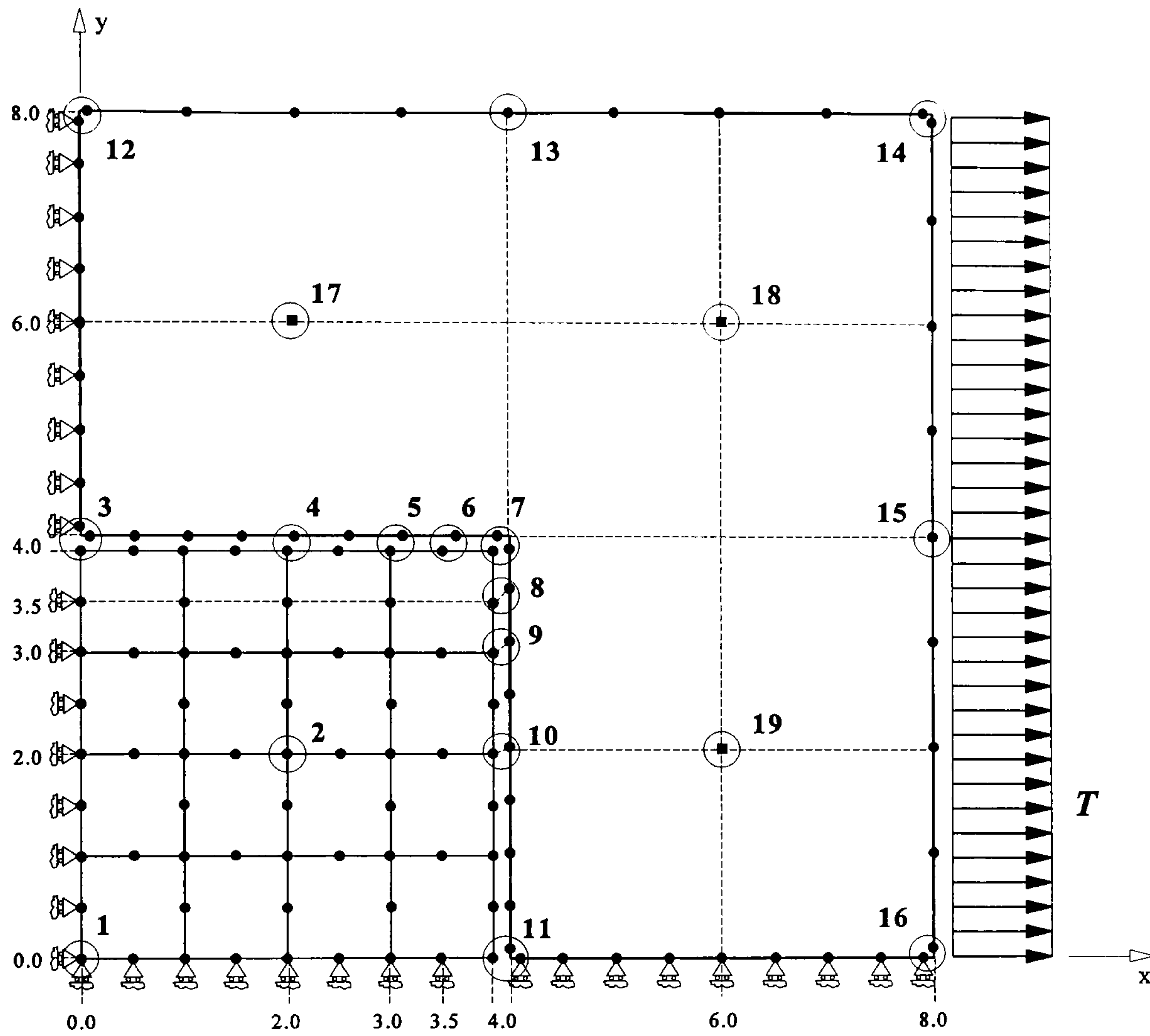


Figure 8.3 Model of the patch test with an internal interface corner.

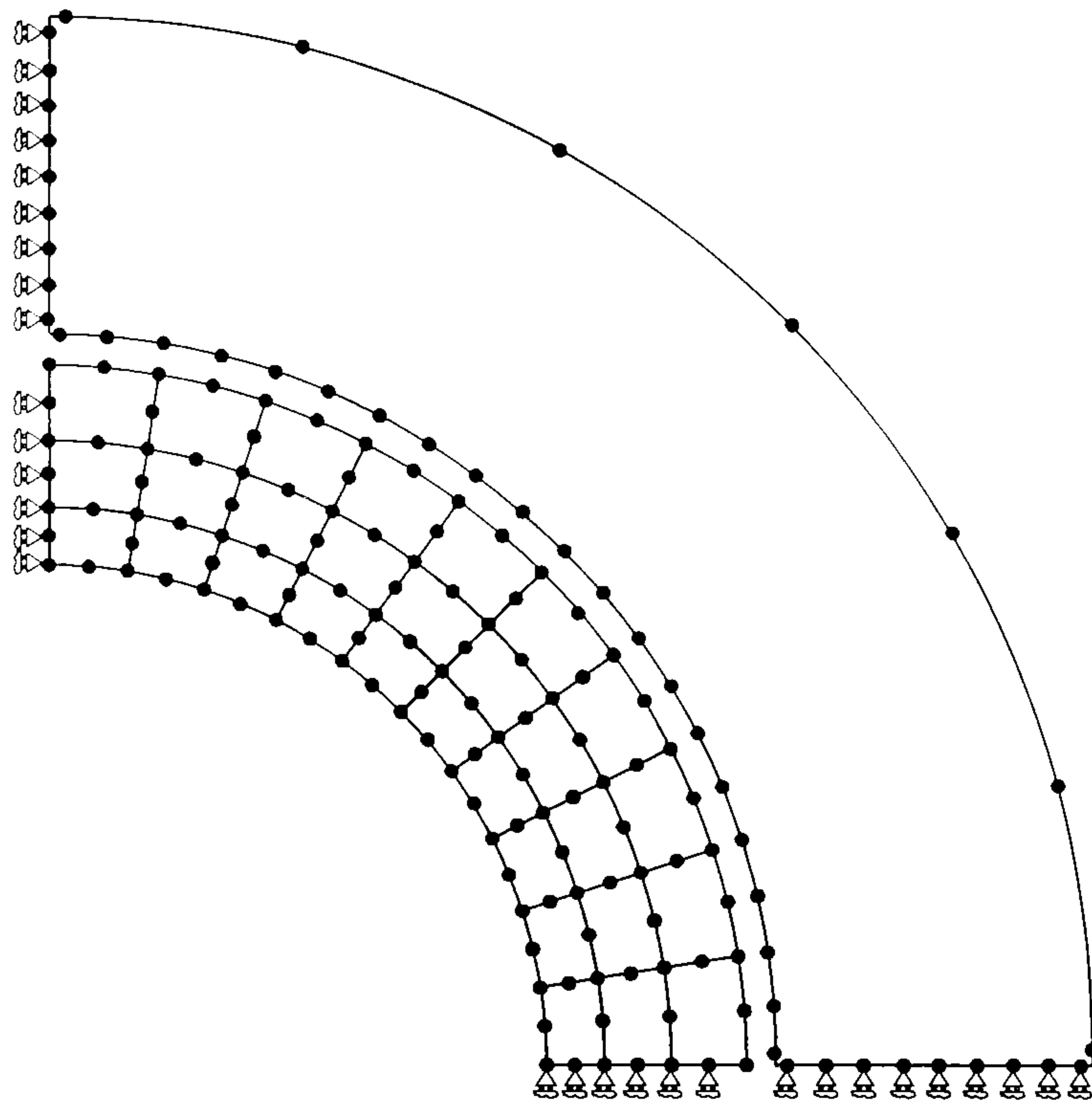


Figure 8.4 Model of the cylinder with coupled FEM/BEM.

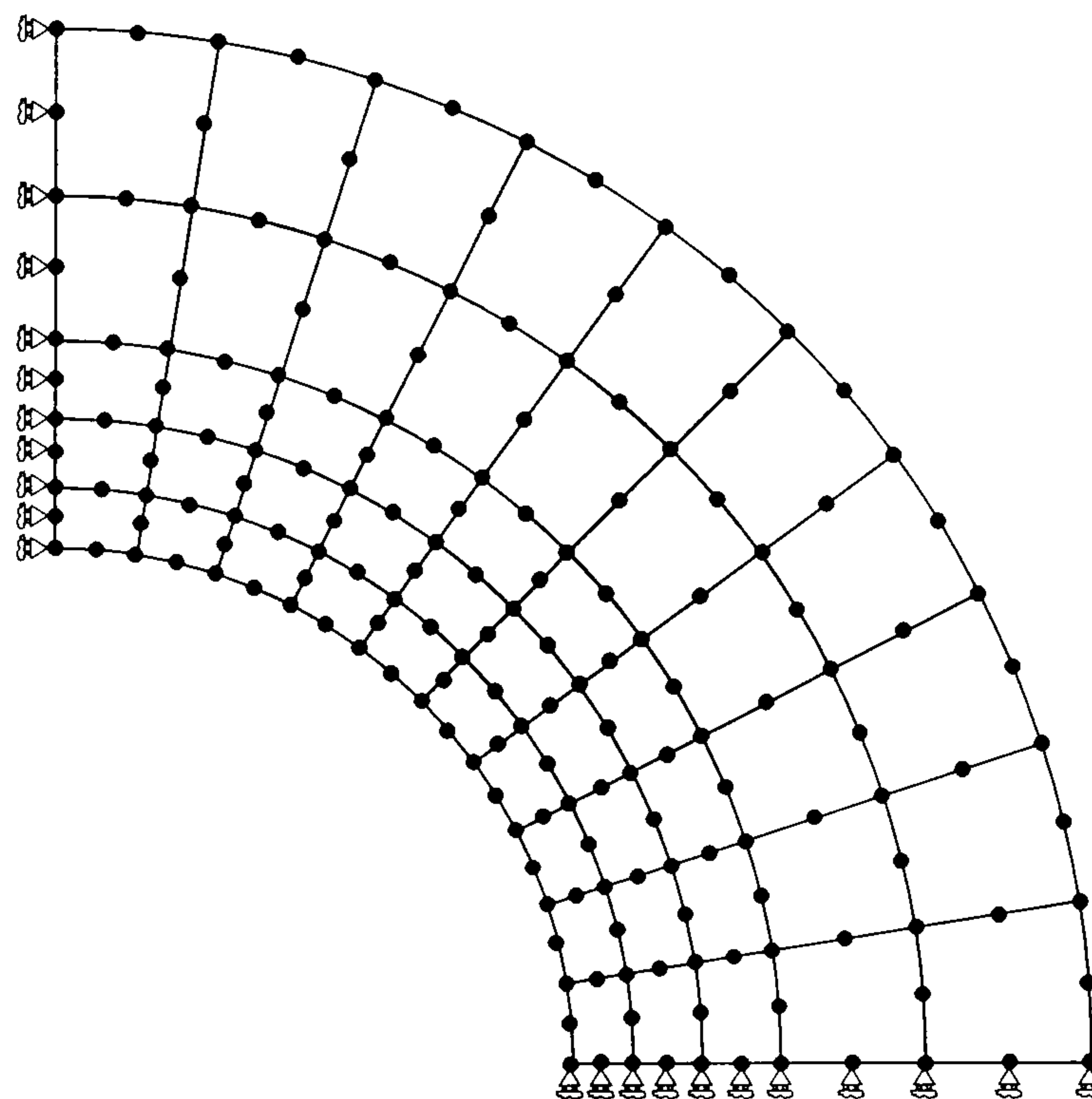


Figure 8.5 Model of the cylinder with FEM.

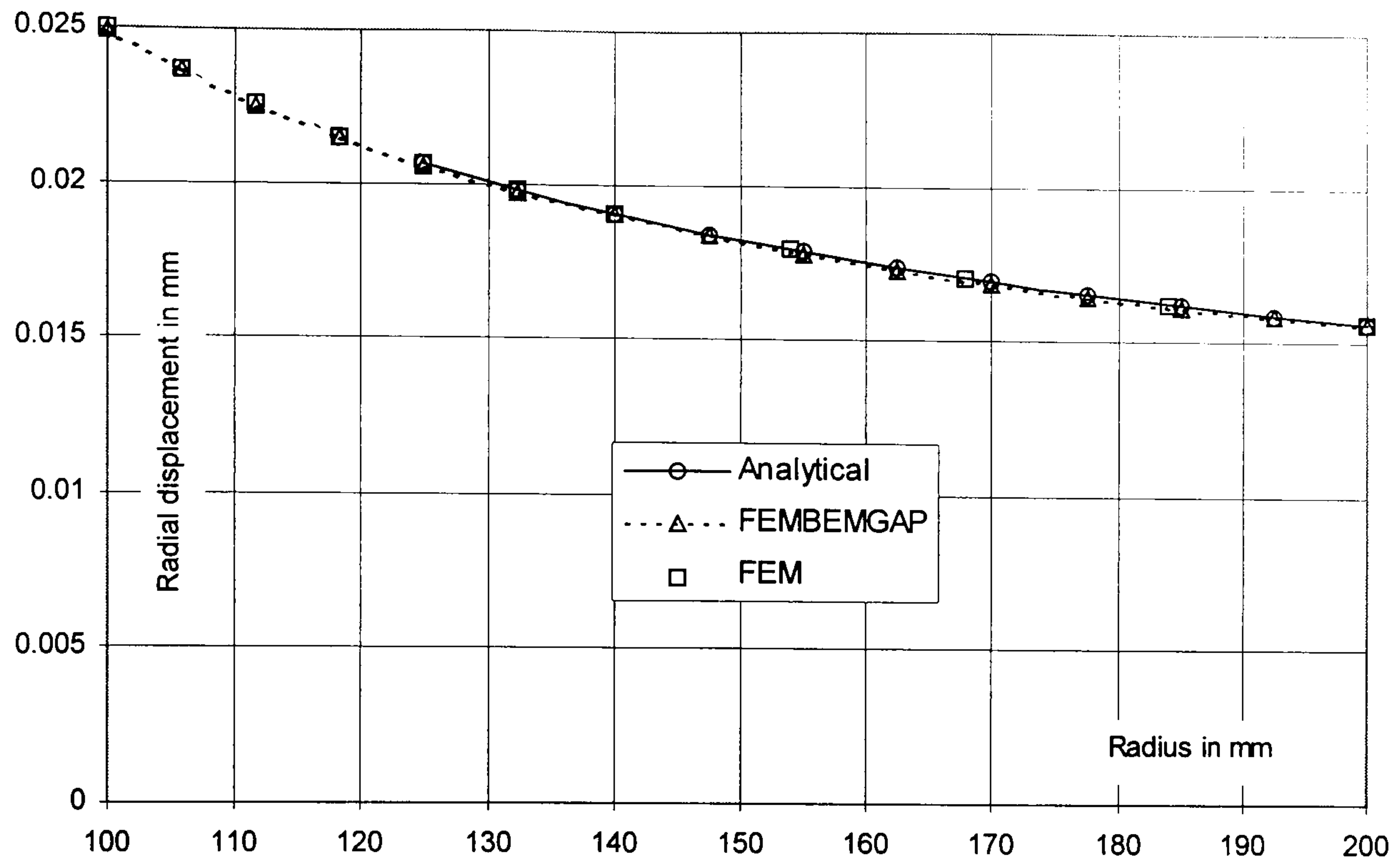


Figure 8.6 Radial displacements of the pressurised cylinder.

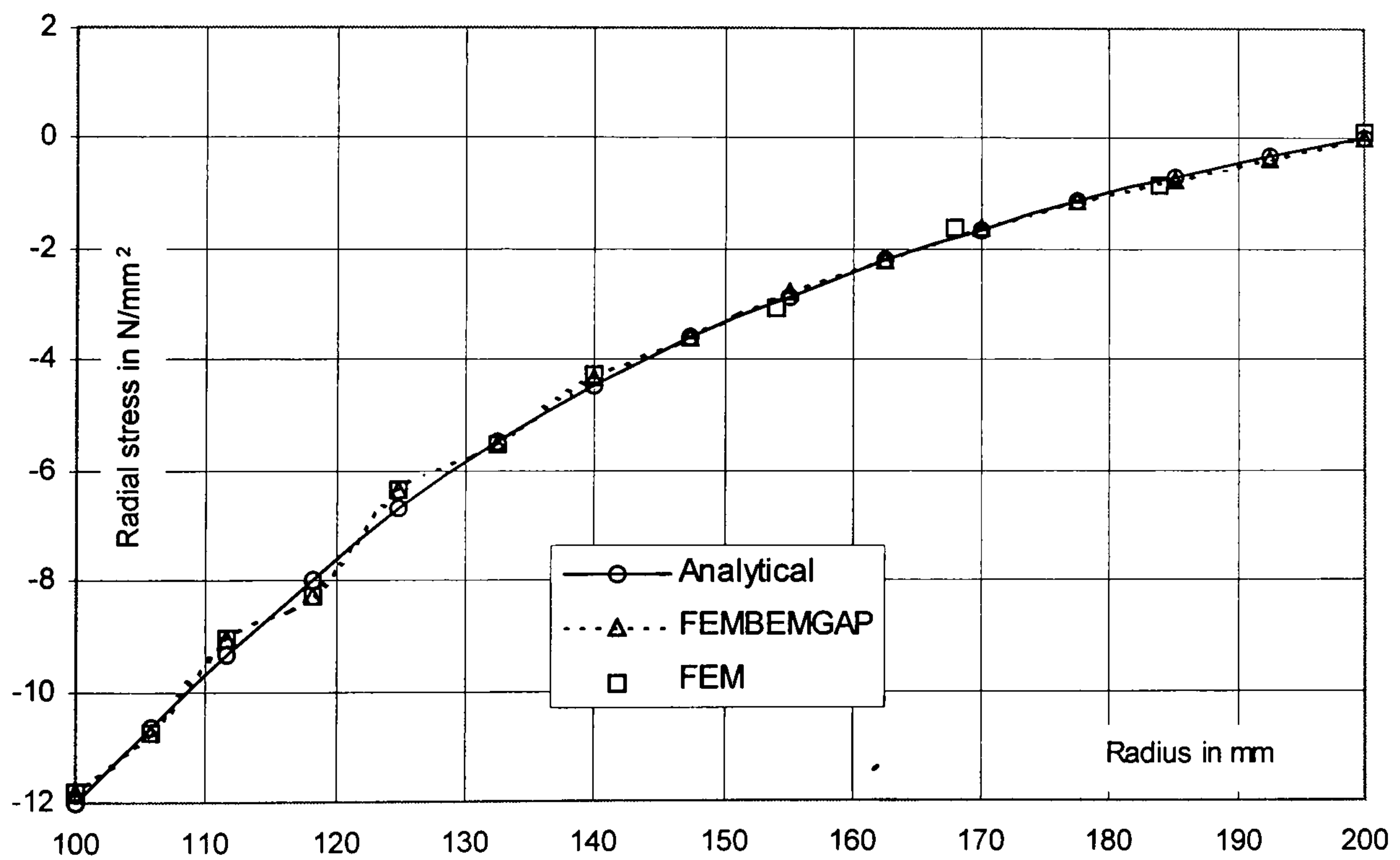


Figure 8.7 Radial stress of the pressurised cylinder.

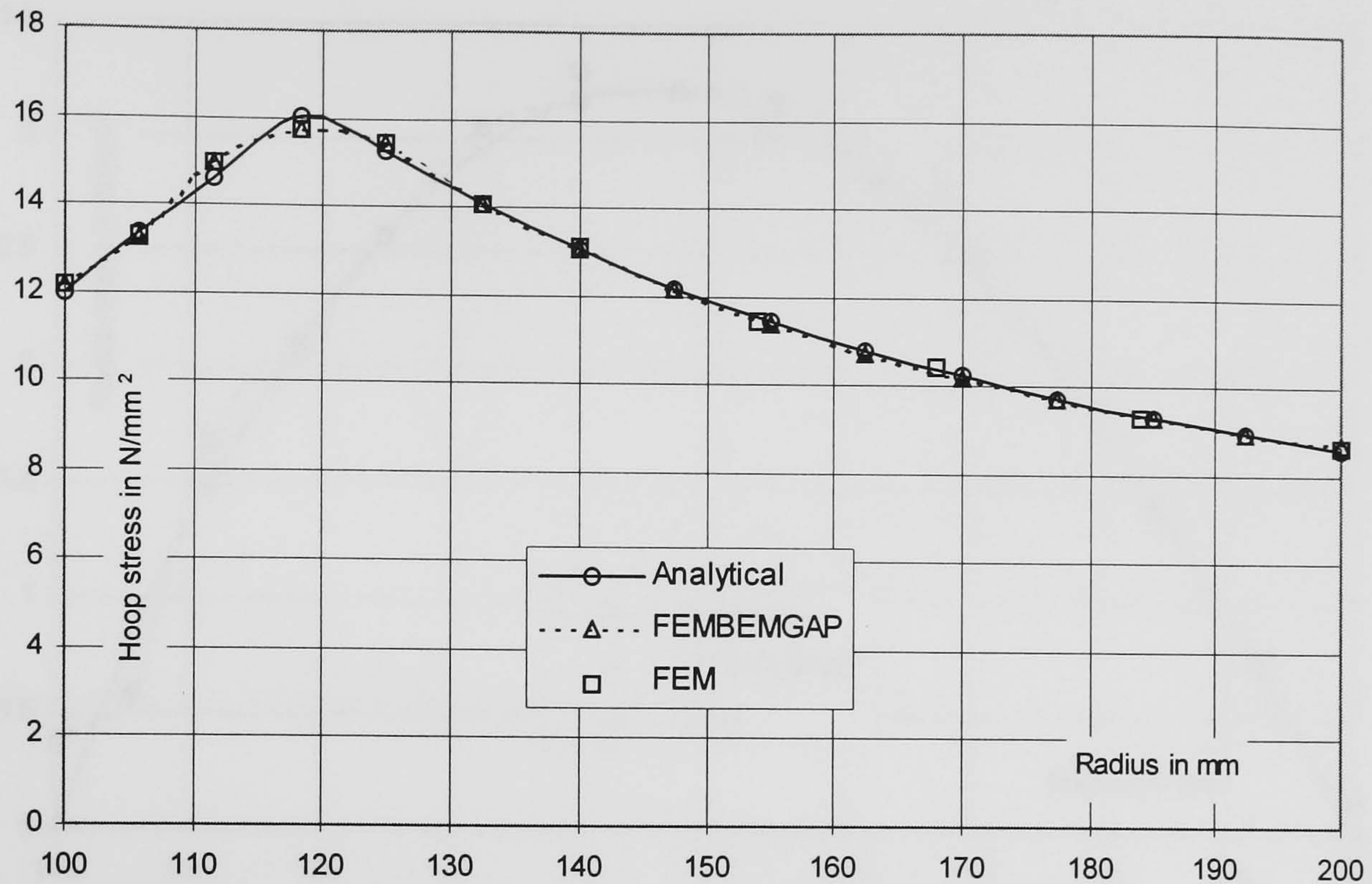


Figure 8.8 Hoop stress of the pressurised cylinder.

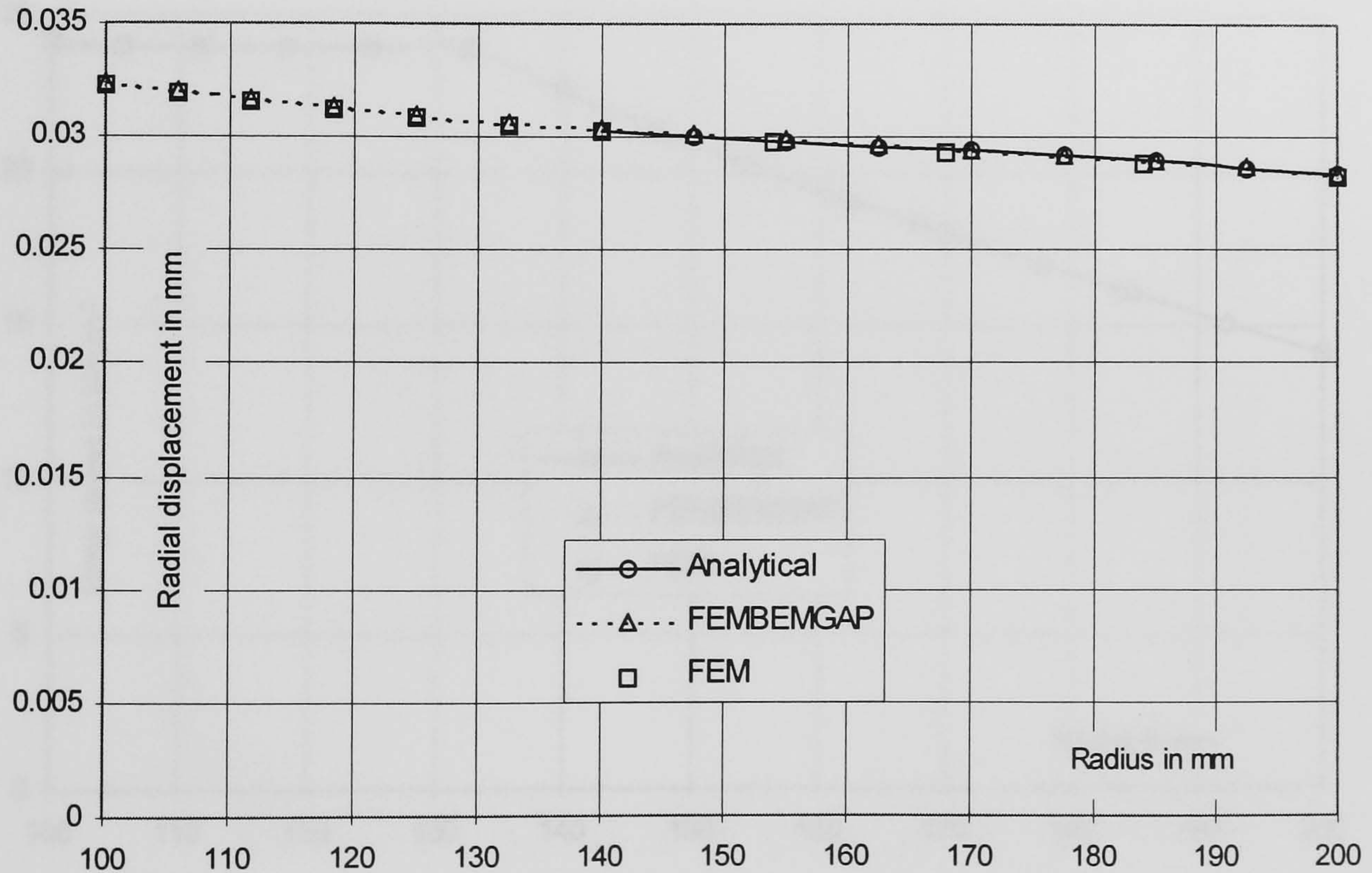


Figure 8.9 Radial displacement of the rotating disc.

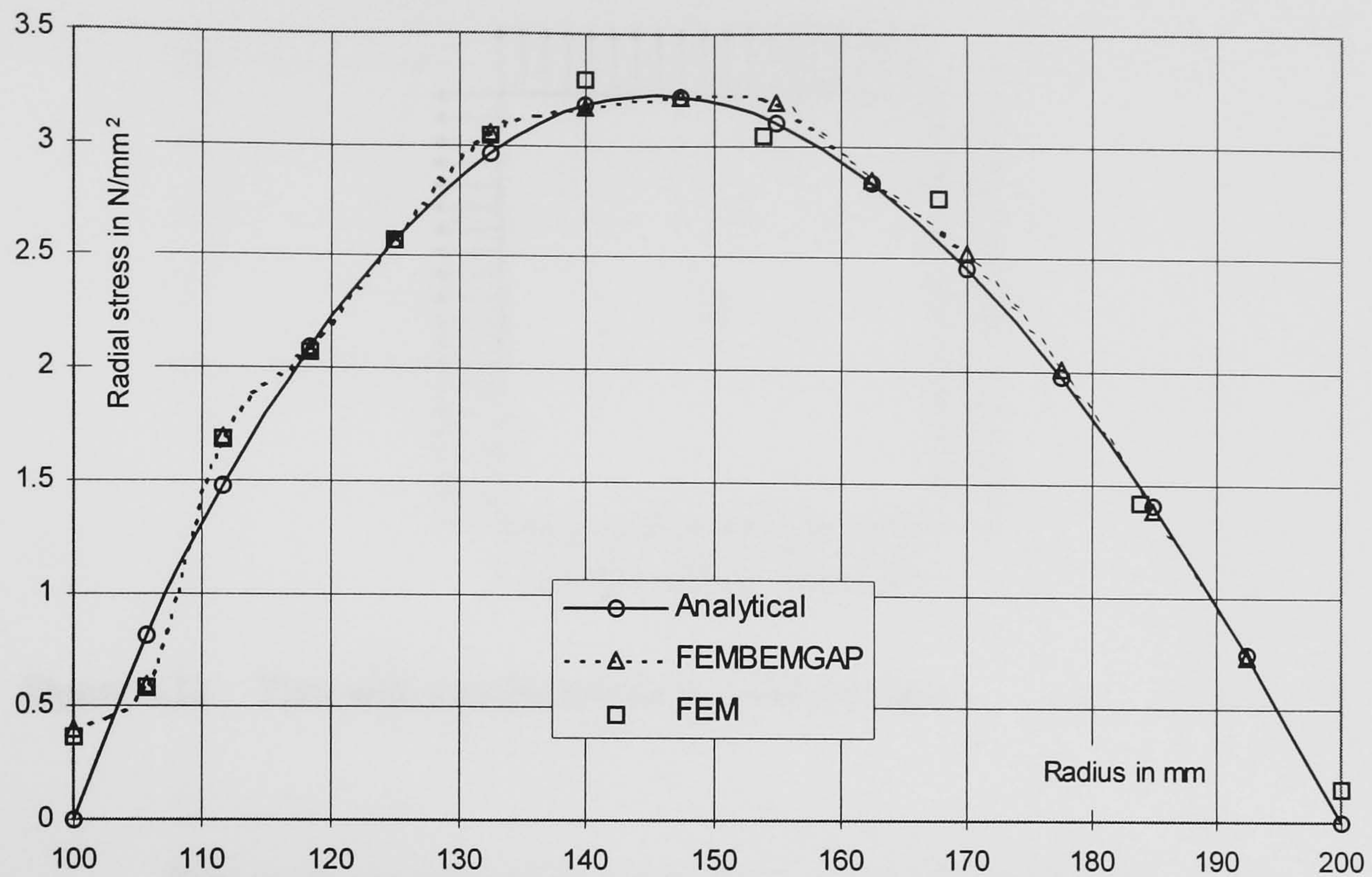


Figure 8.10 Radial stress of the rotating disc.

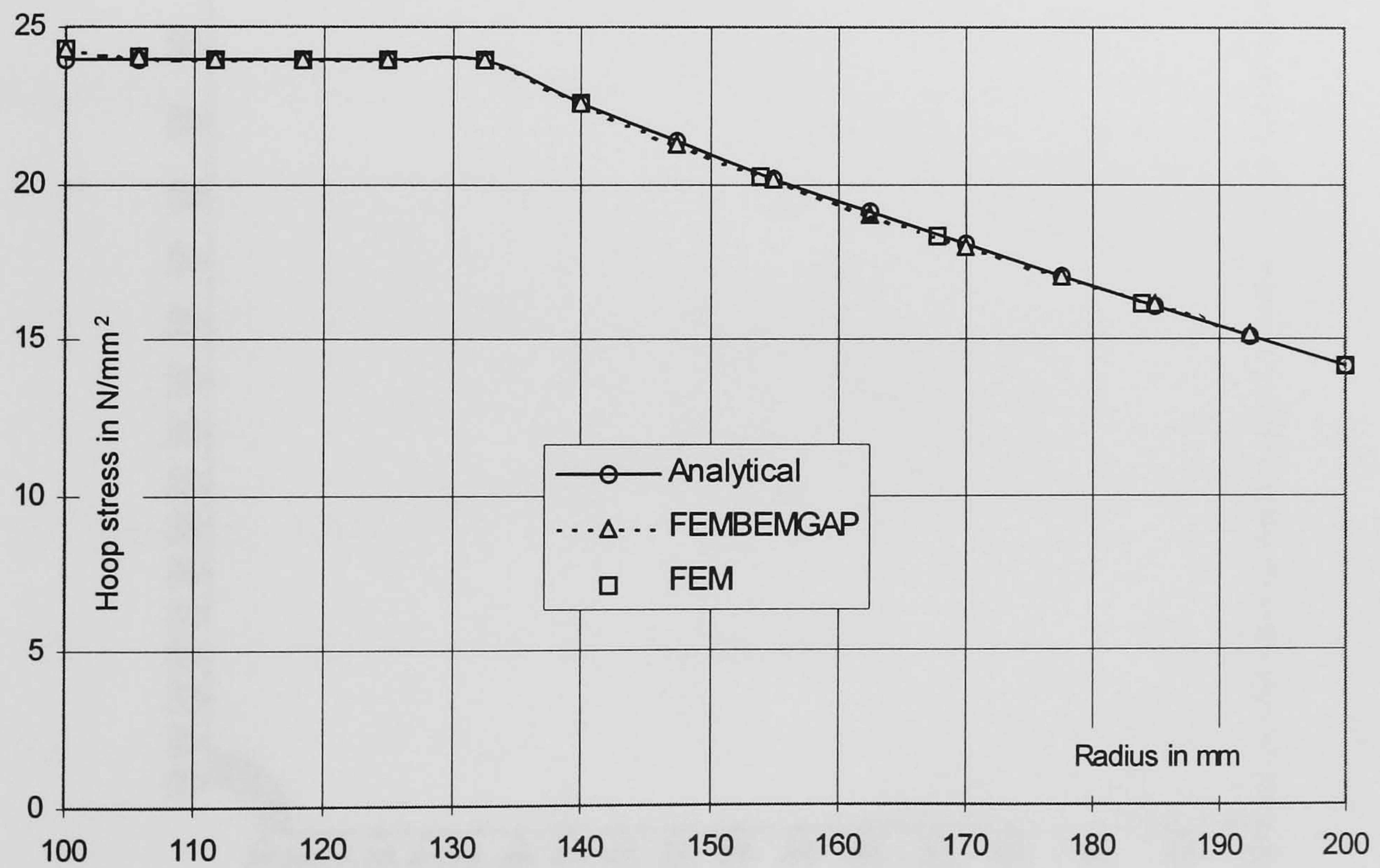


Figure 8.11 Hoop stress of the rotating disc.

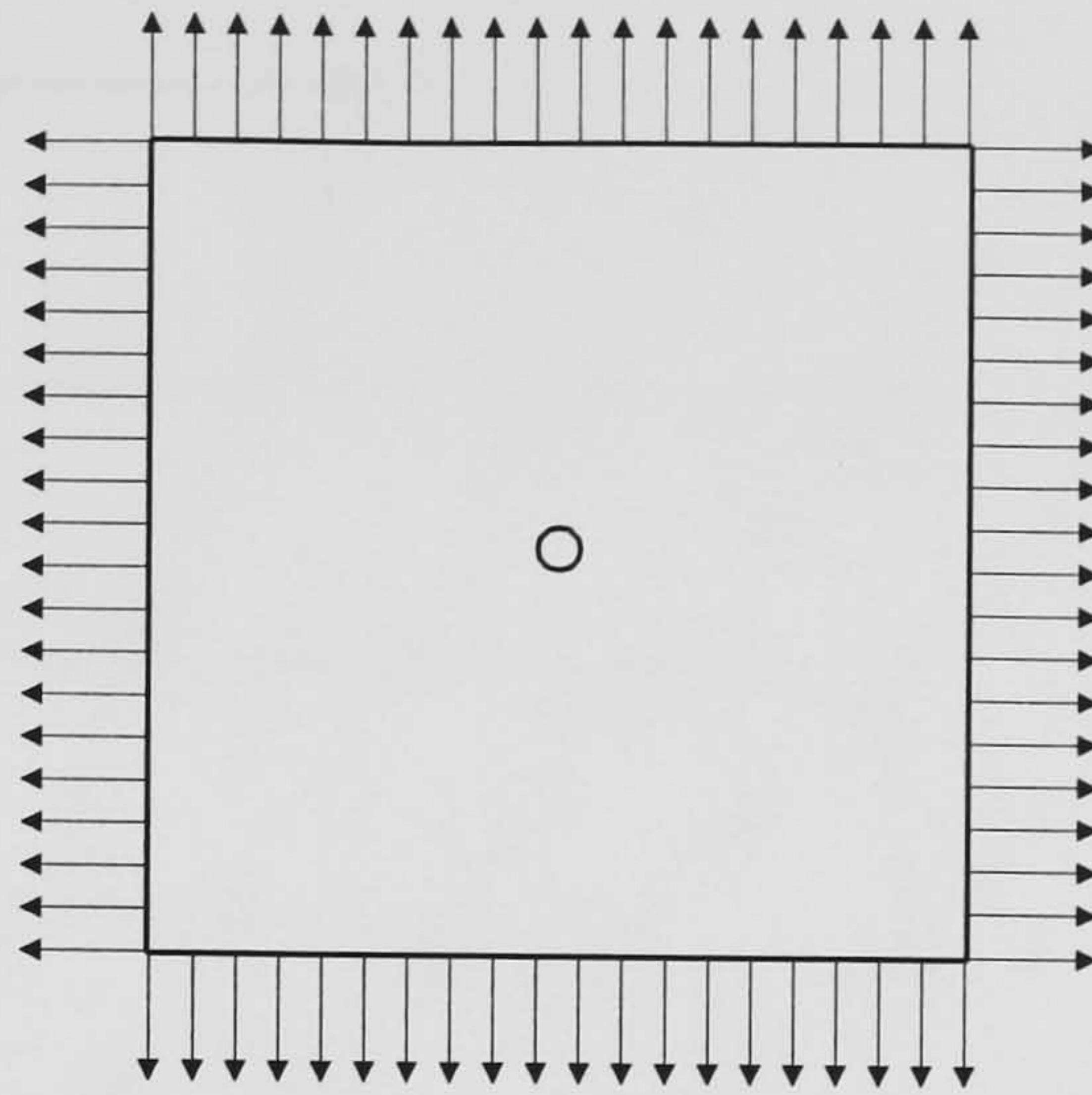


Figure 8.12 Plate with a small hole under biaxial tension.

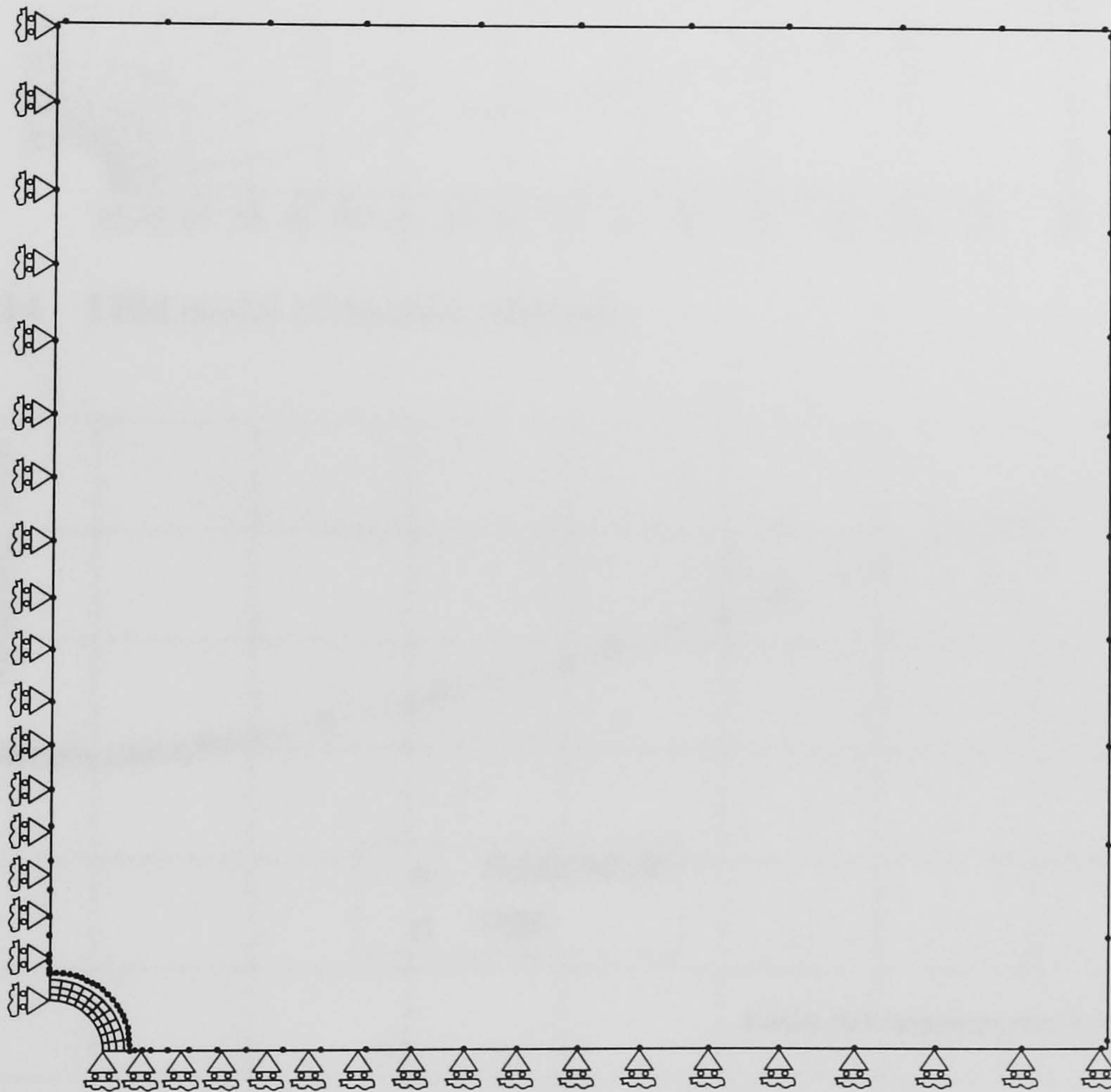


Figure 8.13 FEM/BEM model of the plate with a hole.

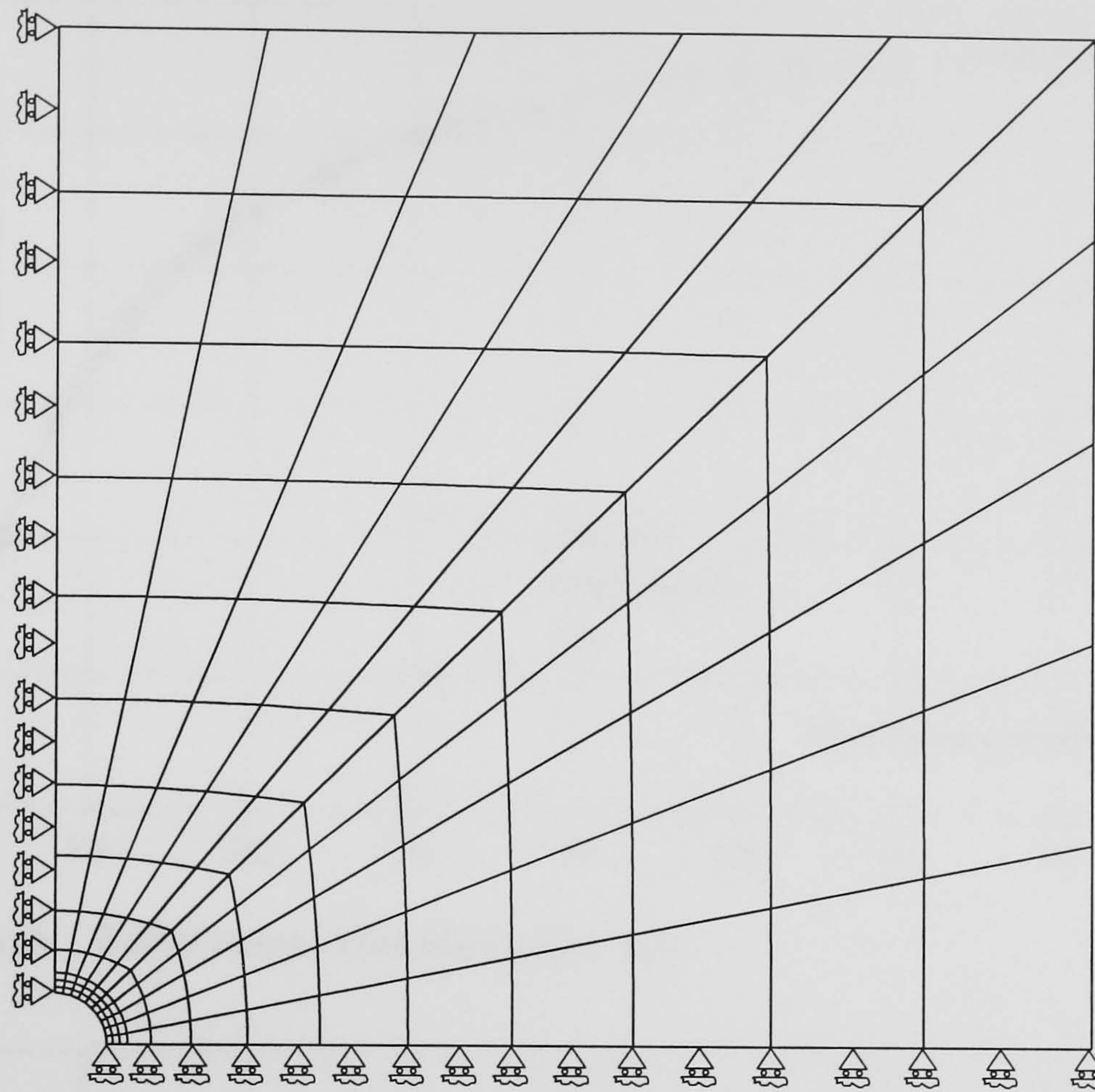


Figure 8.14 FEM model of the plate with hole.

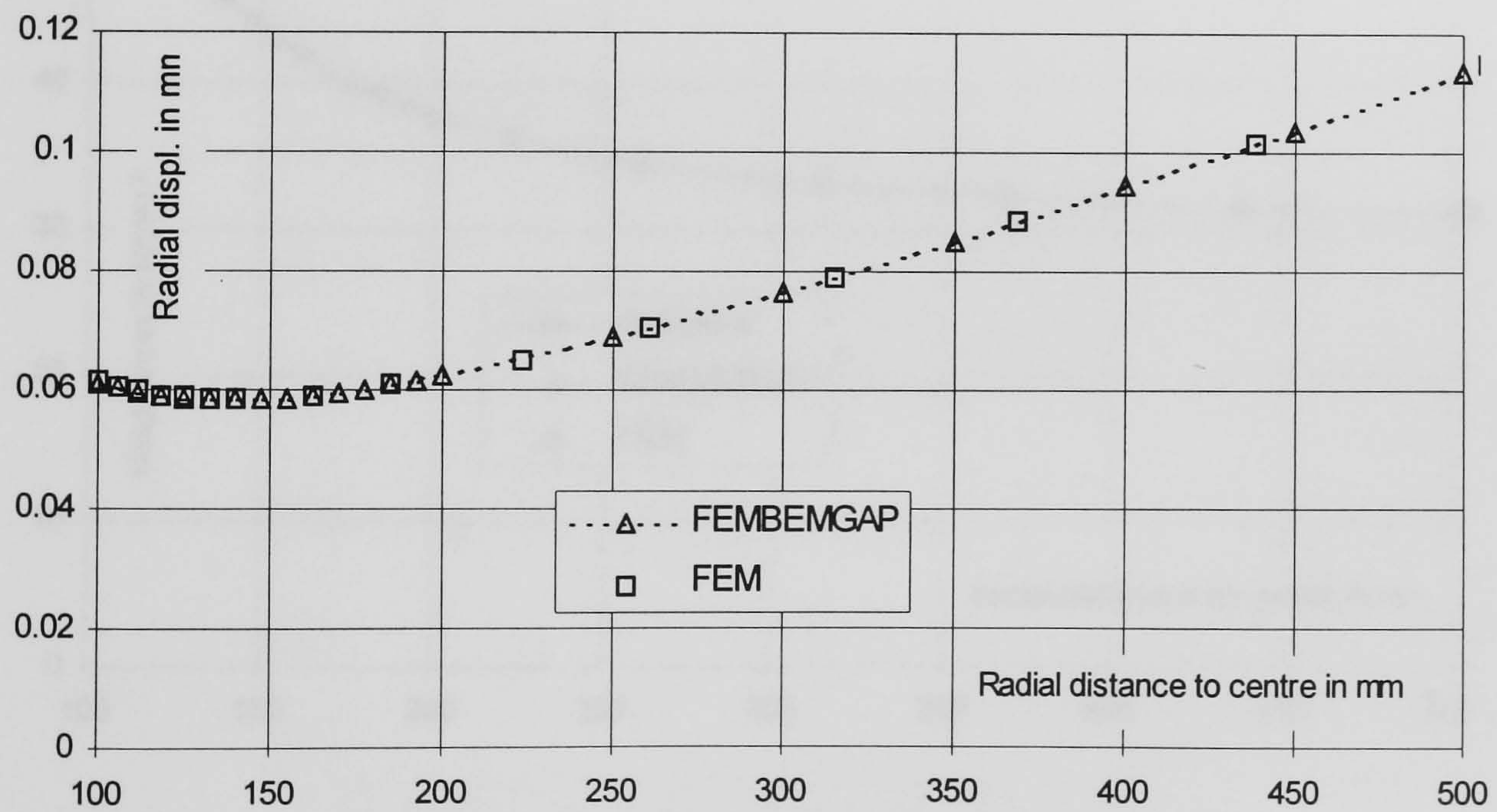


Figure 8.15 Radial displacement of the plate with a hole.

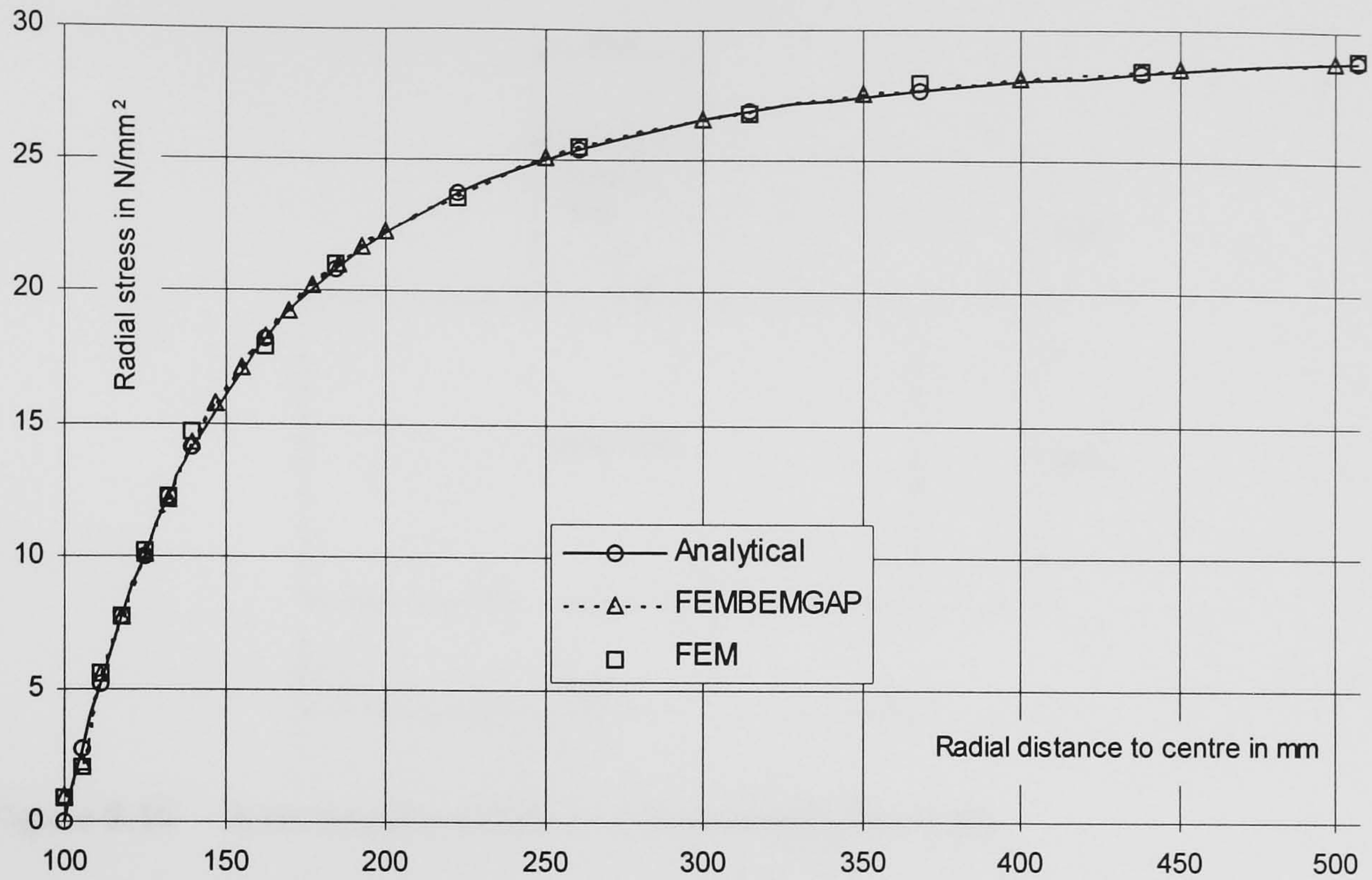


Figure 8.16 Radial stress of the plate with a hole.

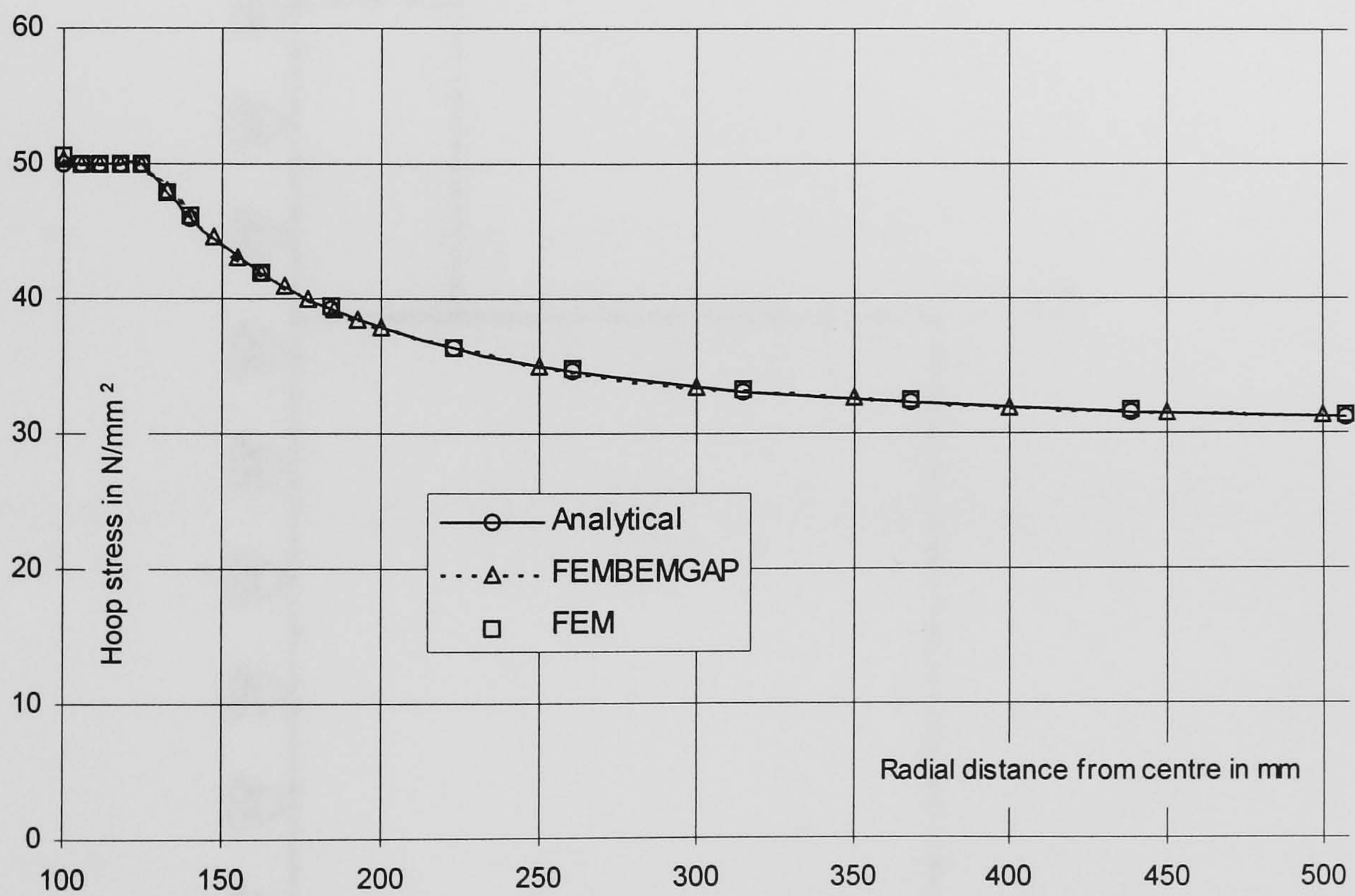


Figure 8.17 Hoop stress of the plate with a hole.

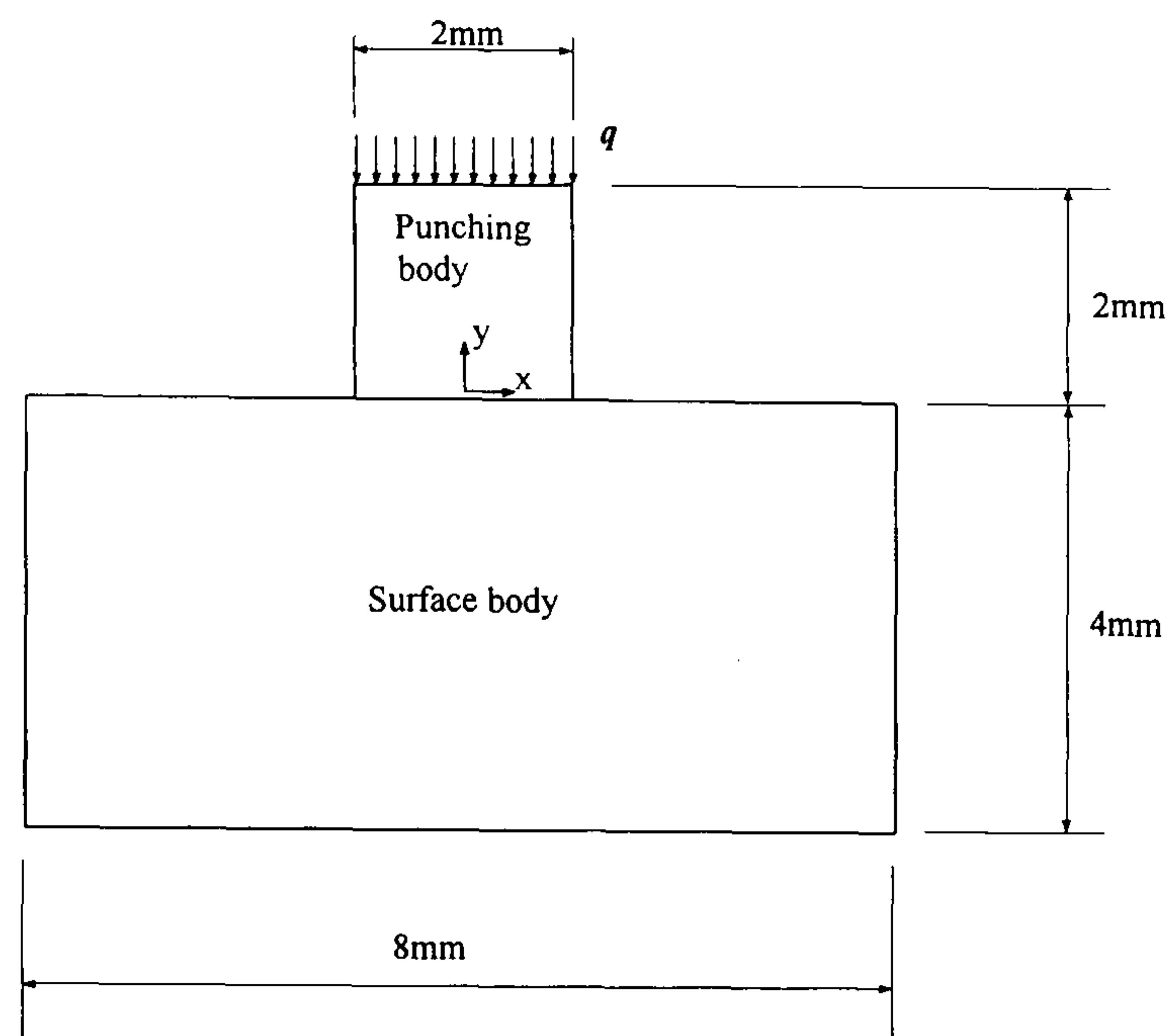


Figure 8.18 A rectangular punch on a large rectangular body.

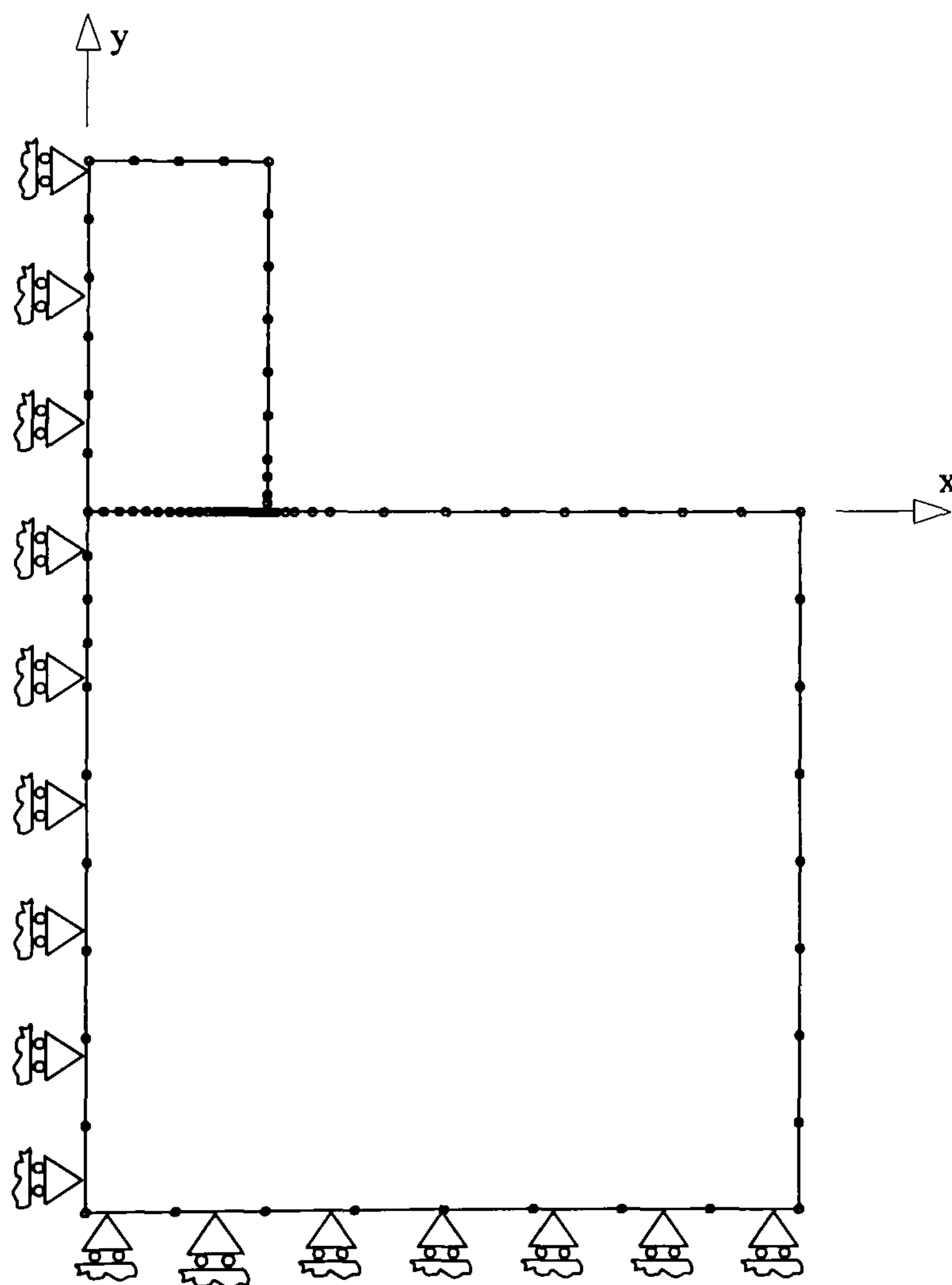


Figure 8.19 BEM model of the rectangular punch on a rectangular surface.

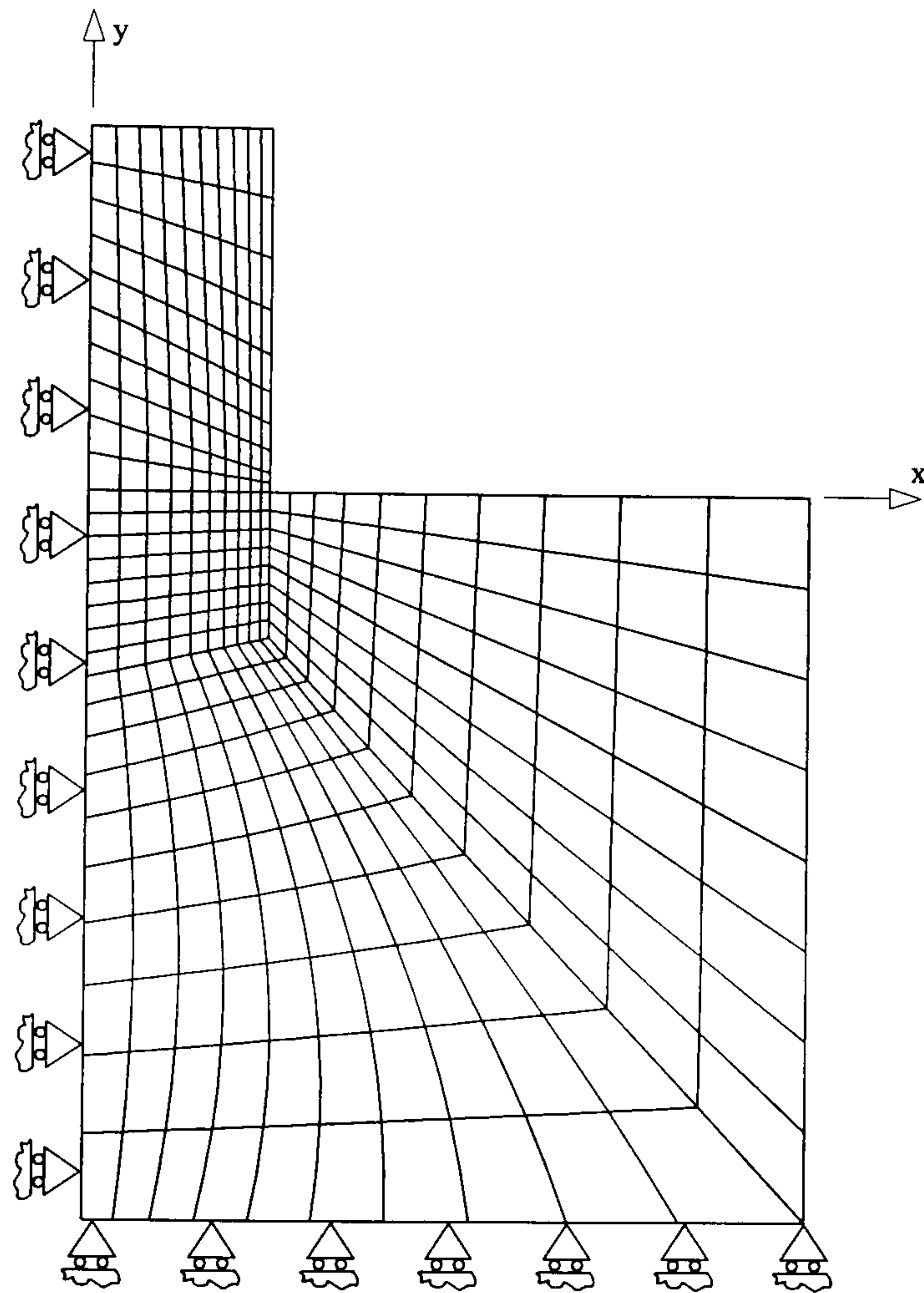


Figure 8.20 FEM model of the rectangular punch on a rectangular surface.

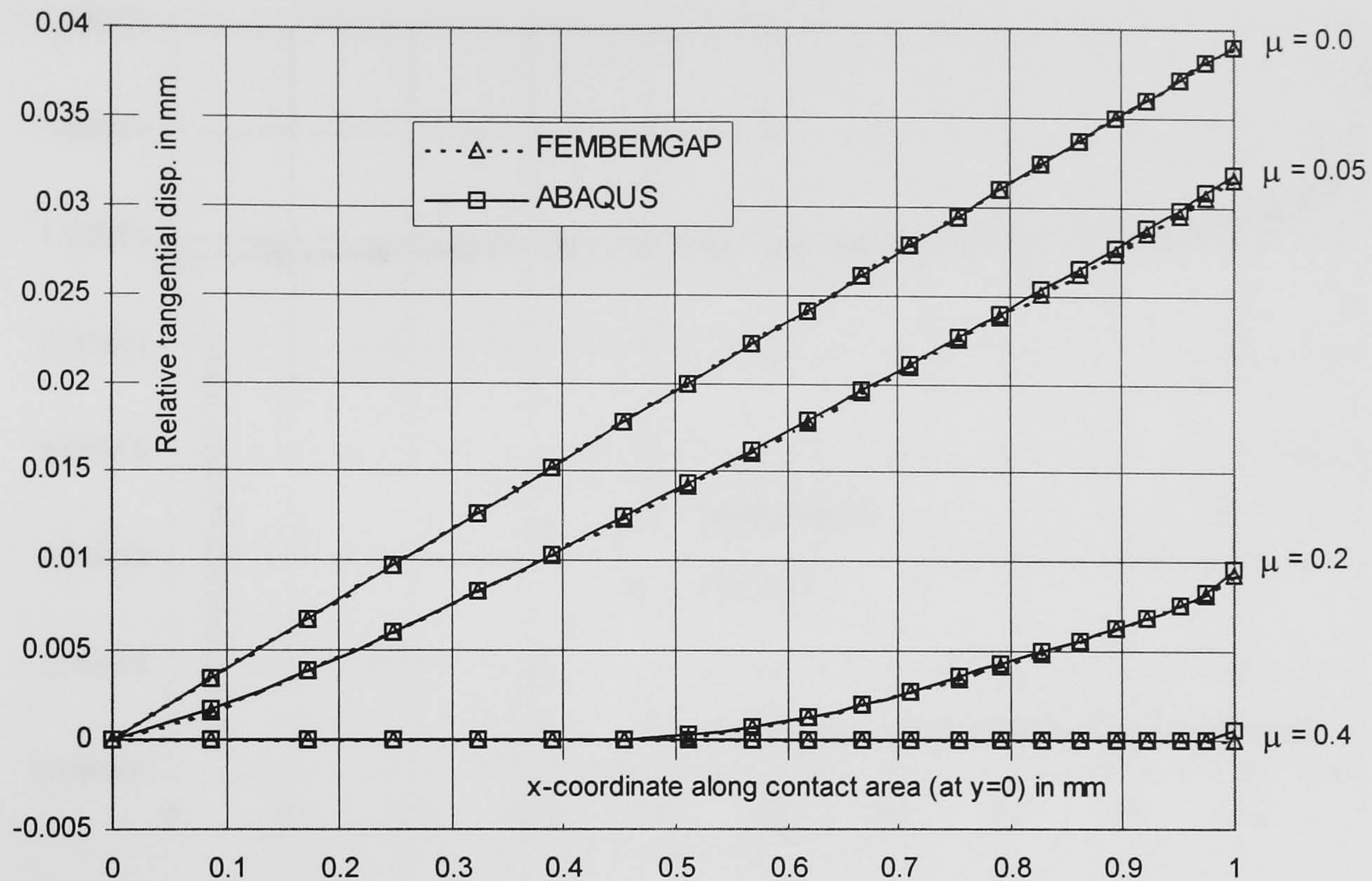


Figure 8.21 Relative tangential displacements of the contacting bodies of the elastic rectangular punch case along the contact area (at $y=0$ mm).

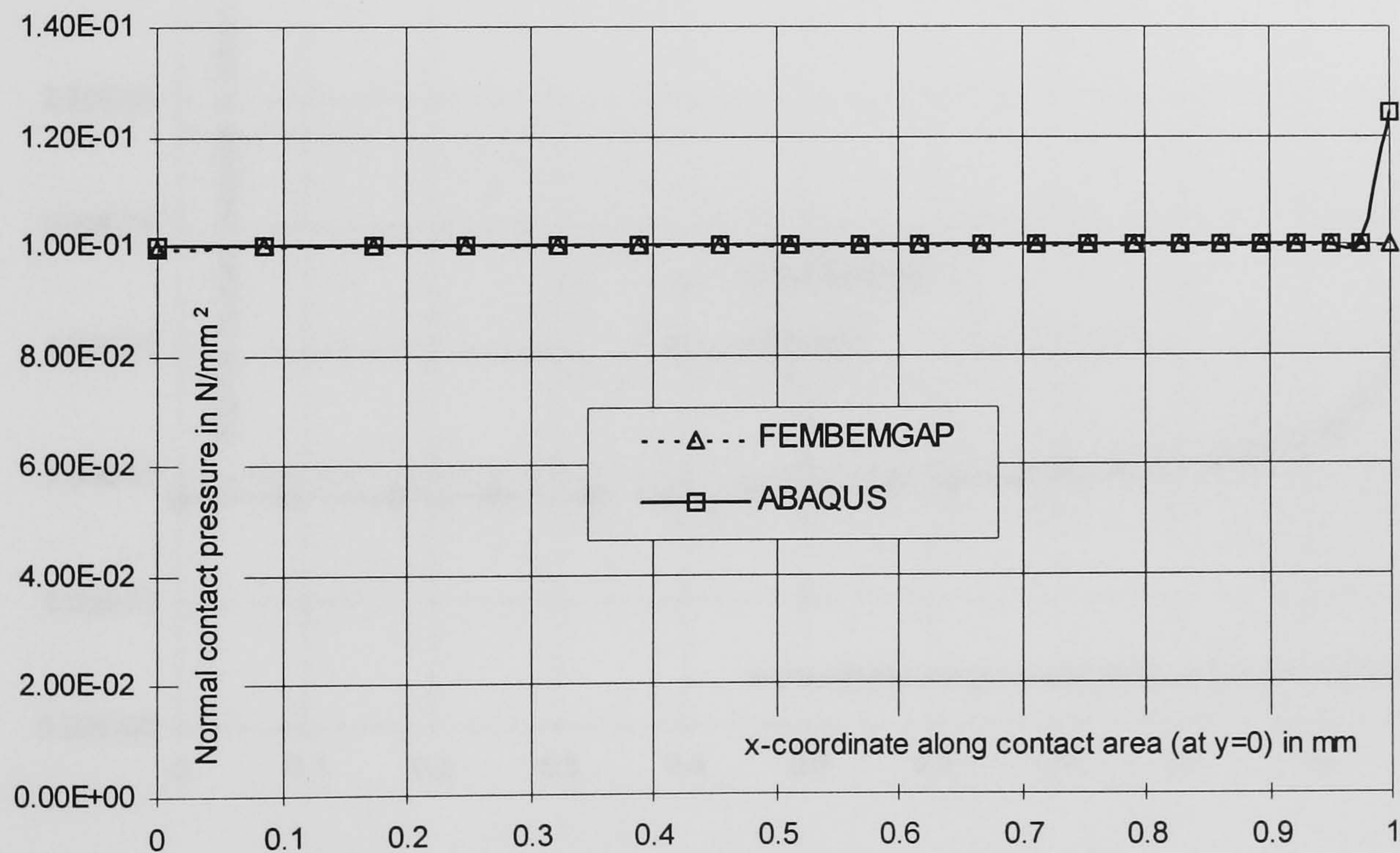


Figure 8.22 Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.0$.

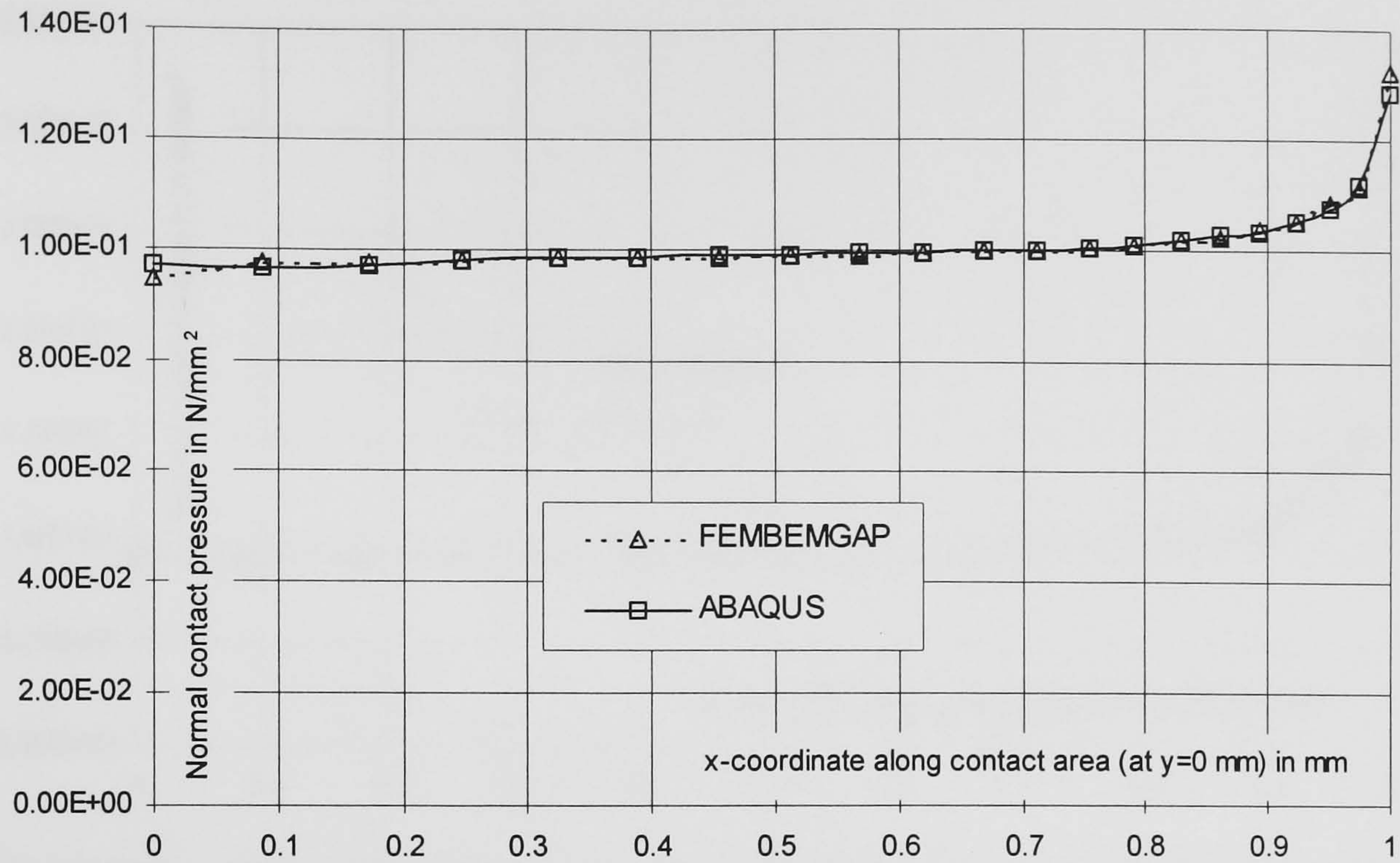


Figure 8.23 Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.05$.

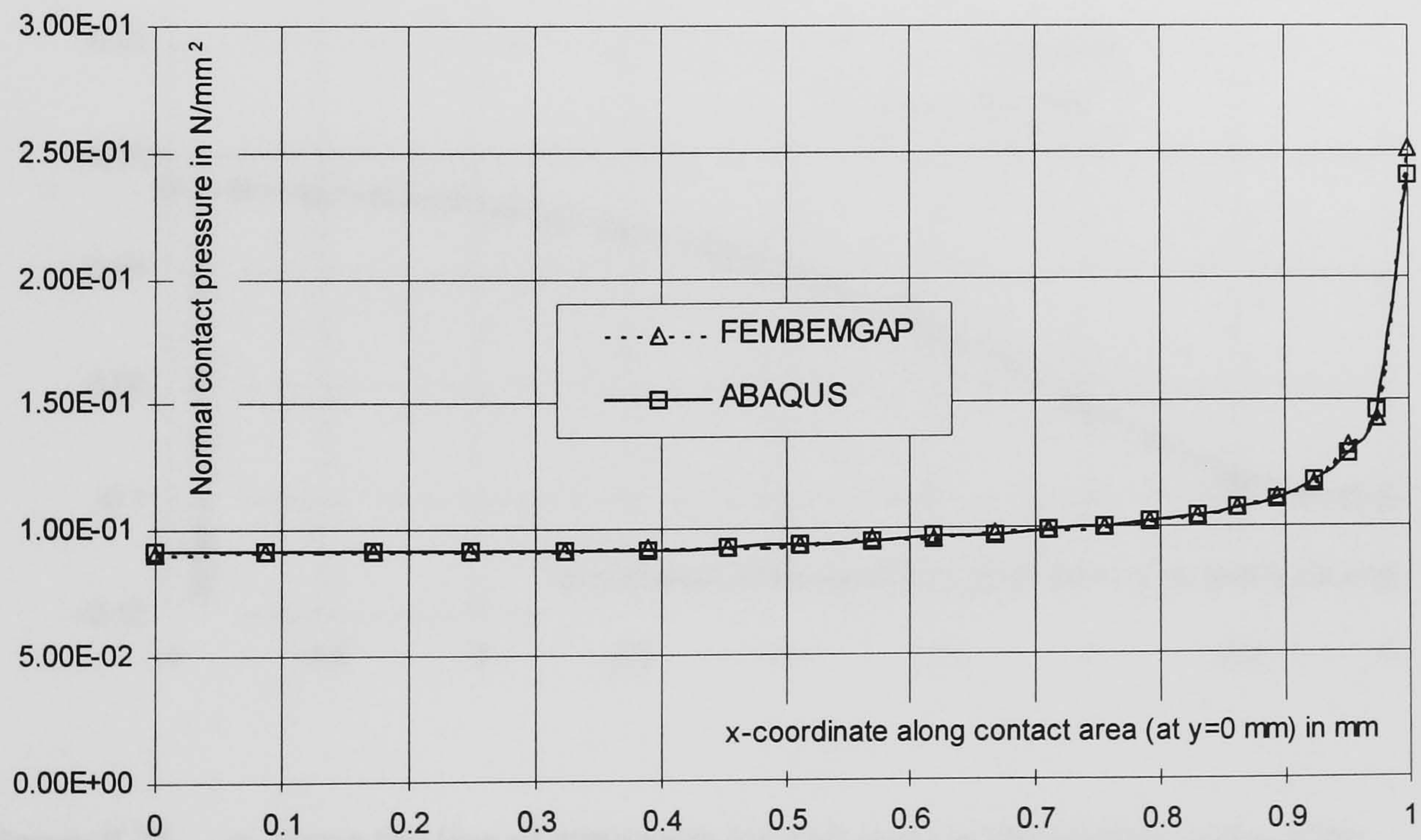


Figure 8.24 Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.2$.

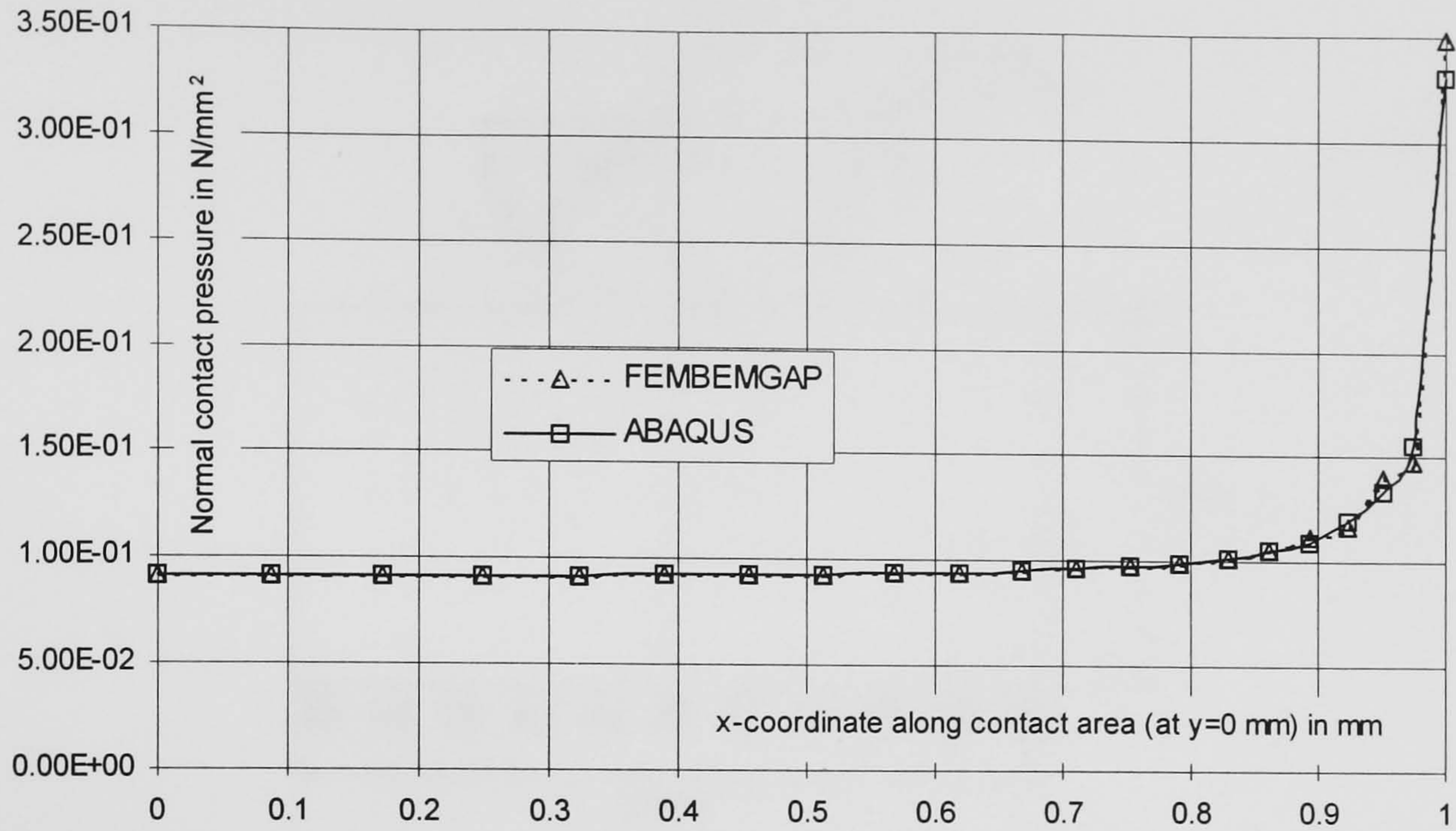


Figure 8.25 Normal contact pressure along the contact area of the rectangular punch case for $\mu=0.4$.

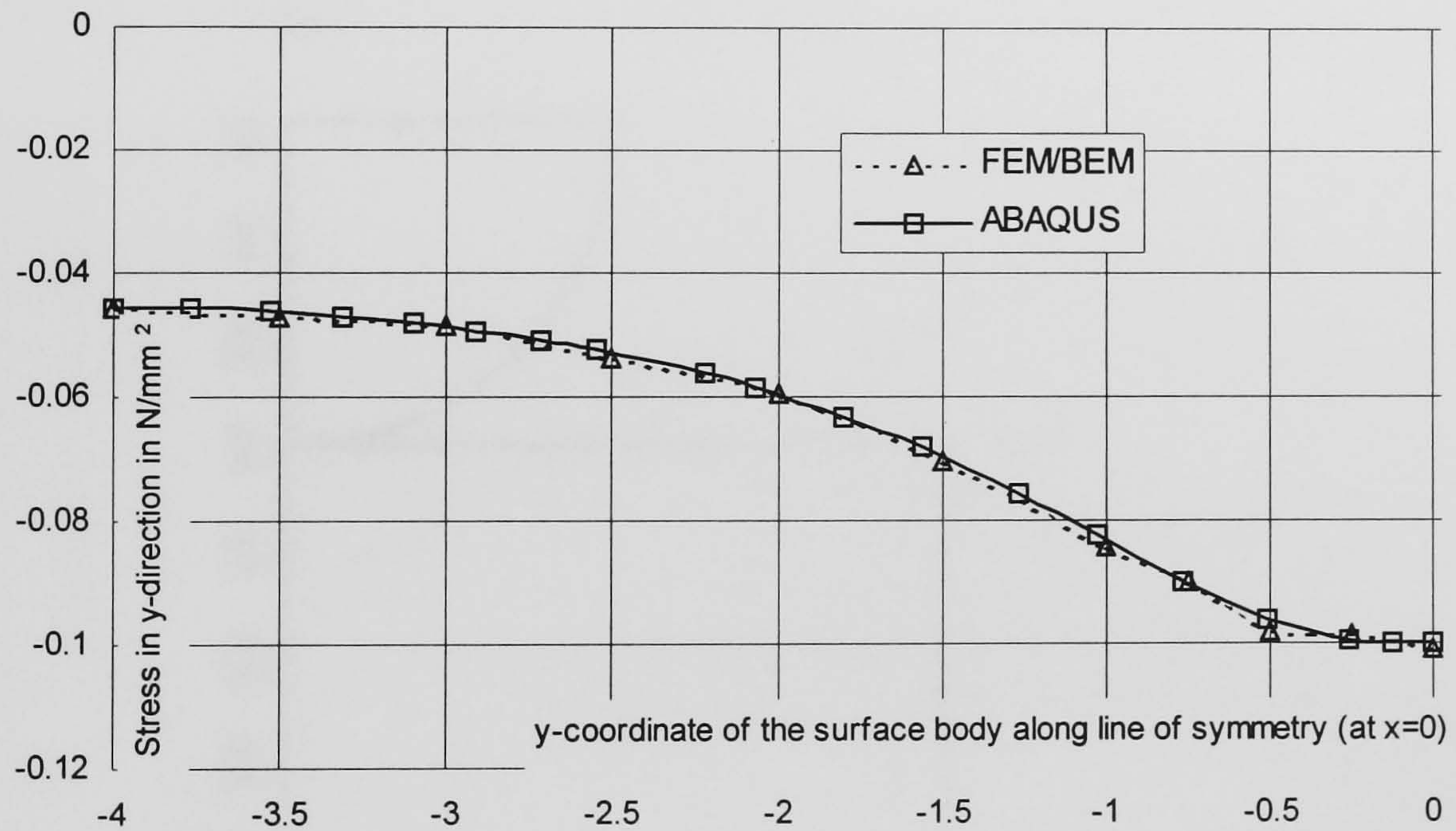


Figure 8.26 σ_y along the line of symmetry (at $x=0$ mm) of the surface body of the rectangular punch case for $\mu=0.0$.

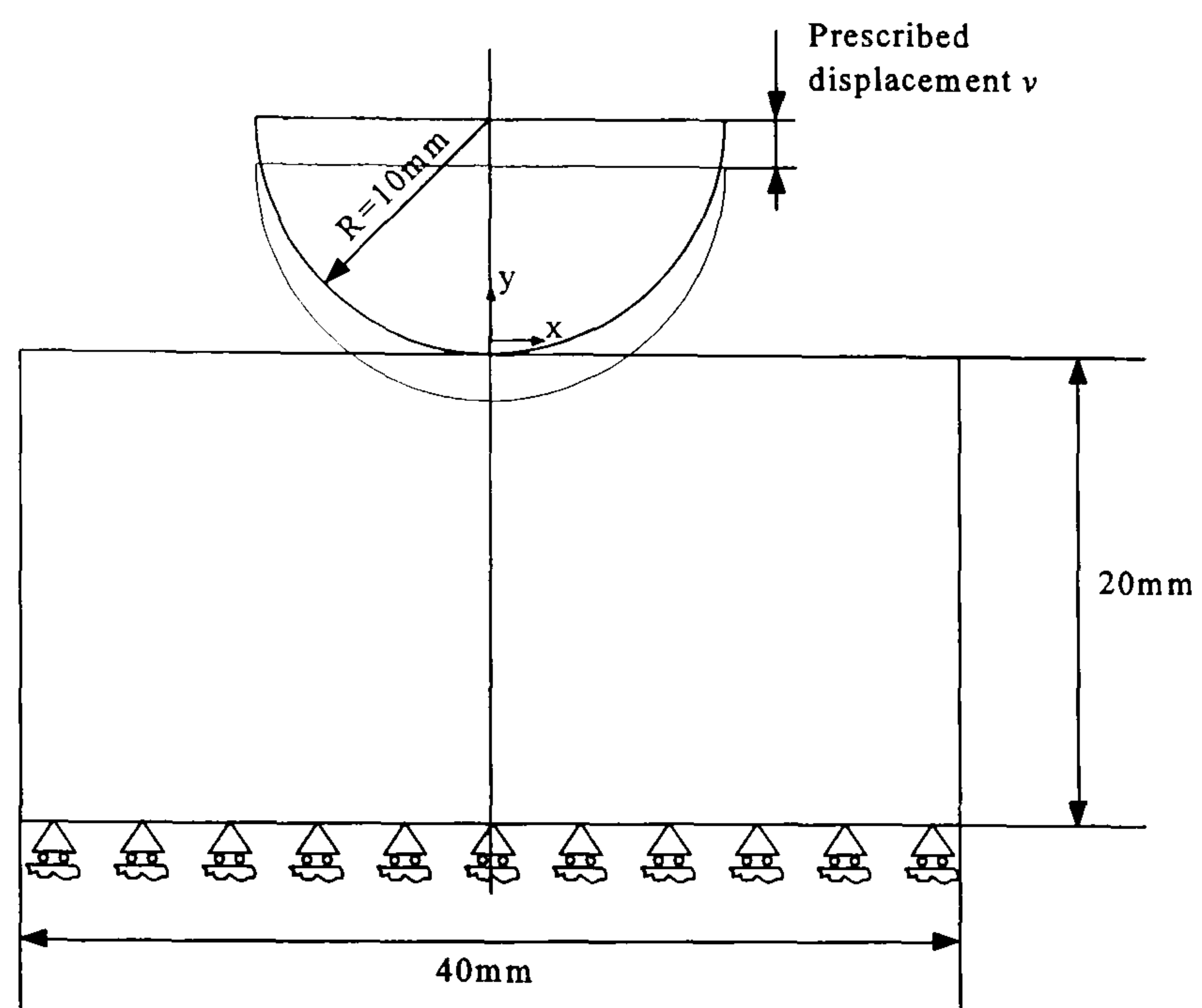


Figure 8.27 A cylindrical body is pressed onto a large rectangular body.

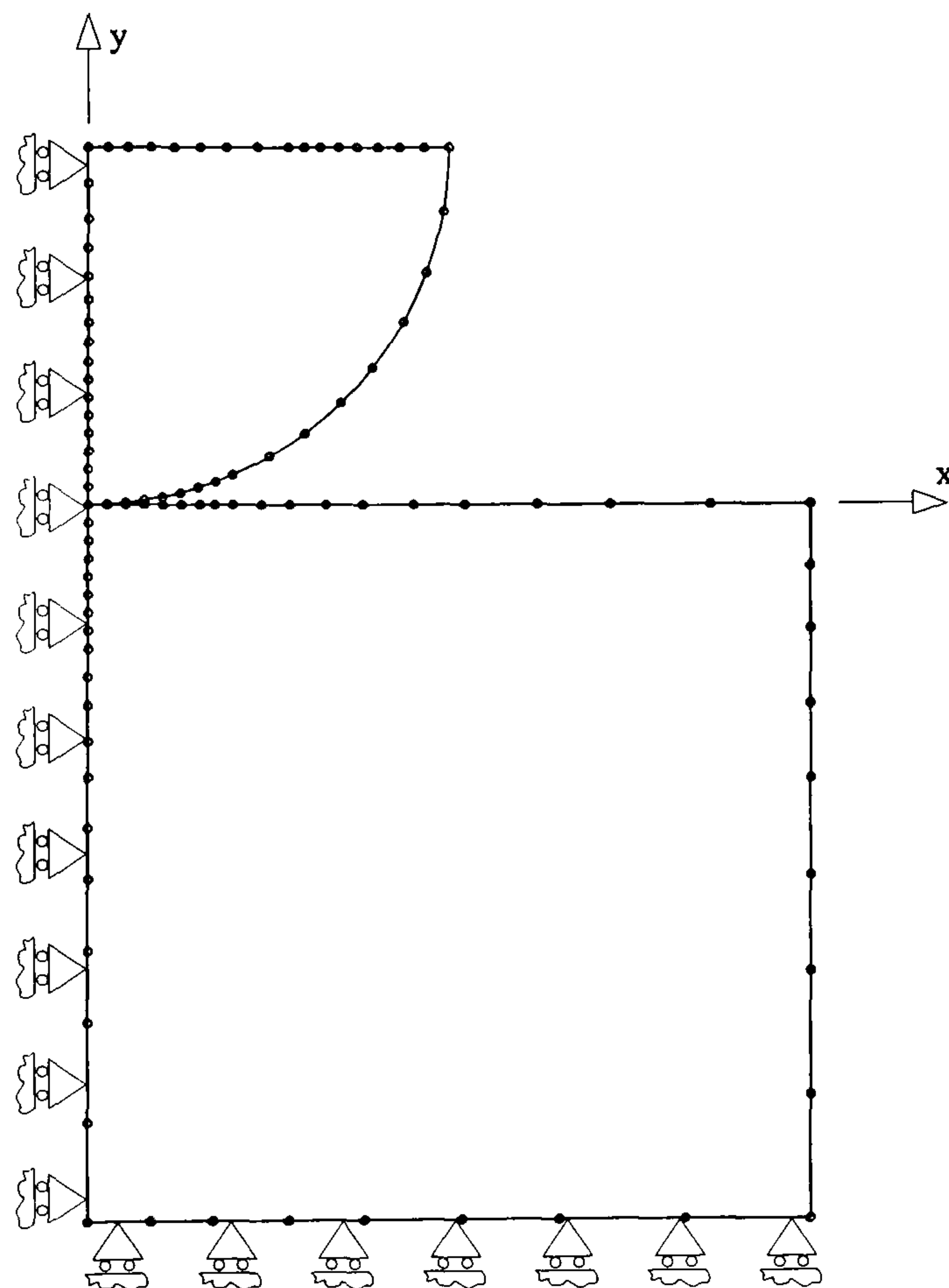


Figure 8.28 BEM model for the contact of the cylinder and surface.

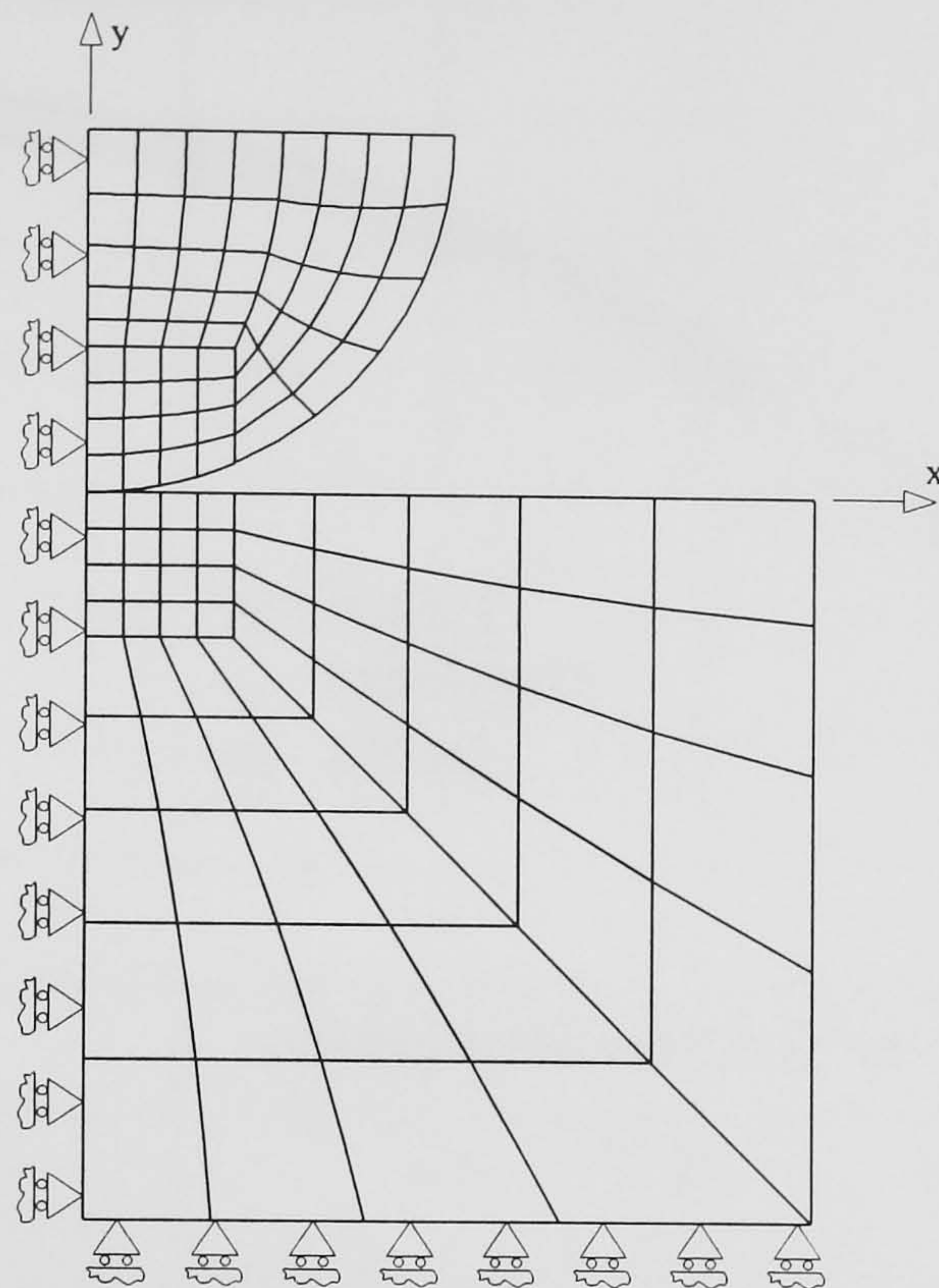


Figure 8.29 FEM model for the contact of the cylinder and surface.

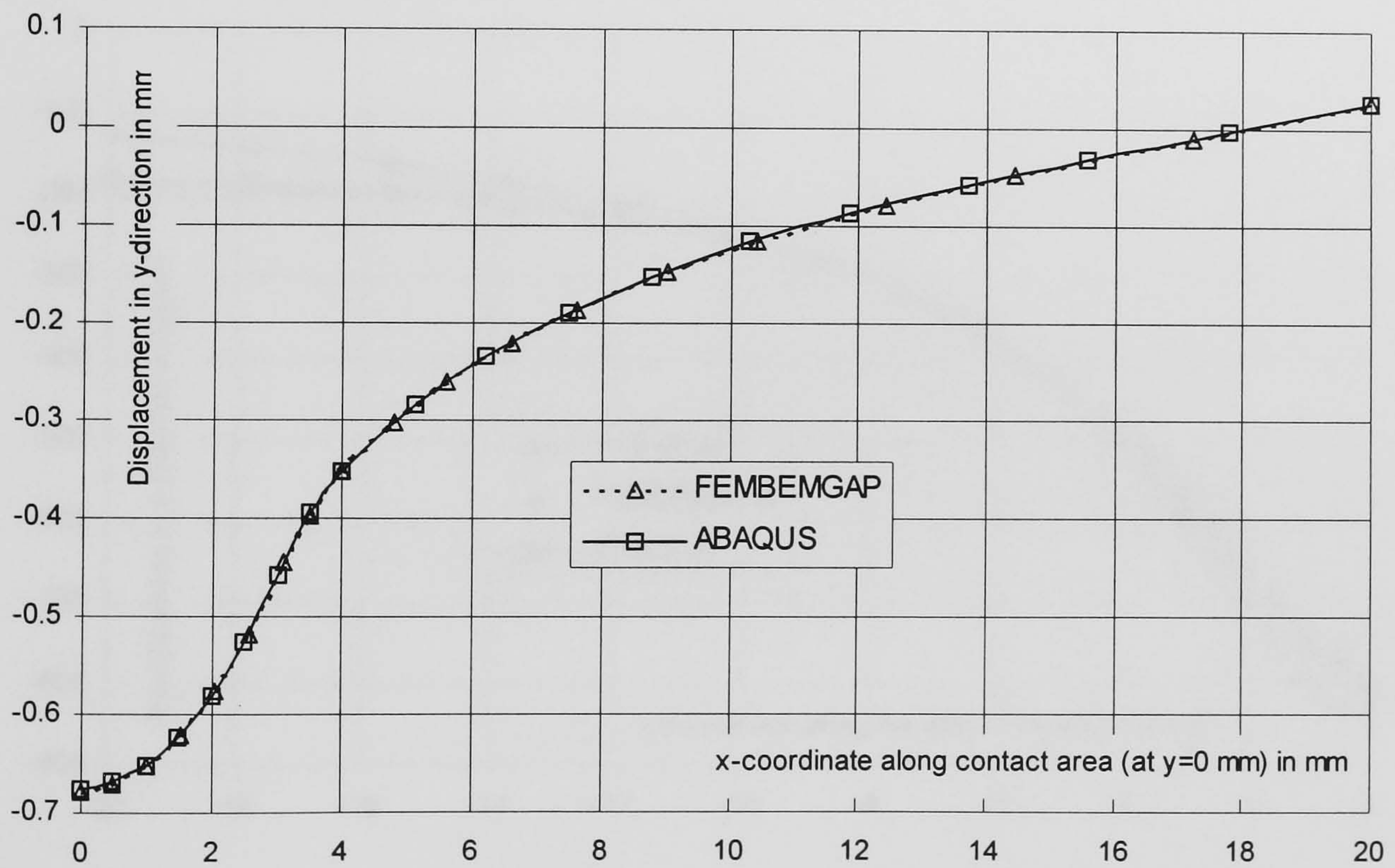


Figure 8.30 Displacement in y-direction of the surface body along the contact area (at $y=0$ mm) for the elastic cylinder/surface case.

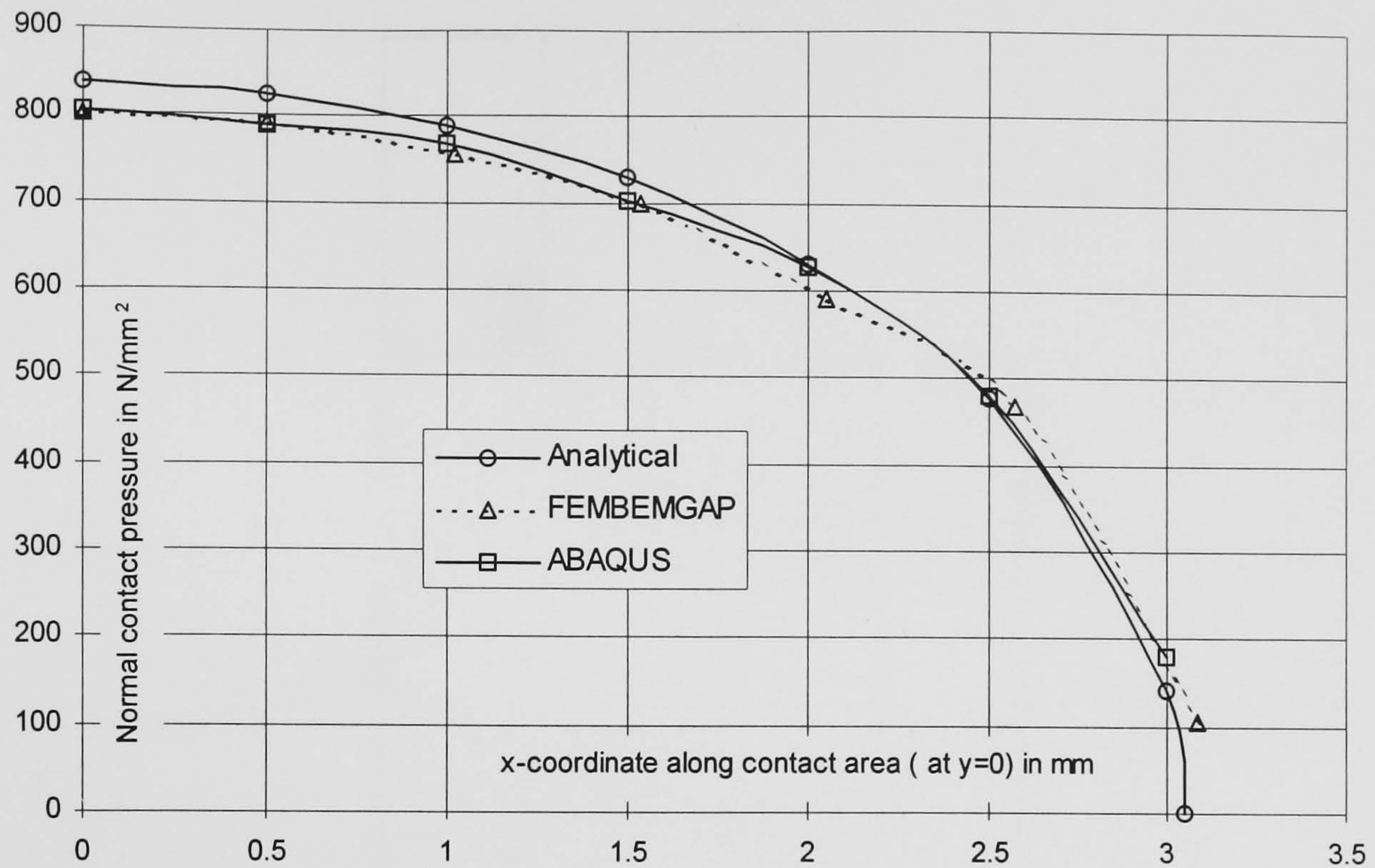


Figure 8.31 Normal contact pressure along the contact area (at $y=0$ mm) of the elastic cylinder/surface case.

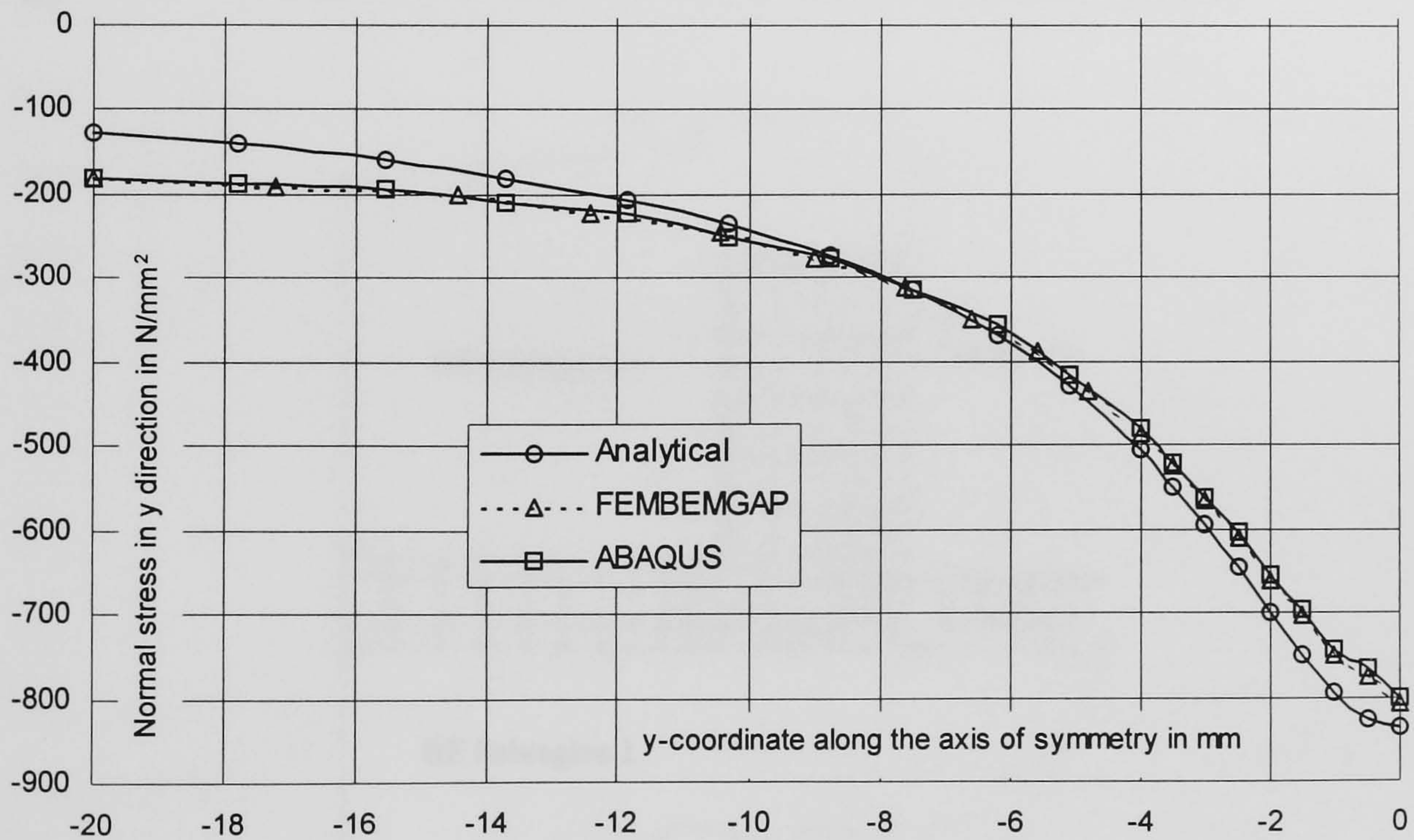


Figure 8.32 σ_y along the axis of symmetry (at $x=0$ mm) of the surface body for the elastic cylinder/surface case.

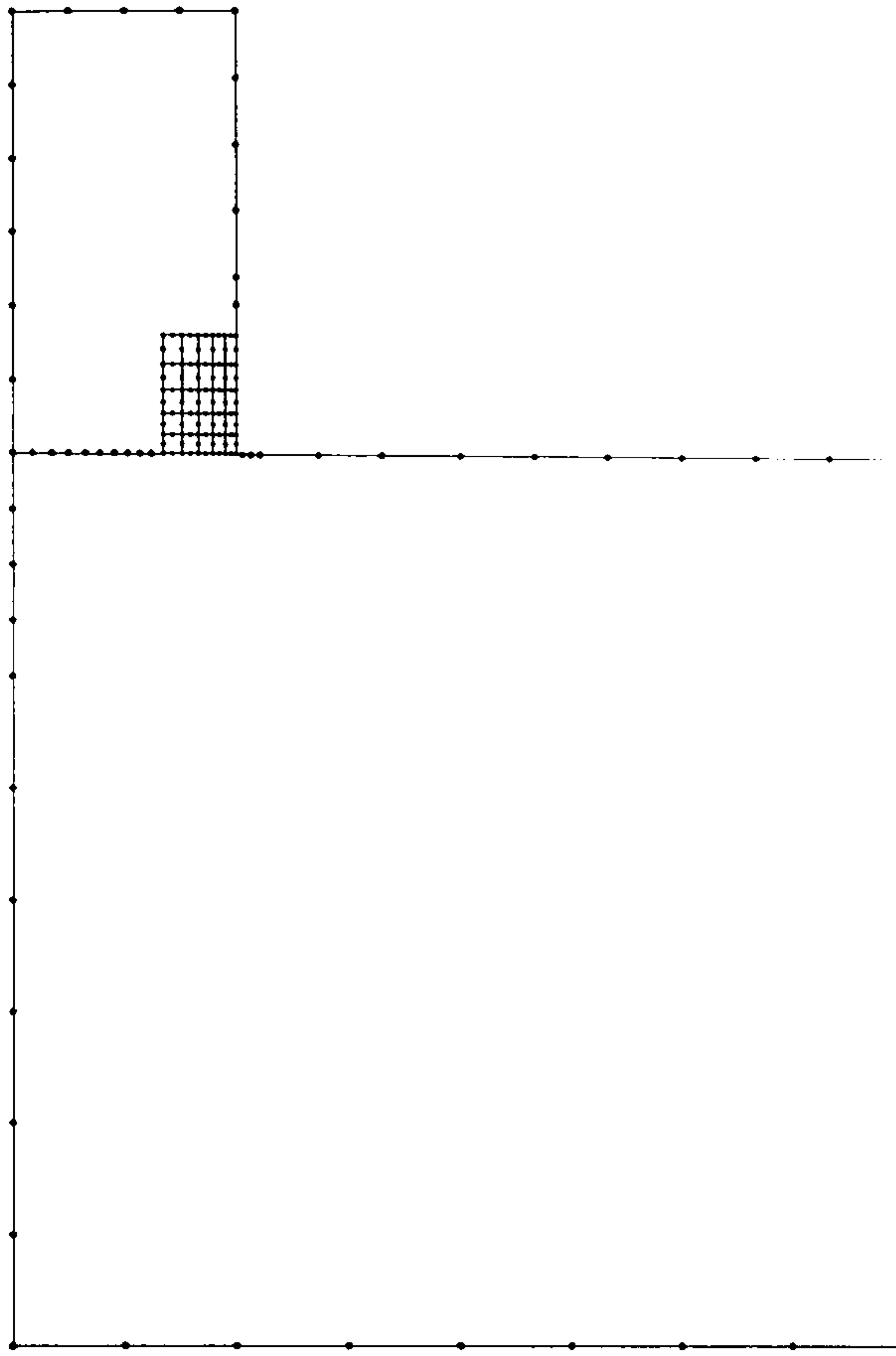


Figure 8.33 FEMBEMGAP model of the elastoplastic punch on a surface.

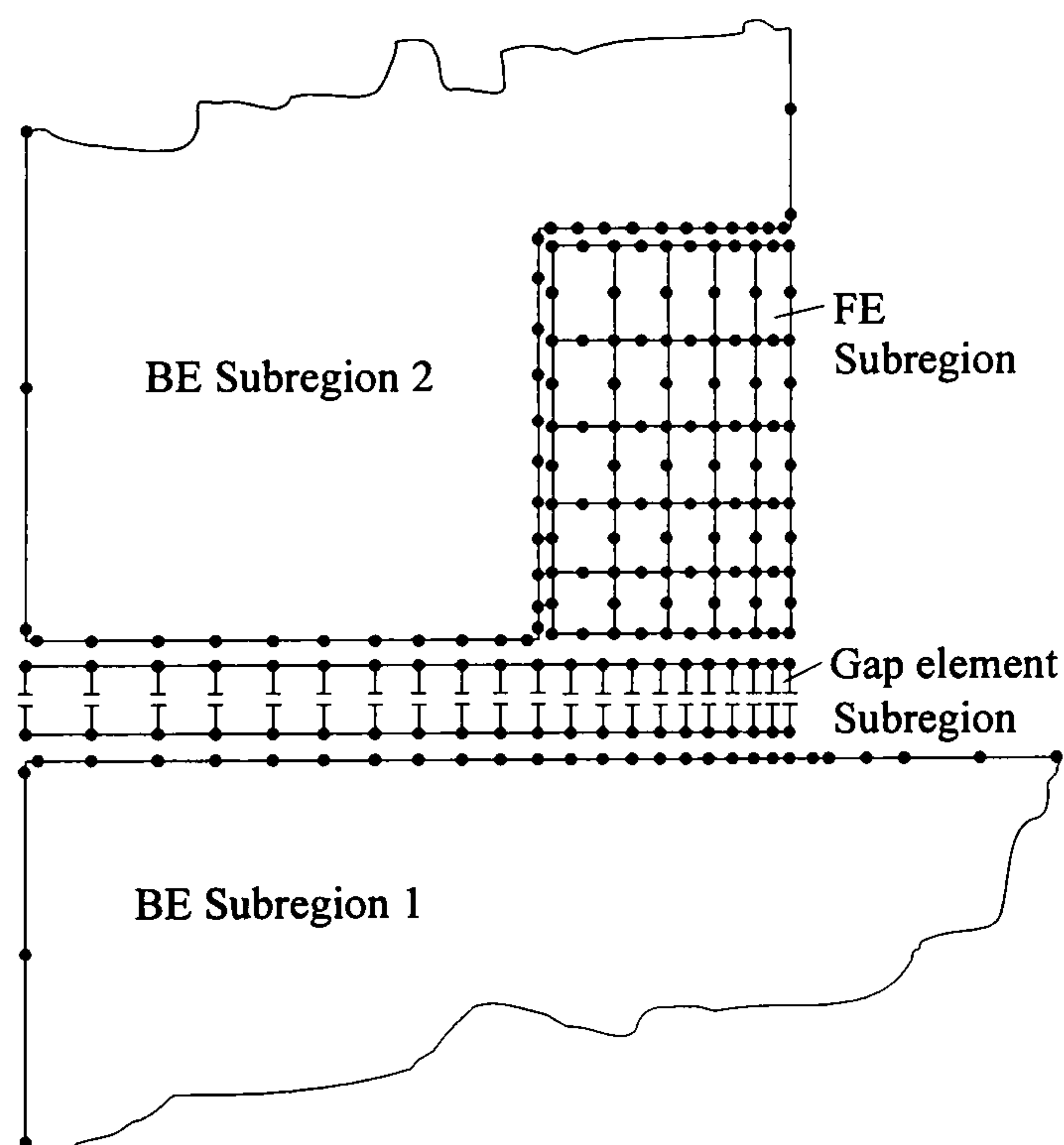


Figure 8.34 Different subregions defined in the FEMBEMGAP model of the elastoplastic punch.

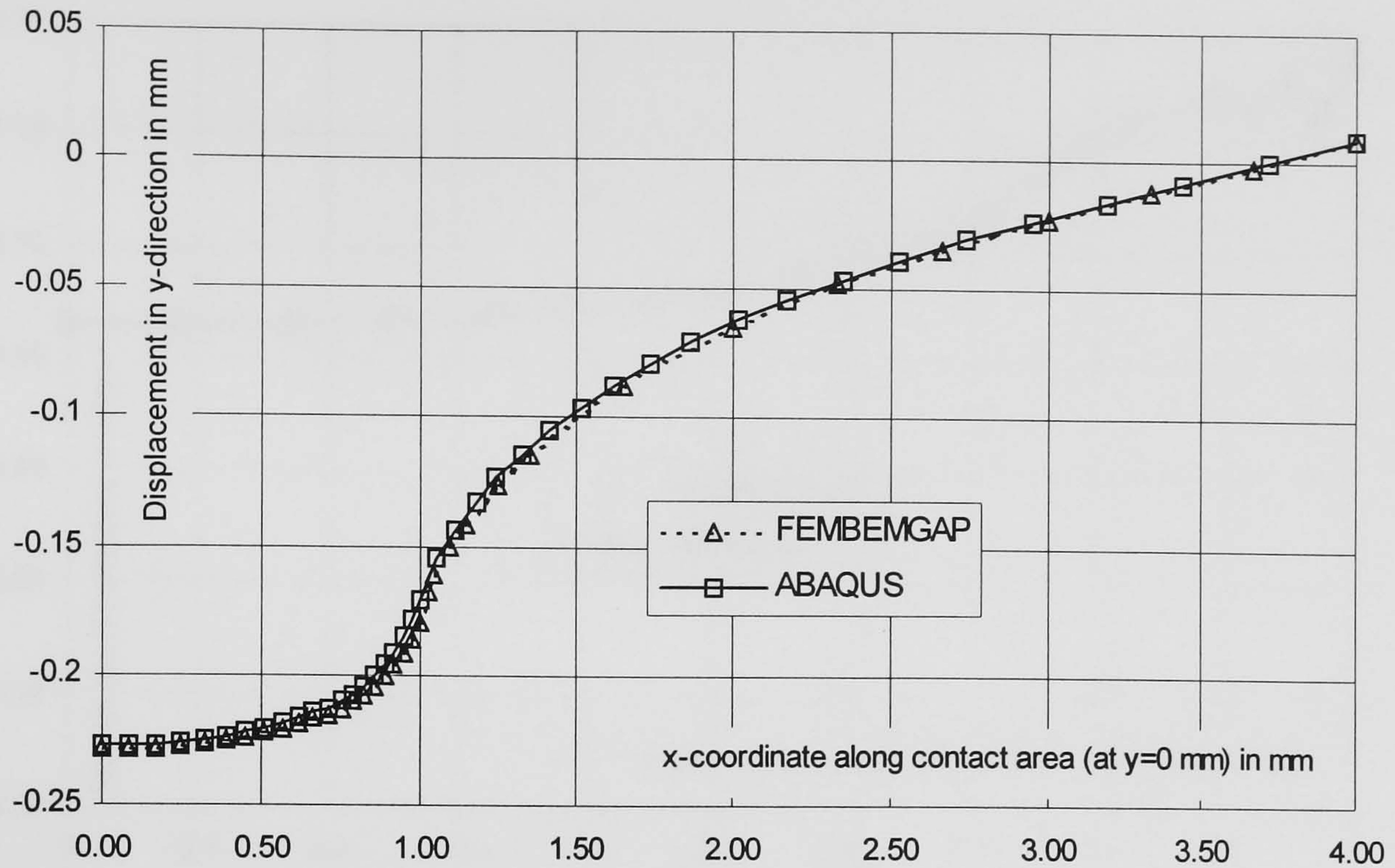


Figure 8.35 Displacement in y-direction along the contact area (at $y=0$ mm) of the surface body of the elastoplastic punch case.

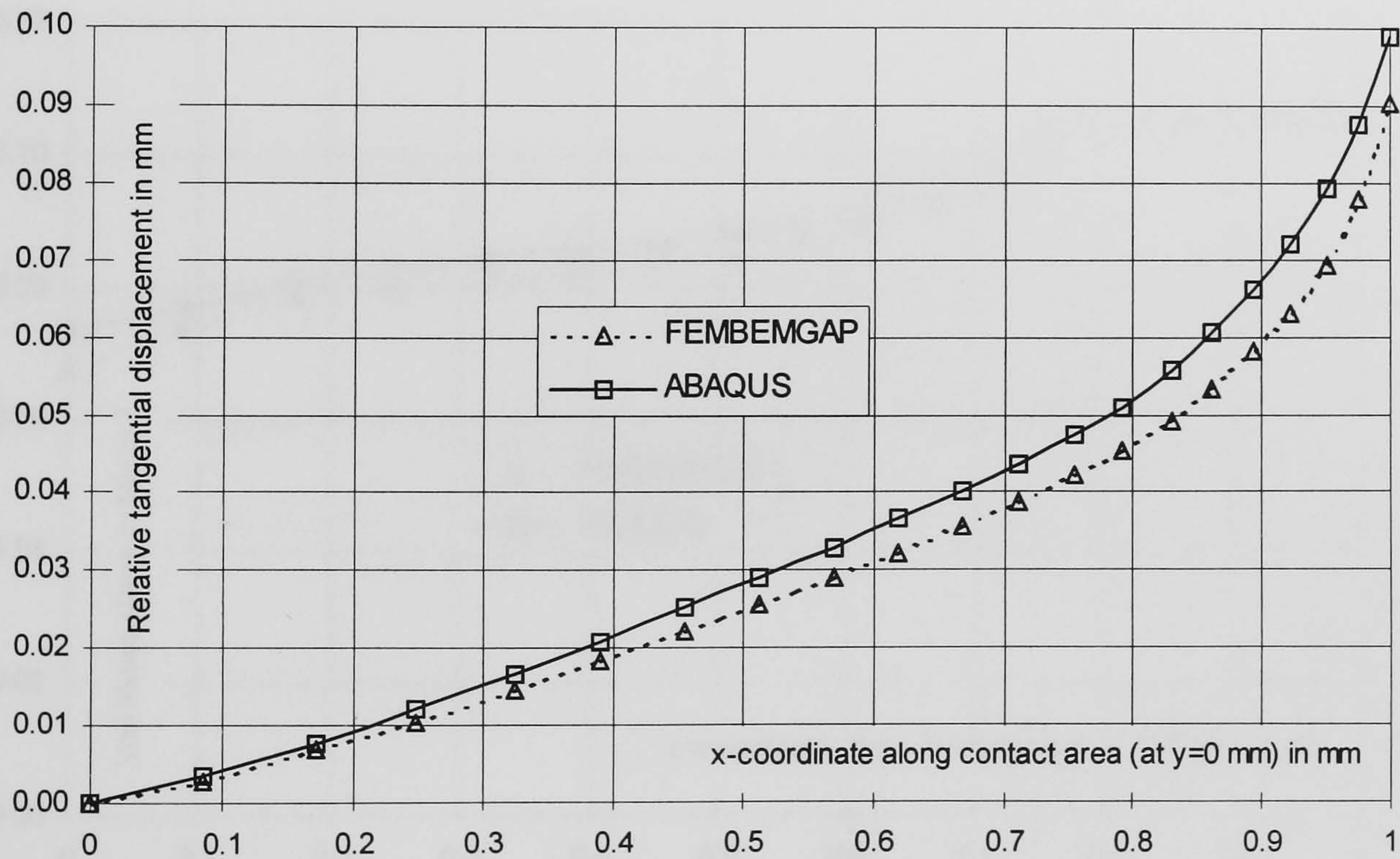


Figure 8.36 Relative tangential displacement of the two contacting bodies along the contact area (at $y=0$ mm) of the elastoplastic punch case.

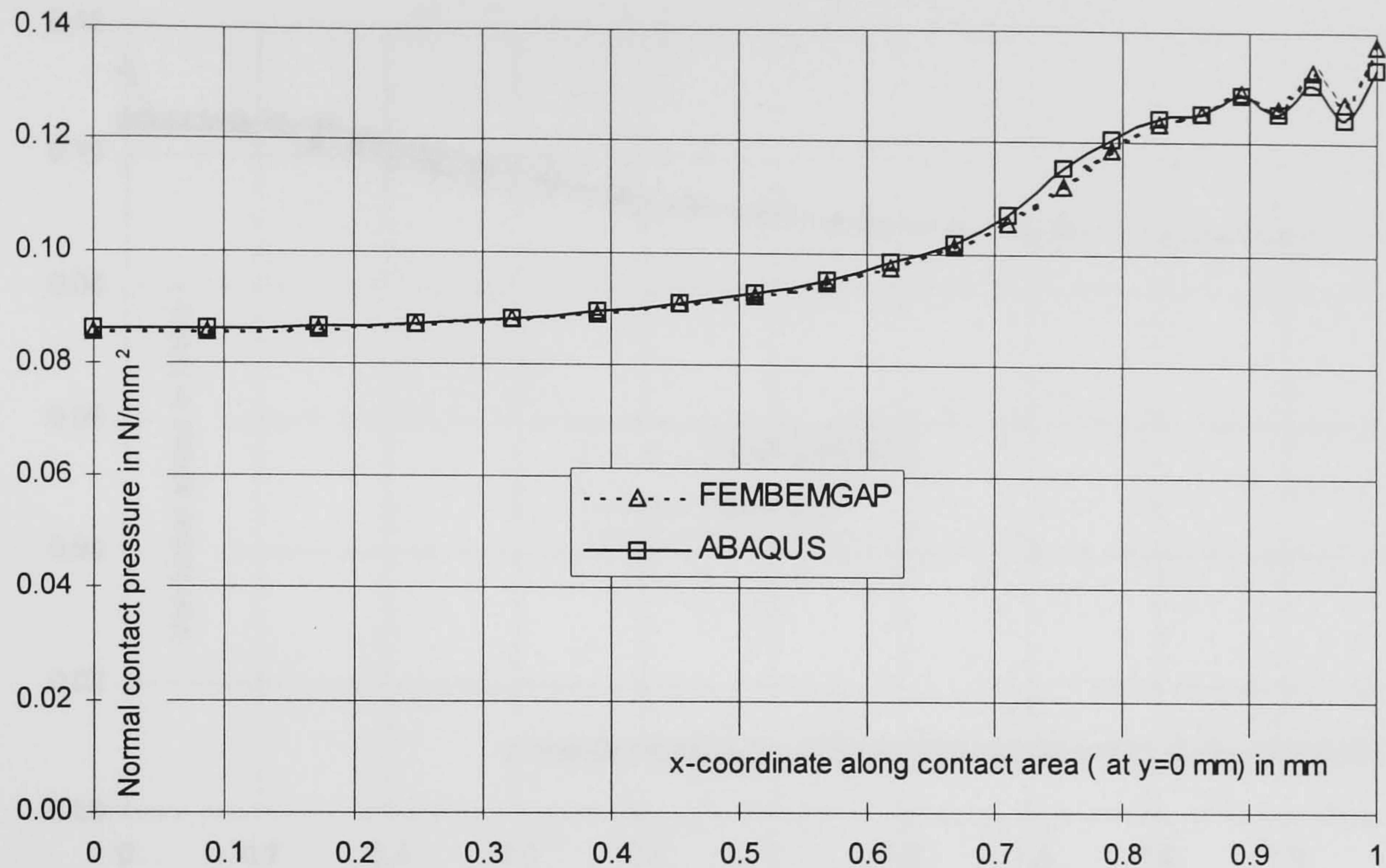


Figure 8.37 Normal contact pressure along the contact area (at $y=0$ mm) of the elastoplastic punch case.

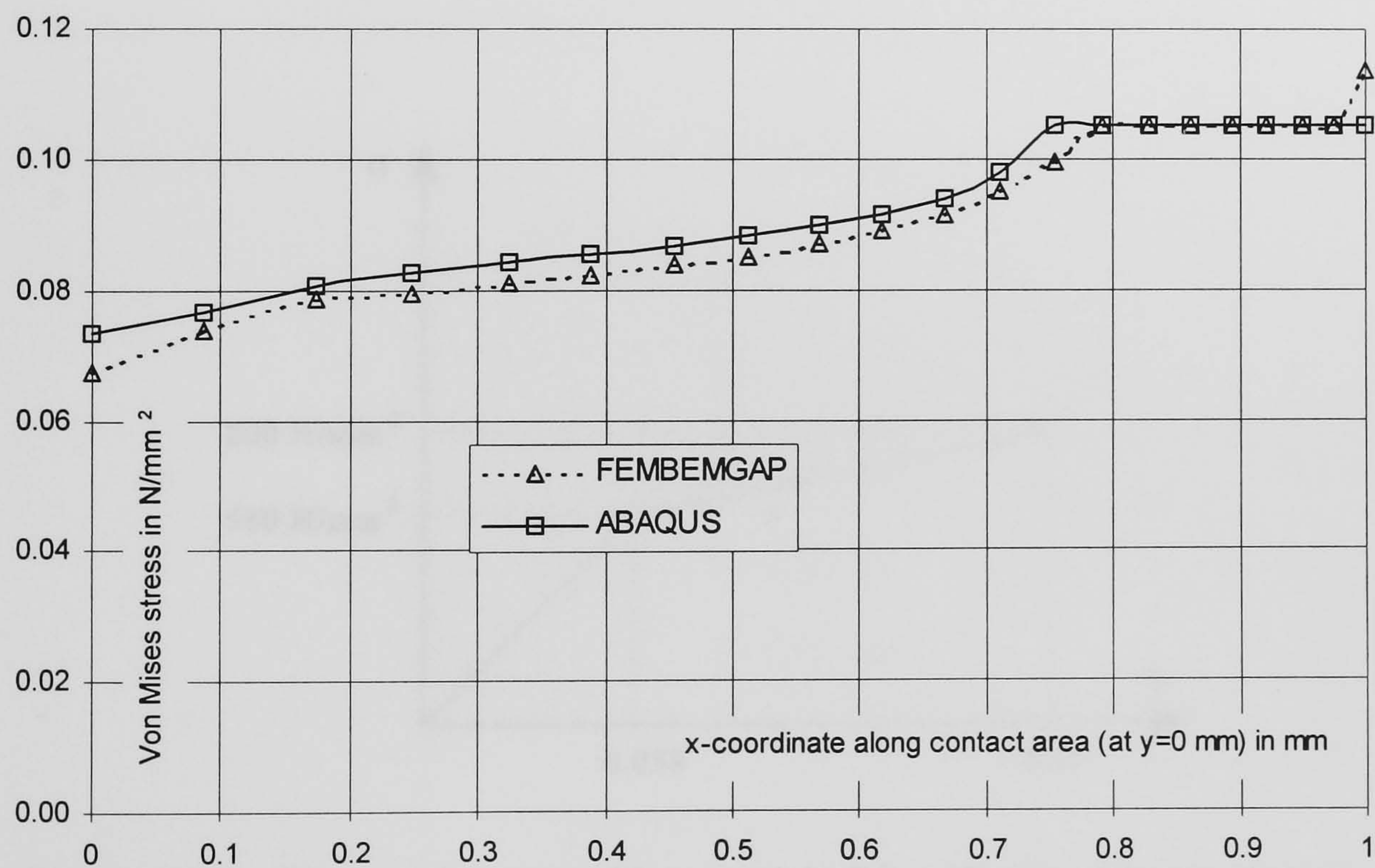


Figure 8.38 Von Mises stress along the contact area (at $y=0$ mm) of the punching body of the elastoplastic punch case.

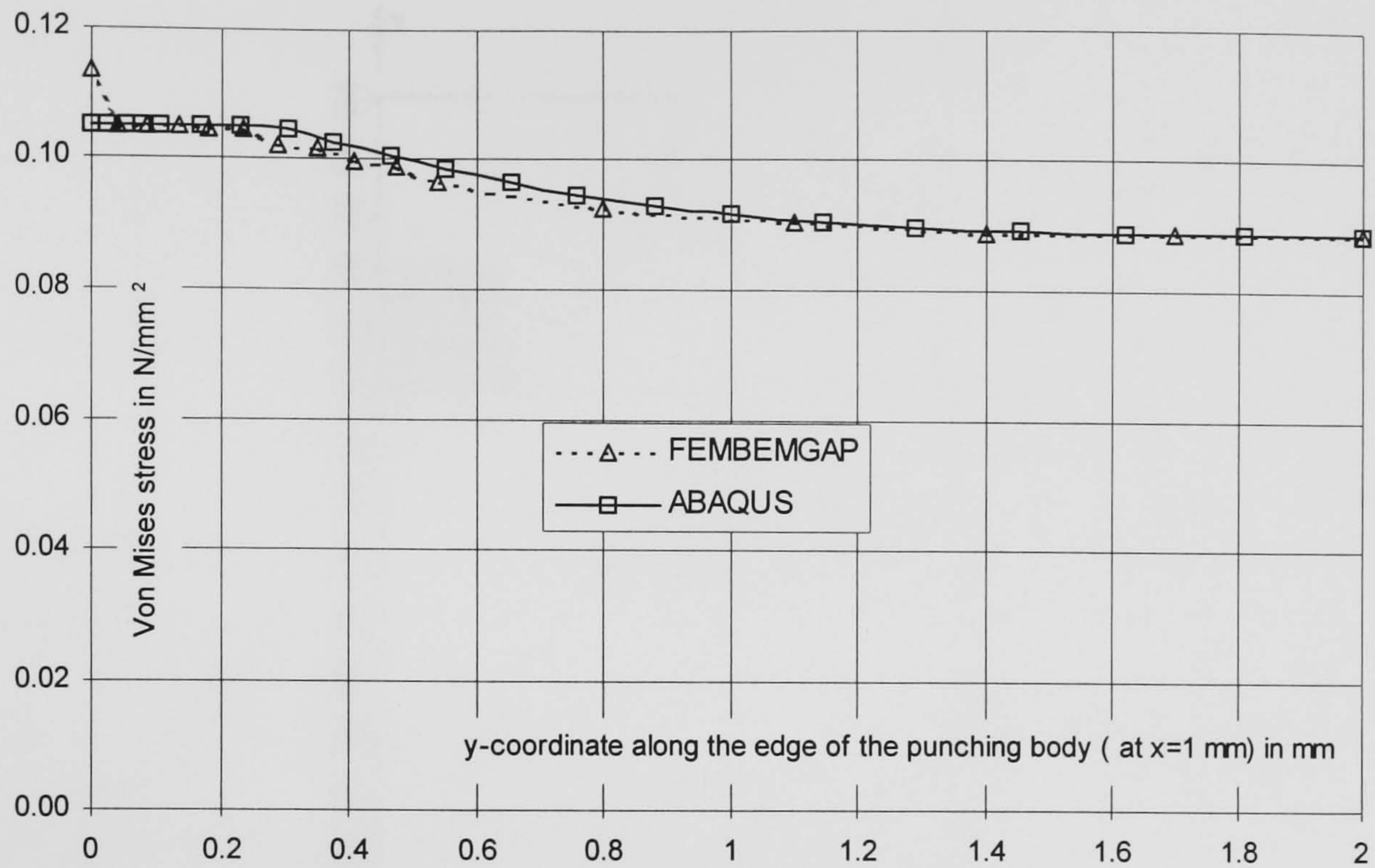


Figure 8.39 Von Mises stress along the right edge of the punching body (at $x=1$ mm) of the elastoplastic punch case.

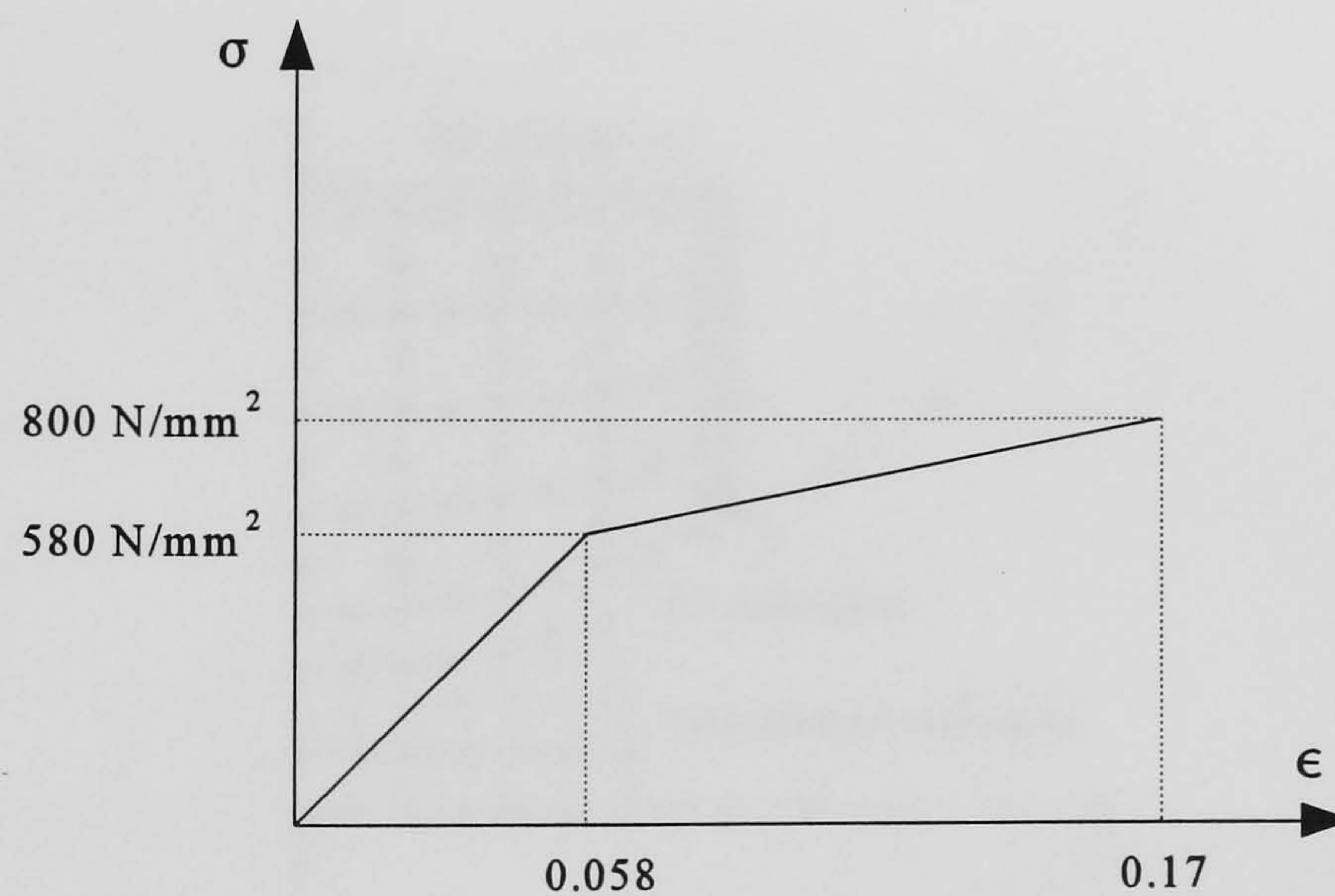


Figure 8.40 Stress-strain curve of the material of the cylinder of the elastoplastic cylinder/surface case.

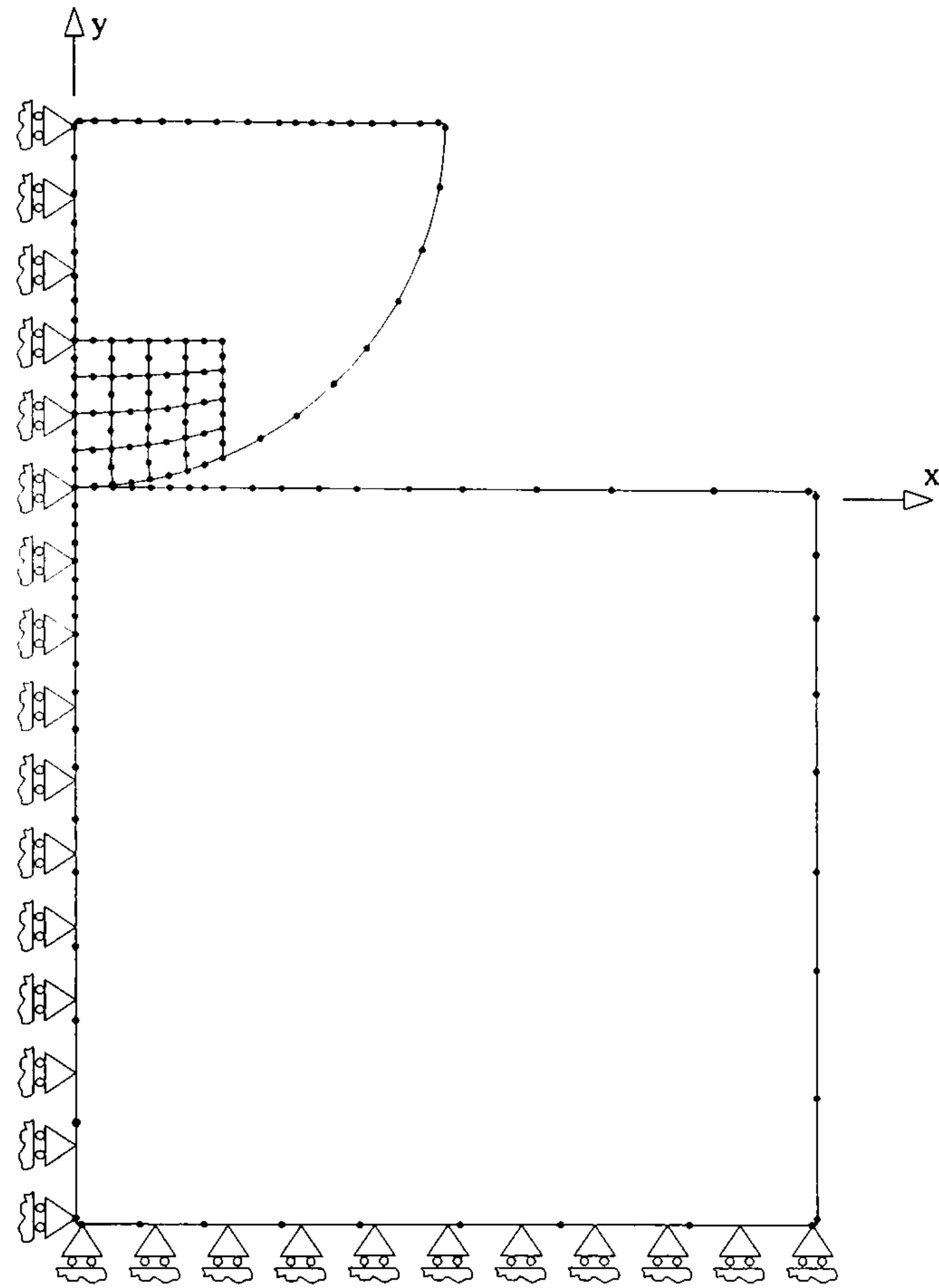


Figure 8.41 FEMBEMGAP model of the elastoplastic cylinder/surface case.

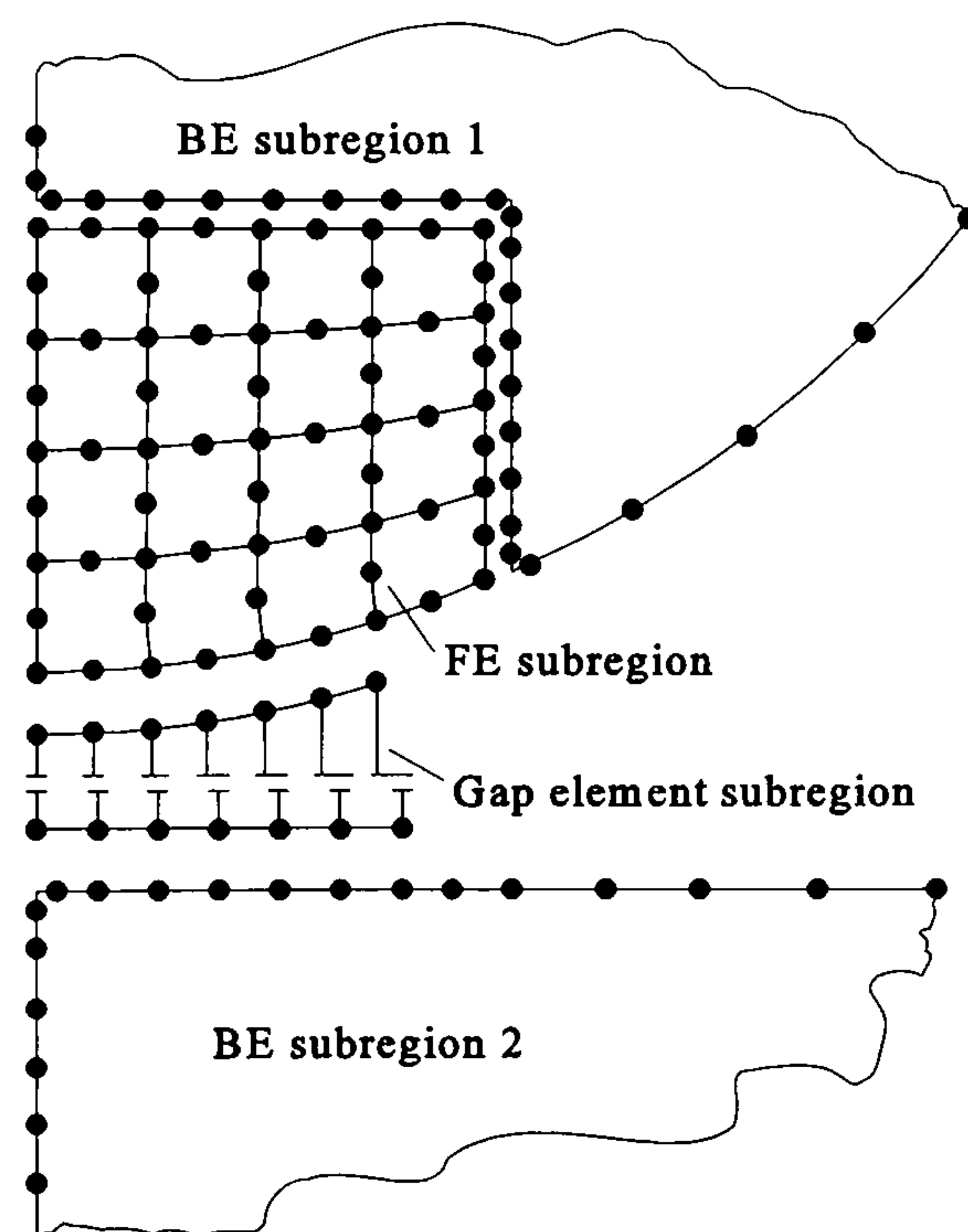


Figure 8.42 Different subregions defined in the FEMBEMGAP model of the cylinder/surface case.

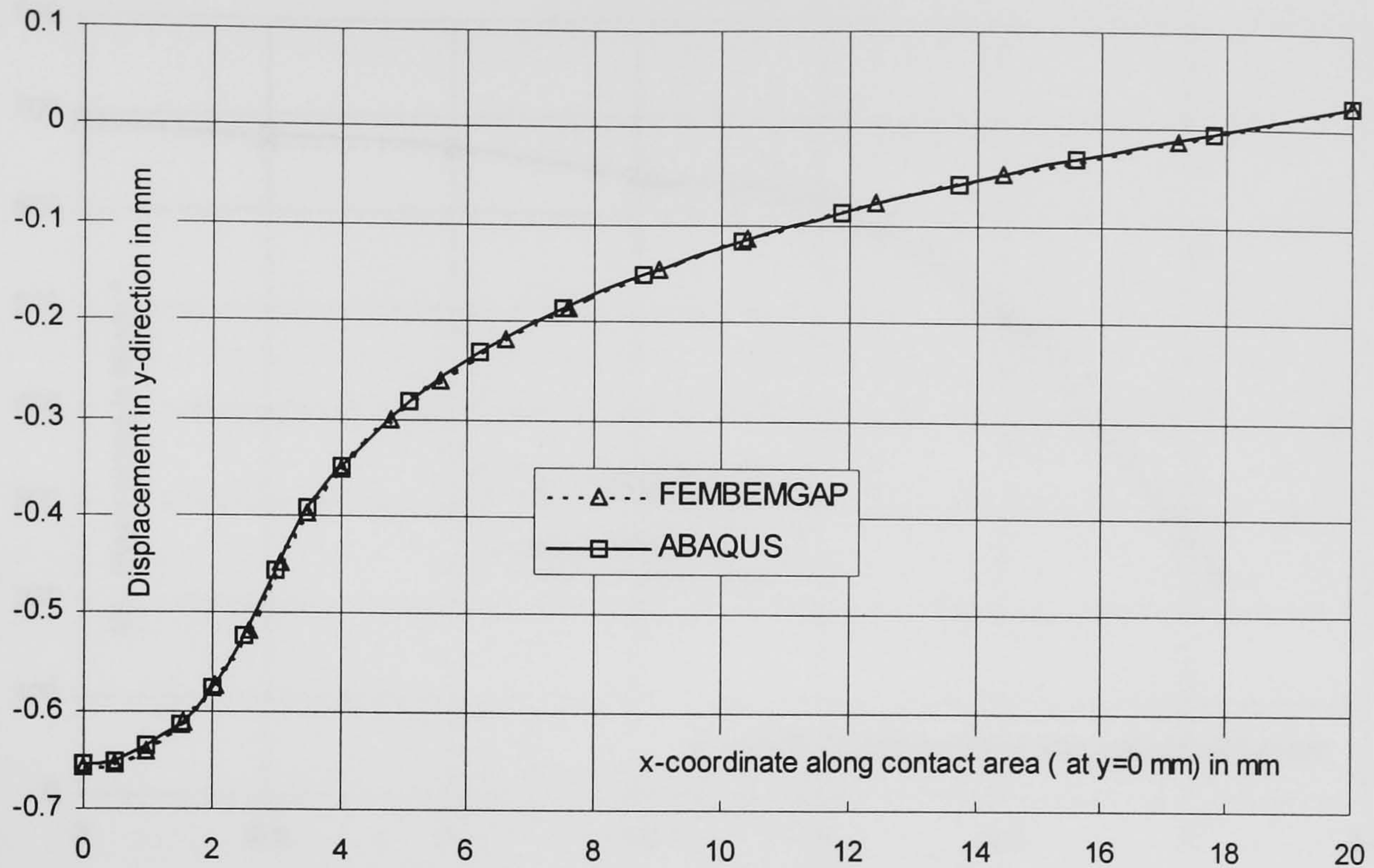


Figure 8.43 Displacement in y-direction of the upper edge of the surface body of the elastoplastic cylinder/surface case.

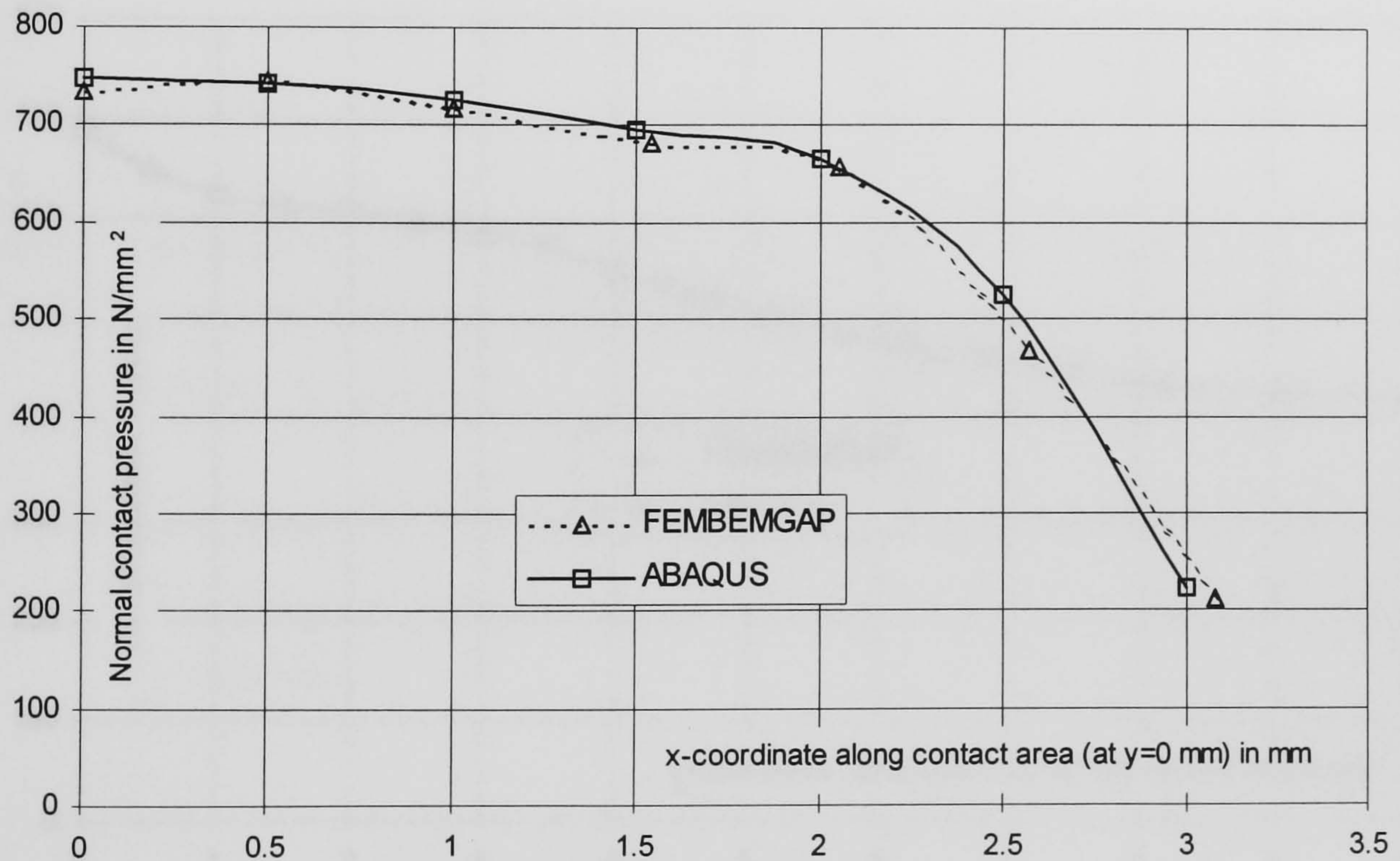


Figure 8.44 Normal contact pressure along the contact area (at $y=0$ mm) of the elastoplastic cylinder/surface case.

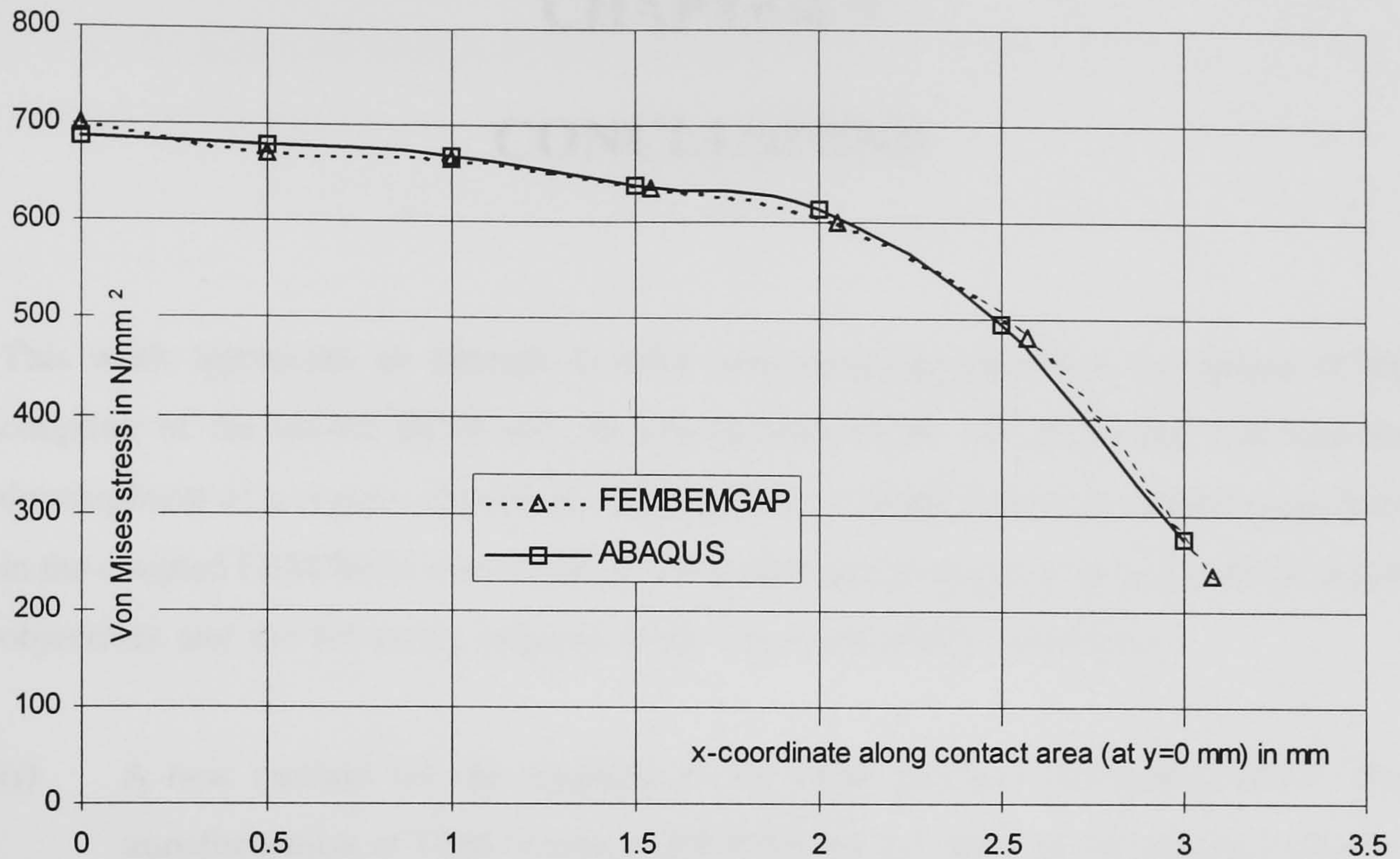


Figure 8.45 Von Mises stress along the contact area of the cylindrical body of the elastoplastic cylinder/surface case.

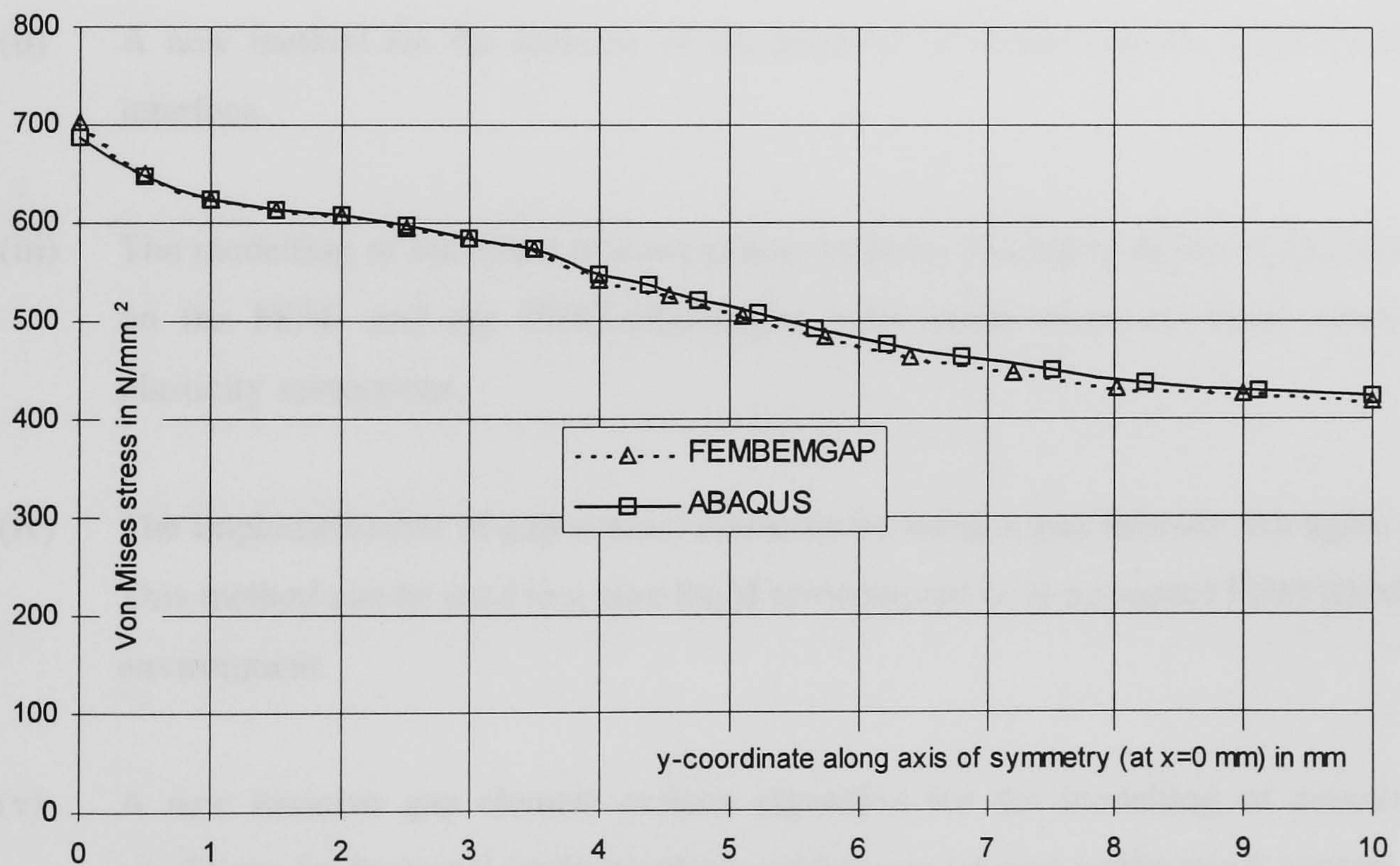


Figure 8.46 Von Mises stress along the axis of symmetry (at x=0 mm) of the cylindrical body of the elastoplastic cylinder/surface case.

CHAPTER 9

CONCLUSIONS

This work represents an attempt to solve elastoplasticity problems by means of the coupling of the elastic BEM and the elastoplastic FEM. An additional task was the development of a contact algorithm which is able to model frictional contact conditions in the coupled FEM/BEM environment. The author has managed to achieve all the major objectives and the following original work was successfully developed:

- (i) A new method for the coupling of the FEM and the conforming BEM. The transformation of FEM format to BEM format is limited to the interface degrees of freedom and the coupling procedure allows for the modelling of body forces in FEM and BEM subregions.
- (ii) A new method for the solution of the problem of corners on the FEM/BEM interface.
- (iii) The modelling of the effect of elastoplastic material conditions in the FEM zone on the FEM- and the BEM-subdomains with initial stress or initial strain plasticity corrections.
- (iv) The implementation of gap contact elements by using a gap element subregion. This method can be used in a pure BEM environment or in a coupled FEM/BEM environment.
- (v) A new iterative gap element contact algorithm for the modelling of contact conditions for frictional (or frictionless) conforming (or non-conforming) contact. The algorithm can be applied for elastic and elastoplastic analysis.

A package based on the presented theory was developed in FORTRAN 77 and tested on a PC. A number of patch tests and case studies were run and compared to analytical solutions or the commercial finite element package ABAQUS in order to validate the developments. Considering the results of these case studies the following conclusions can be drawn:

- (i) The FEM/BEM package works very successfully and there are no noticeable accuracy losses caused by the coupling procedure.
- (ii) The problems arising from interface corners could be solved. The results at external and internal interface corners are very accurate, although there are slight local result deviations near an internal interface corner. However these deviations are in an acceptable range and do not interfere with results in the other parts of the body.
- (iii) The elastoplastic application of the coupled FEM/BEM is also very successful since very good accuracy is given in the elastic BEM subregions and the elastoplastic FEM subregions.
- (v) The modelling of the contact of an FEM surface and a BEM surface is a very difficult task because the unknown contact forces are of a different nature at the contacting surfaces. In this context the gap element subregion was found to be a simple and versatile method and it is straight forward to implement compared to other known contact methods.
- (vi) The inclusion of a gap element subregion does not introduce any noticeable inaccuracies. The results of all elastic and elastoplastic contact case studies were found to be very good, which shows that the implemented contact/friction algorithm works in a stable and accurate manner.

It has already been concluded in the literature review that the FEM is superior to the BEM in elastoplastic problems as far as ease of implementation, computational

procedures and CPU time are concerned. Hence the most popular and most commonly used numerical method for elastoplastic applications is the FEM. A comparison between the pure FEM and the developed coupled FEM/BEM method for elastoplastic applications leads to the following conclusions:

- (i) The coupled FEM/BEM is by far superior to the pure FEM as far as modelling effort is concerned. Great savings in pre-processing times could be achieved, especially for problems with plastic areas being small compared to the size of the bodies.
- (ii) The coupled FEM/BEM plasticity procedure can supply the same reliability and numerical stability as the established pure FEM.
- (iii) In elastoplasticity problems the coupled FEM/BEM is as accurate as the pure FEM and it was found that the stress evaluation in the elastic BEM zones is even slightly more accurate.

Recommendations for future work:

- (i) An investigation into the CPU time associated with the solution of the resulting coupled system of equations should be undertaken. In this context the following measures might be found to decrease the CPU time:
 - * A more efficient solver than the inverse matrix solver. The solver should account for the occurring zero blocks and the symmetric finite and gap element contributions.
 - * The use of a symmetrised BEM formulation and a symmetric matrix solver (Although the effect of a symmetrised BEM formulation on the result accuracy has to be carefully investigated especially with respect to the incremental and iterative character of the developed package).

- (ii) An investigation into different ways of modelling corners on the FEM/BEM interface. Although acceptable results have been achieved with the presented method a possibility for modelling interface corners without any loss of accuracy should be found.
- (iii) The gap element algorithm could be changed such that it allows for surface contact modelling rather than node-to-node contact which would prevent the overlap which occurs in the elastoplastic punch case.
- (iv) The developed package can be applied to fracture mechanics problems where the small plastic zone at the crack tip could be modelled with the FEM and the rest of the structure with the BEM.

REFERENCES AND BIBLIOGRAPHY

Authors, Year	Title	Source
Abascal, R. Dominguez, J. 1993	Analysis of the frictional slip between a layer and a half space	Boundary Element 15, pp. 209-219, Computational Mechanics Publications, Southampton, England, 1993
Al-Edani, A.N. 1990	Efficient fracture mechanics programming system for linear and non-linear problems using finite element and boundary element methods.	PhD thesis, Cranfield University, Cranfield, GB, 1990.
Bai, X. Zhao, X. 1988	Analysis of large deformation elastoplastic contact through finite gap elements	Computers & Structures Vol. 30, No. 4, pp.975-978, 1988
Banerjee, P.K. Butterfield, R. 1981	Boundary element methods in engineering science	Mc Graw-Hill, London, GB, 1981
Bathe, K.J. Chaudhary, A. 1985	A solution method for planar and axisymmetric contact problems	International Journal for Numerical Methods in Engineering Vol. 21, pp. 65-88, 1985
Beaudoin, A.J. Dawson, P.R. 1995	A BEM/FEM framework for simulation of a plastic workpiece deformed by an elastic tool	Journal of Engineering for Industry -Transactions of the ASME Vol. 117, pp.372-377, 1995
Becker, A.A. 1992	The boundary element method in engineering	Mc Graw-Hill, London, GB, 1992
Beer, G. 1986	Implementation of combined boundary element -finite element analysis with applications in geomechanics	Developments in Boundary Element Methods-4, Banerjee, P.K. (edt.), pp.191-225, Elsevier, London, 1986

Authors, Year	Title	Source
Beer, G. 1983a	Finite element, boundary element and coupled analysis of unbounded problems in elastostatics	International Journal for Numerical Methods in Engineering Vol. 19, pp.567-580, 1983
Beer, G. 1983b	BEFE- a combined boundary element finite element computer program	Adv. Eng. Software Vol. 6, No. 2, pp. 103-109, 1983
Beer, G. Meek, J.L. 1981	The coupling of boundary and finite element methods for infinite domain problems	Boundary Element Methods, Brebbia, C.,A. (edt), pp. 575-591, Springer Verlag, Berlin, 1981
Belytschko, T. Lu, Y.Y. 1991	Singular integration in variationally coupled FE-BE method	Journal of Engineering Mechanics Vol. 117, No.4, pp.820-835, 1991
Belytschko, T. Lu, Y.Y. 1994	A variationally coupled FE-BE method for transient problems	International Journal for Numerical Methods in Engineering Vol. 37, pp. 91-105, 1994
Belytschko, T. Lu, Y.Y. Chang, H.S. 1989	A variationally coupled finite element - boundary element method	Computers & Structures Vol. 33, No.1, pp.17-20, 1989
Bettes, P. 1981	Operations count for boundary integral and finite element methods	International Journal for Numerical Methods in Engineering Vol. 17, pp.306-308, 1981
Bossavit, A. 1991	Mixed methods and the marriage between finite elements and boundary elements	Numerical Methods for Partial Differential Equations Vol. 7, No. 4, pp.347-362, 1991
Brebbia, C.A. 1978	The boundary element method for engineers	Pentech Press ltd., Plymouth, GB, 1978

Authors, Year	Title	Source
Brebbia, C.A. Georgiou, P. 1979	Combination of boundary and finite elements in elastostatics	Applied Mathematical Modelling Vol. 3, pp. 212-220, 1979
Brink, U. Klaas, O. Niekamp, R. Stein, E. 1995	Coupling of adaptively refined dual mixed finite elements and boundary elements in linear elasticity	Advances in Engineering Software Vol. 24, pp. 13-16, 1995
Carstensen, C. Zarrabi, D. Stephan, E.P. 1996	On the h-adaptive coupling of FE and BE for viscoplastic and elasto-plastic interface problems	Journal of Computational and Applied Mathematics Vol.75, pp. 345-363, 1996
Carstensen, C. 1996	Coupling of FEM and BEM for interface problems in viscoplasticity and plasticity with hardening	SIAM, J. numer. anal. Vol. 33, No. 1, pp. 171-207, 1996
Carter, J.P. Xiao, B. 1994	A 2D - coupled finite element and boundary element scheme to simulate the elastic behaviour of jointed rocks	International Journal for Numerical and Analytical Methods in Geomechanics Vol. 18, pp.49-71, 1994
Chen, W.H. Chen, T.C. 1992	Boundary element analysis for contact problems with friction	Computers & Structures Vol. 45, No. 3, pp. 431-438, 1992
Chia Chin Lin Lawton, E.C. Caliendo, J.A. Anderson, L.R. 1996	An iterative finite element - boundary element algorithm	Computers & Structures Vol. 59, No.5, pp.899-909, 1996
Choi, C.K. Chung, G.T. 1996	A gap element for three dimensional elasto-plastic contact problems	Computers & Structures Vol. 61, No.6, pp. 1155-1167, 1996

Authors, Year	Title	Source
Crim, A.W. Basu, P.K. 1993	Combined Finite and Boundary Element Methods in P--Version	Recent Developments in Computational Mechanics ASME, Aerospace Division, ASME New York, Vol. 39, pp. 13- 28, 1993
Cruse, T.A. Wilson, R.B. 1978	Advanced applications of boundary integral equation methods	Nuclear Engineering and Design Vol. 46, pp. 223-234, 1978
Cruse, T.A. Osias, J.R. 1991	Issues in merging the finite element and boundary integral equation methods	Mathematical Computational Modelling Vol. 15, No. 3-5, pp. 103- 118, 1991
De Paula, F.A. Telles, J.C.F. 1989	A comparison between point collocation and Galerkin for stiffness matrices obtained by boundary elements	Engineering Analysis with Boundary Elements Vol. 6, No. 3, pp.123-128, 1989
Dostal, Z. Malik, J. 1995	Symmetric FE-BE coupling with iterations on the interface	International Journal for Numerical Methods in Engineering Vol.38, pp. 27-35, 1995
Eberhardsteiner, J. Mang, H.A. Torzicky, P. 1990	Three dimensional elasto-plastic hybrid BE-FE stress analysis of the driving of tunnels	Computational Engineering with Boundary Elements: Vol2: Solid and Computational Procedures, pp.139-159, Computational Mechanics Publications, Southampton, England, 1990
El-Zafrany, A. 1993	Techniques of the boundary element method	Ellis Horwood, Chichester, GB, 1993
El-Zafrany, A.M. 1998a	Lecture notes on finite element methods	Cranfield University, Cranfield, G.B.
El-Zafrany, A.M. 1998b	Lecture notes on plasticity	Cranfield University, Cranfield, G.B.

Authors, Year	Title	Source
Ezawa, Y. Okamoto, N. 1995	Development of contact stress analysis programs using the hybrid method of FEM and BEM	Computers & Structures Vol. 57, No. 4, pp. 691-698, 1995
Fadhil, S. Haynes, P. El-Zafrany, A. 1996	A coupled finite element - boundary element analysis for composite layered plates on elastic foundation	Boundary Elements 18, Computational Mechanics Publications, Southampton, G.B., 1996
Fenner, R.T. 1986	Engineering elasticity	Ellis Horwood, Chichester, GB, 1986
Floren, H. Duddeck, H. 1993	Efficient coupling of BEM and FEM in geomechanics	Boundary element 15, pp.527-537, Computational Mechanics Publications, Southampton, England, 1993
Fossa, G. Nappi, A. 1991	A critical discussion on possible variable changes related to elastic-plastic collocation BEM analysis	Engineering Analysis with Boundary Elements Vol. 8, pp. 24-35, 1991
Gakwaya, A. Lambert, D. Cardou, A. 1992	A boundary element and mathematical programming approach for frictional contact problems	Computers & Structures Vol. 42, No.3, pp. 341-353, 1992
Galin, L.A. 1953	Contact problems in the theory of elasticity	Gostekhizdat, Moskov, 1953.
Gangming, L. 1989	A new boundary element method coupled with FEM packages	Communications in Applied Numerical Methods Vol. 5, pp.365-371, 1989
Garrido, J.A. Foces, A. Paris, F. 1994	An incremental procedure for three dimensional contact problems with friction	Computers & Structures Vol. 50, No. 2, pp. 201-215, 1994

Authors, Year	Title	Source
Gerstle, W.H. Prasad, N.N.V. Xie, M. 1992	Solution method for coupled elastostatic BEM and FEM domains	7th International Conference on Boundary Element Technology, BETECH 92, pp.213-226 Computational Mechanics Publications, Southampton, England, 1992
Gladwell, G.M.L. 1980	Contact problems in the classical theory of elasticity	Sijthoff and Nordhoff, Netherlands, 1980.
Goodier, J.N. Hodge, P.G. 1958	Elasticity and plasticity	John Wiley & Sons, Inc., London, GB, 1958
Gosz, M. Moran, B. 1993	On the formulation and local implementation of a variationally coupled finite element and boundary element method	Computer Methods in Applied Mechanics and Engineering Vol. 107, pp.159-172, 1993
Graf, G.L. Gebre-Giorgis, Y. 1986	Combining finite element and boundary element analyses	International Off Highway and Powerplant Congress - Computer Applications, Analysis in Design, publ. by SAE, Warrendale, USA, pp. 75-87, 1986
Guarracino, F. Minutolo, V. Nunziante, L. 1992	A simple analysis of soil structure interaction by BEM-FEM coupling	Engineering Analysis with Boundary Elements Vol.10, pp.283-289, 1992
Guzelbey, I.H. 1992	Finite and boundary element analysis of elastoplastic finite strain contact problems	PhD thesis, Cranfield University, Cranfield, GB, 1992.
Haussler-Combe, U. 1996	Coupling boundary elements and finite elements - A structured approach	Computer methods in applied mechanics and engineering, Vol. 134, pp. 117-134, 1996

Authors, Year	Title	Source
Hartmann, F. 1981	The derivation of stiffness matrices from integral equations	Applied Mathematical Modelling Vol. 5, pp. 355-365, 1981
Hellen, T.K. 1988	A gap element facility in BERSAFE	CEGB Report TPRD/B/1043/R88
Hill, R. 1964	The mathematical theory of plasticity	Oxford University Press, London, GB, 1964
Hills Novell Sackfield 1993	Mechanics of elastic contacts	Butterworth-Heinemann, Oxford, GB, 1993.
Hinks, T. 1991	The application of the finite element method and boundary element method to fatigue and fracture	PhD thesis, Cranfield University, Cranfield, GB, 1991.
Hisatake, M. Ho, T. Ueda, H. 1983	Three dimensional symmetric coupling of boundary and finite element methods	Boundary Elements , Brebbia, C.,A. (edt), Springer Verlag, Berlin, 1983
Hitchings, D. 1988	Contact analysis using finite elements	Institute of physics conference - "Contact stress analysis I", 7.12.1988, I.O.P. Publishing
Huesmann, A. Kuhn, G. 1995	Automatic load incrementation technique for plane elastoplastic frictional contact problems using boundary element method	Computers & Structures Vol. 56, No. 5, pp. 733- 744, 1995
Hunek, I. 1992	On a penalty formulation for contact impact problems	Computers & Structures Vol. 48, No. 2, pp. 193- 203, 1993

Authors, Year	Title	Source
Husain, I.A. 1989	Finite element and boundary element methods for elastoplastic stress analysis of two-dimensional and axisymmetric problems	PhD thesis, Cranfield University, Cranfield, GB, 1989.
Israil, A.S.M. Banerjee, P.K. 1992	Advanced development of boundary element method for two-dimensional dynamic elastoplasticity	International Journal for Solids and Structures Vol. 29, No. 11, pp. 1433-1451, 1992
Johnson, W. Mellor, P.B. 1983	Engineering plasticity	Ellis Horwood ltd, Chichester, GB, 1983
Johnson, K.L. 1987	Contact mechanics	Cambridge University Press, Cambridge, GB, 1987.
Kanarakos, A. Provatidis, C. 1988	On the symmetrization of the BEM formulation	Computer Methods in Applied Mechanics and Engineering Vol. 71, pp.151-165, 1988
Karabalis, D.L. 1993	Formulation of 3D dynamic SSI analysis involving contact nonlinearities by time domain BEM-FEM	Engineering Analysis with Boundary Elements Vol. 11, pp. 277-284, 1993
Karami, G. 1991	Boundary element analysis of elasto-plastic contact problems	Computers & Structures Vol. 41, No. 5, pp. 927-935, 1991
Kelly, D.W. Mustoe, G.G.W. Zienkiewicz, O.C. 1979	Coupling boundary element methods with other numerical methods	Developments in Boundary Element Methods-1, Chapter 10, Banerjee, P.K. & Butterfield, R. (eds.), Applied Science Publishers, London, 1979
Kita, E. Kamiya, N. 1994	Subregion boundary element method	JSME International Journal Series A, Vol. 37, No. 4, 1994

Authors, Year	Title	Source
Kong, X.A. Gakawaya, A. Cardou, A. Cloutier, L. 1993	A numerical solution of general frictional contact problems by the direct boundary element and mathematical programming approach.	Computers & Structures Vol. 45, No. 1, pp. 95-112, 1993
Krishnamurthy, T. Raju, I.S. 1993	Coupling of finite and boundary element methods for two dimensional potential problems	International Journal for Numerical Methods in Engineering Vol.36, pp. 3595-3616, 1993
Krishnamurthy, T. Raju, I.S. Sistla, R. 1993	Coupling finite and boundary element methods for 2D elasticity problems	AIAA/ASME Structures, Structural Dynamics and Materials Conference - Part 3, pp.1274-1290, AIAA Washington, 1993
Kurz, S. Fetzer, J. Lehner, G. 1995	An improved algorithm for the BEM-FEM coupling method using domain decomposition	IEEE Transactions on Magnetics Vol.31, No. 3, pp.1737-1740, 1995
Landenberger, A. 1995	Finite element versus boundary element for elastic contact problems using gap finite elements	MSc thesis, Cranfield University, Cranfield, GB, 1995
Landenberger, A. El-Zafrany, A. 1996	Coupling of gap finite elements and boundary elements for elastic contact problems	Boundary Elements 18, pp. 373-383, Computational Mechanics Publications, Southampton, G.B., 1996
Lee, S.H. 1993	Rudimentary considerations for adaptive gap/friction element based on the penalty method	Computers & Structures Vol. 47, No. 6, pp. 1043-1056, 1993
Leung, K.L. 1995	2-D elastostatic analysis by a symmetric BEM/FEM scheme	Engineering Analysis with Boundary Elements Vol. 15, pp. 67-68, 1995

Authors, Year	Title	Source
Li, H.-B. Mang, H. Han, G.-H. Torzicky, P. 1986	A new method for the coupling of finite element and boundary element discretized subdomains of elastic bodies	Computer Methods in Applied Mechanics and Engineering Vol. 54, pp. 161-185, 1986
Liu, M. Farris, T.N. 1993	Three dimensional infinite boundary elements for contact problems	International Journal for Numerical Methods in Engineering Vol. 36, pp. 3381-3398, 1993
Liu, S.B. Tan, C.L. 1990	Boundary element contact mechanics analysis of pin loaded lugs	Computers & Structures Vol. 37, No.5, pp.625-635, 1990
Lu, X. Du, Q. 1994	Finite elements in combination with infinite boundary elements for concrete arch dam foundation	Boundary Element 16, pp. 585-592, Computational Mechanics Publications, Southampton, England, 1993
Lu, Y.Y. Belytschko, T. Liu, W.K. 1991	A variationally coupled FE-BE method for elasticity and fracture mechanics	Computer Methods in Applied Mechanics and Engineering Vol. 85, No. 1, pp. 21-37, 1991
Ma, J. Le, M. 1992	A new method for coupling of boundary element method and finite element method	Applied Mathematical Modelling Vol. 16, pp.43-46, 1992
Man, K.W. Aliabadi, M.H. 1993	BEM frictional contact analysis: Modelling considerations	Engineering Analysis with Boundary Elements Vol.11, pp.77-85, 1993
Man, K.W. Aliabadi, M.H. Rooke, D.P. 1993	BEM frictional contact analysis: Load incremental technique	Computers & Structures Vol. 47, No. 6, pp. 893-905, 1993

Authors, Year	Title	Source
Mang, H.A. Torzicky, P. Chen, Z.Y. 1989	On the mechanical inconsistency of symmetrization of unsymmetric coupling matrices for BEFEM discretization of solids	Computational Mechanics Vol. 4, pp.301-308, 1989
Mazurkiewicz, M. Ostachowicz, W. 1983	Theory of finite element method for elastic contact problems of solid bodies	Computers & Structures Vol. 17, No. 1, pp.51-59, 1983
Mendelson, A. 1968	Plasticity : Theory and Application	The Macmillan Company, New York Collier-Macmillan ltd, London, G.B., 1968
Mitra, A.K. Ingber, M.S. 1993	A multiple-node method to resolve the difficulties in the boundary element method caused by corners and discontinuous boundary conditions	International Journal for Numerical Methods in Engineering Vol. 36, pp. 1735-1746, 1993
Mushkelishvili, N.I. 1953	Some basic problems of mathematic theory of elasticity	Nordhoff, Netherlands, 1953.
Okada, H. Atluri, S.N. 1994	Recent developments in the field-boundary element method for finite/small strain elastoplasticity	International Journal for Solids and Structures Vol. 31, No. 12/13, pp. 1737-1775, 1994
Olukoko, O.A. Becker, A.A. Fenner, R.T. 1993a	Three benchmark examples for frictional contact modelling using finite element and boundary element methods	Journal of Strain Analysis Vol. 28, No. 4, pp. 293-301, 1993
Olukoko, O.A. Becker, A.A. Fenner, R.T. 1993b	A new boundary element approach for contact problems with friction	International Journal for Numerical Methods in Engineering Vol. 36, pp. 2625-2642, 1993

Authors, Year	Title	Source
Olukoko, O.A. Becker, A.A. Fennner, R.T. 1993c	Independent meshing of contacting surfaces using fictitious nodes in boundary element analysis	Computers & Structures Vol. 48, No. 5, pp. 787-801, 1993
Osias, J.R. Wilson, R.B. Seitelmann, L.H. 1977	Combined boundary integral equation / finite element analysis of cracked solids	International Symposium on Innovative Numerical Analysis in Applied Engineering Science, Versailles, France, 1977
Owen, D.R.J. Hinton, E. 1980	Finite elements in plasticity: theory and practise.	Pineridge Press ltd, Swansea, G.B., 1980
Papia, M. 1988	Analysis of infilled frames using a coupled finite element and boundary element scheme	International Journal for Numerical Methods in Engineering Vol. 26, pp. 731-742, 1988
Paris, F. Foces, A. Garrido, J.A. 1992	Application of boundary element method to solve three dimensional elastic contact problems without friction	Computers & Structures Vol. 43, No. 1, pp.19-30, 1992
Paris, F. Blazquez, A. Canas, J. 1995	Contact problems with nonconforming discretizations using boundary element method	Computers & Structures Vol. 57, No. 5, pp. 829-839, 1995
Park, Y.H. Becker, A.A. Fenner, R.T. 1995	Contact mechanics analysis of frictional slip of inclusions embedded in dissimilar matrices using the boundary element method	Journal of Strain Analysis Vol. 30, No. 4, pp. 245-255, 1995
Pavlatos, G.D. Beskos, D.E. 1994	Dynamic elastoplastic analysis by BEM/FEM	Engineering Analysis with Boundary Elements Vol. 14, pp. 51-63, 1994

Authors, Year	Title	Source
Polizotto, C. Zito, M. 1994	Variational formulations for coupled BE/FE methods in elastostatics	ZAMM- Z. angew. Math. Mech. Vol 74, No. 11, pp.533-543, 1994
Prager, W. 1955	The theory of plasticity: a survey of recent achievements	Proc. Instn. Mech. Engrs. Vol. 169, pp 31-57, 1955
Remacle, J.F. Nicolet, A. Ume, M. Genon, A. 1993	Hybrid solver for BEM-FEM coupling	Boundary Element 15, pp.505-516 Computational Mechanics Publications, Southampton, England, 1993
Runeson, K. 1993	Formulation and implementation of conditions for frictional contact	Engineering Computations Vol. 10, pp. 3-14, 1993
Sachdeva, T.D. Ramakrishnan, C.V. Natarajan, R. 1981	A finite element method for the elastic contact problem.	Trans ASME Journal Applied Mechanics Vol. 103, pp. 456-461, 1981
Savin, G.N. 1961	Stress concentration around holes	Pergamon Press, Oxford, 1961.
Schnack, E. 1987	A hybrid BEM model	International Journal for Numerical Methods in Engineering Vol. 24, pp. 1015-1025, 1987
Shaw, R.P. Falby, W. 1977	FEBIE- A combined finite element - boundary integral equation method	International Symposium on Innovative Numerical Analysis in Applied Engineering Science, Versailles, France, 1977
Shtaerman, I.Y. 1949	Contact problems in the theory of elasticity	Gostekhizdat, Moskov, 1949.

Authors, Year	Title	Source
Simunovic, S. Saigal, S. 1992	Frictionless contact with BEM using quadratic programming	Journal of Engineering Mechanics Vol. 118, No. 9, pp. 1876-1891, 1992
Simunovic, S. Saigal, S. 1995a	Quadratic programming contact formulation for elastic bodies using boundary element method	AIAA Journal Vol. 33, No. 2, pp. 325-331, 1995
Simunovic, S. Saigal, S. 1995b	A linear programming formulation for incremental contact analysis	International Journal for Numerical Methods in Engineering Vol. 38, pp. 2703-2725, 1995
Simunovic, S. Saigal, S. 1995c	Contact surface optimization using the boundary element method	Computers & Structures Vol. 56, No. 5, pp. 745-750, 1995
Sipicic, S.R. Rabinovich, V.L. 1993	Boundary element tools for the finite body unilateral contact problem	Boundary Element 15, pp.65-72, Computational Mechanics Publications, Southampton, England, 1993
Spann, W. 1997	Error estimates for the boundary element approximation of a contact problems in elasticity	Mathematical Methods in the Applied Sciences Vol. 20, pp. 205-217, 1997
Stadler, J.T. Weiss, R.O. 1979	Analysis of contact through finite element gaps	Computers & Structures Vol. 10, pp. 867-873, 1979
Stephan, E.P. 1992	Coupling of finite elements and boundary elements for some non-linear interface problems.	Computer Methods in Applied Mechanics and Engineering Vol. 101, pp.61-72, 1992
Stern, M. Taylor, L.M. 1987	Coupling boundary integral and finite element formulations for nonlinear halfspace problems	Advanced Boundary Element Methods, Cruse, T.A. (edt.), pp.397-407, Springer Verlag, Berlin, 1987

Authors, Year	Title	Source
Takahashi, S. Brebbia, C.A. 1991	Validation of the boundary element flexibility approach for elastic contact analysis	Computer Methods in Applied Mechanics and Engineering Vol. 93, pp. 151-168, 1991
Tullberg, O. Bolteus, L. 1982	A critical study of different boundary element stiffness matrices	Boundary Element Methods in Engineering , Brebbia, C.,A. (edt), pp. 621-635, Springer Verlag, Berlin, 1982
Turke, K. Schnack, E. 1996	The Neumann series - a new possibility for coupling of FEM and BEM	Boundary Elements 18, Computational Mechanics Publications, Southampton, G.B., 1996
Wearing, J.L. Sheikh, M.A. Burstow, M.C. 1990	A combined finite element boundary element approach for elasto-plastic analysis	Boundary Integral Methods, Morino, L. (edt.), pp.499-509, Springer Verlag, Berlin, 1990
Wearing, J.L. Burstow, M.C. 1994	Elasto-plastic analysis using a coupled boundary element finite element technique	Engineering Analysis with Boundary Elements Vol. 14, pp. 39-49, 1994
Xiao, B. Carter, J.P. Alehossen, H. 1994	Elastoplastic analysis of jointed rocks using a coupled finite element and boundary element method	International journal for numerical and analytical methods in geomechanics Vol. 18, pp. 445-466, 1994
Yamazaki, K. Mori, M. 1989	Analysis of an elastic contact problem by the boundary element method	JSME International Journal Series I, Vol. 32, No. 4, pp. 508-513, 1989
Yamazaki, K. Sakamoto, J. Takumi, S. 1994	Penalty method for three dimensional elastic contact problems by boundary element method	Computers & Structures Vol. 52, No. 5, pp. 895-903, 1994

Authors, Year	Title	Source
Zeng, X. Kallivokas, L.F. Bielak, J. 1993	A symmetric variational finite element boundary integral equation coupling method	Computers & Structures Vol. 46, No.6, pp.995- 1000, 1993
Zhu, C. 1995	A new boundary element mathematical programming approach for contact problems with friction	Communications in Numerical Methods in Engineering Vol. 11, pp. 683-690, 1995
Ziegler, H. 1959	A modification of Prager's hardening rule	Quart. Appl. Math. Vol. 17, pp. 55-65, 1959
Zienkiewicz, O.C. Kelly, D.W. Bettess, P. 1977	The coupling of the finite element method and boundary solution procedures	International Journal for Numerical Methods in Engineering Vol. 11, pp. 355-375, 1977

APPENDIX A

FUNDAMENTAL SOLUTION PARAMETERS

$$G_{11} = \frac{1}{8 \pi \mu_s (1-p)} \left[-(3-4p) \log r + \left(\frac{\partial r}{\partial x} \right)^2 \right]$$

$$G_{12} = \frac{1}{8 \pi \mu_s (1-p)} \frac{\partial r}{\partial x} \frac{\partial r}{\partial y}$$

$$G_{21} = \frac{1}{8 \pi \mu_s (1-p)} \frac{\partial r}{\partial x} \frac{\partial r}{\partial y}$$

$$G_{22} = \frac{1}{8 \pi \mu_s (1-p)} \left[-(3-4p) \log r + \left(\frac{\partial r}{\partial y} \right)^2 \right]$$

$$F_{11} = - \frac{1}{4 \pi (1-p) r} \frac{\partial r}{\partial n} \left[(1-2p) + 2 \left(\frac{\partial r}{\partial x} \right)^2 \right]$$

$$F_{12} = - \frac{1}{4 \pi (1-p) r} \left[2 \frac{\partial r}{\partial n} \frac{\partial r}{\partial x} \frac{\partial r}{\partial y} + (1-2p) \left(m \frac{\partial r}{\partial x} - l \frac{\partial r}{\partial y} \right) \right]$$

$$F_{21} = - \frac{1}{4 \pi (1-p) r} \left[2 \frac{\partial r}{\partial n} \frac{\partial r}{\partial x} \frac{\partial r}{\partial y} - (1-2p) \left(m \frac{\partial r}{\partial x} - l \frac{\partial r}{\partial y} \right) \right]$$

$$F_{22} = - \frac{1}{4 \pi (1-p) r} \frac{\partial r}{\partial n} \left[(1-2p) + 2 \left(\frac{\partial r}{\partial y} \right)^2 \right]$$

where:

$$r = \sqrt{(x-x_i)^2 + (y-y_i)^2}$$

$$\frac{\partial r}{\partial x} = \frac{x-x_i}{r}$$

$$\frac{\partial r}{\partial y} = \frac{y-y_i}{r}$$

$$\frac{\partial r}{\partial n} = l \frac{\partial r}{\partial x} + m \frac{\partial r}{\partial y}$$

APPENDIX B

CORNER MODEL

Consider the case of a boundary as shown in figure B.1. The boundary is separated into the boundary without corner $\Gamma - \Gamma_\epsilon$ and the circular arc with radius ϵ which is denoted Γ_ϵ . It is clear that the following relation is valid for such a case:

$$\oint_{\Gamma} f d\Gamma = \lim_{\epsilon \rightarrow 0} \int_{\Gamma - \Gamma_\epsilon} f d\Gamma + \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} f d\Gamma$$

where the boundary without the effect of the corner may be denoted as Γ' and hence:

$$\oint_{\Gamma'} f d\Gamma = \lim_{\epsilon \rightarrow 0} \int_{\Gamma - \Gamma_\epsilon} f d\Gamma$$

Therefore the boundary integral equation in x -direction (equation (3.14)) may be rewritten as follows:

$$\begin{aligned} C_i u_i + \oint_{\Gamma'} (F_{11} u + F_{21} v) d\Gamma + \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} (F_{11} u + F_{21} v) d\Gamma \\ = \oint_{\Gamma'} (G_{11} T_x + G_{21} T_y) d\Gamma + \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} (G_{11} T_x + G_{21} T_y) d\Gamma + U(x_i, y_i) \end{aligned}$$

and it can be proved that (Hussain(1989)):

$$\lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} (G_{11} T_x + G_{21} T_y) d\Gamma = 0$$

Thus:

$$\begin{aligned} C_i u_i + \oint_{\Gamma'} (F_{11} u + F_{21} v) d\Gamma + \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} (F_{11} u + F_{21} v) d\Gamma \\ = \oint_{\Gamma'} (G_{11} T_x + G_{21} T_y) d\Gamma + U(x_i, y_i) \end{aligned}$$

It is also clear from the mean value theorem that if the source point is located at the corner:

$$\lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} (F_{11} u + F_{21} v) d\Gamma = \left(\lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} F_{11} d\Gamma \right) u_i + \left(\lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} F_{21} d\Gamma \right) v_i$$

and therefore:

$$\begin{aligned} C_{11}(x_i, y_i) u_i + C_{12}(x_i, y_i) v_i + \int_{\Gamma'} (F_{11} u + F_{21} v) d\Gamma \\ = \int_{\Gamma'} (G_{11} T_x + G_{21} T_y) d\Gamma + U(x_i, y_i) \end{aligned}$$

where:

$$\begin{aligned} C_{11}(x_i, y_i) &= C_i + \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} F_{11} d\Gamma \\ C_{12}(x_i, y_i) &= \lim_{\epsilon \rightarrow 0} \int_{\Gamma_\epsilon} F_{21} d\Gamma \end{aligned}$$

which is equation (3.47). Notice that equation (3.48) can be derived in a similar way. In order to evaluate the C -coefficients, consider a case where the whole body is translating without any body forces or surface tractions applied. Then:

$$u = u_i = \alpha \quad ; \quad v = v_i = \beta$$

which yields in the above corrected boundary integral equation:

$$C_{11} \alpha + C_{12} \beta + \left(\int_{\Gamma'} F_{11} d\Gamma \right) \alpha + \left(\int_{\Gamma'} F_{21} d\Gamma \right) \beta = 0$$

and by comparing the coefficient it is clear that:

$$\begin{aligned} C_{11} &= - \int_{\Gamma'} F_{11} d\Gamma \\ C_{12} &= - \int_{\Gamma'} F_{21} d\Gamma \end{aligned}$$

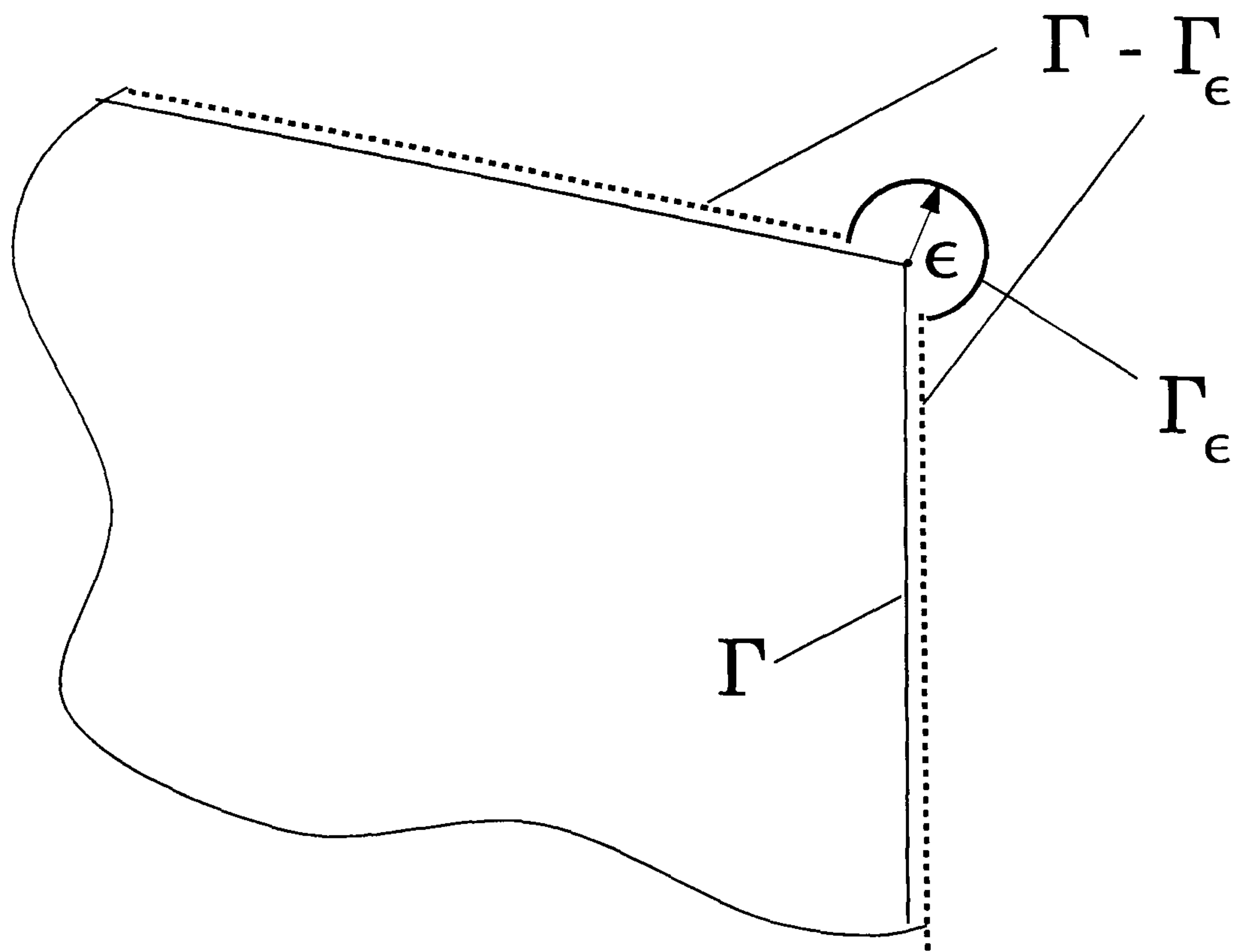


Figure B.1 The modelling of a boundary corner with a circular arc.

APPENDIX C

DERIVATION OF THE ELEMENTAL TRANSFORMATION MATRIX

Consider a n -node finite element side as shown in figure C.1, with a thickness h and a boundary Γ .

An equivalent nodal traction vector $\mathbf{T}_{(e)}$ is defined such that the work done by it is equal to the work done by the actual nodal forces $\mathbf{F}_{(e)}$, during any virtual displacement. The work done by these tractions during virtual displacement δu in the x -direction and δv in the y -direction can be expressed as follows:

$$\delta W = \int_{\Gamma} (T_x \delta u + T_y \delta v) h d\Gamma$$

Writing T_x , T_y , δu and δv in terms of their nodal values, i.e.

$$T_x = \sum_{j=1}^n (T_x)_j N_j(\xi)$$

$$\delta u = \sum_{i=1}^n \delta u_i N_i(\xi)$$

etc., where N_i and N_j represent Lagrangian shape functions in terms of the intrinsic boundary coordinate ξ along the finite elements side, then the previous expression of the work done can be rewritten as follows:

$$\delta W = \sum_{i=1}^n \left[\sum_{j=1}^n \left\{ (T_x)_j \int_{\Gamma} N_i(\xi) N_j(\xi) h d\Gamma \right\} \delta u_i \right. \\ \left. + \sum_{j=1}^n \left\{ (T_y)_j \int_{\Gamma} N_i(\xi) N_j(\xi) h d\Gamma \right\} \delta v_i \right]$$

The corresponding work done by the nodal force vector acting on the finite element is:

$$\delta W = \sum_{i=1}^n \left[(F_x)_i \delta u_i + (F_y)_i \delta v_i \right]$$

Comparing the previous two equations, for the equivalence of the work done it can be deduced that:

$$(F_x)_i = \sum_{j=1}^n (T_x)_j \int_{\Gamma} N_i(\xi) N_j(\xi) h d\Gamma$$

$$(F_y)_i = \sum_{j=1}^n (T_y)_j \int_{\Gamma} N_i(\xi) N_j(\xi) h d\Gamma$$

Therefore, a nodal force vector $\mathbf{F}_{(e)}$ of a loaded finite element side may be expressed in terms of an nodal traction vector $\mathbf{T}_{(e)}$ as follows:

$$\mathbf{F}_{(e)} = \mathbf{C}_{(e)} \mathbf{T}_{(e)}$$

where

$$\mathbf{C}_{(e)} = h \int_{\Gamma} \begin{bmatrix} N_1 N_1 & 0 & N_1 N_2 & 0 & \dots & \dots & N_1 N_n & 0 \\ 0 & N_1 N_1 & 0 & N_1 N_2 & \dots & \dots & 0 & N_1 N_n \\ N_2 N_1 & 0 & N_2 N_2 & 0 & \dots & \dots & N_2 N_n & 0 \\ 0 & N_2 N_1 & 0 & N_2 N_2 & \dots & \dots & 0 & N_2 N_n \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ N_n N_1 & 0 & N_n N_2 & 0 & \dots & \dots & N_n N_n & 0 \\ 0 & N_n N_1 & 0 & N_n N_2 & \dots & \dots & 0 & N_n N_n \end{bmatrix} d\Gamma$$

$$\mathbf{F}_{(e)} = \left\{ (F_x)_1 \quad (F_y)_1 \quad (F_x)_2 \quad (F_y)_2 \quad \dots \quad (F_x)_n \quad (F_y)_n \right\}$$

$$\mathbf{T}_{(e)} = \left\{ (T_x)_1 \quad (T_y)_1 \quad (T_x)_2 \quad (T_y)_2 \quad \dots \quad (T_x)_n \quad (T_y)_n \right\}$$

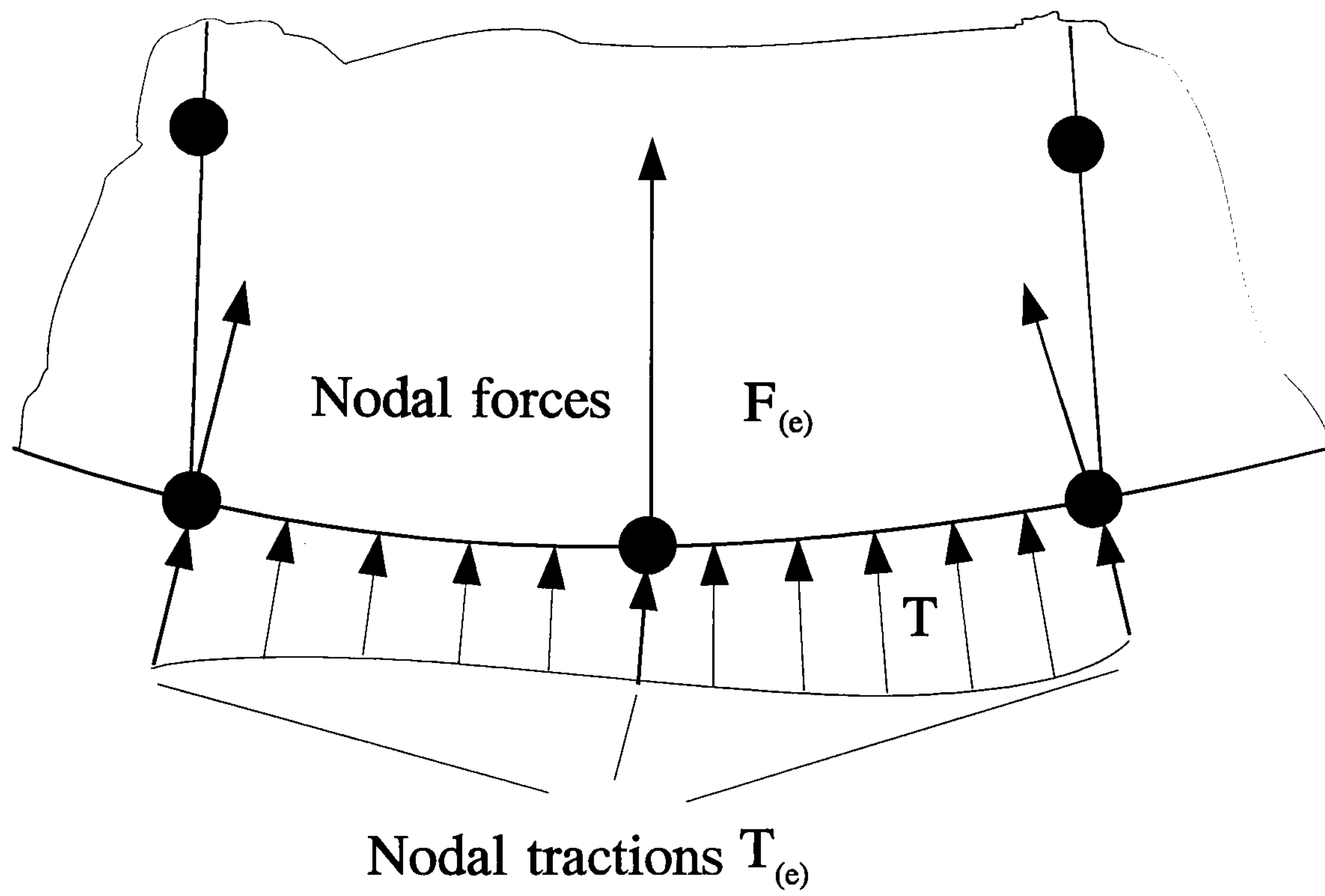


Figure C.1 A finite element side with nodal forces and equivalent nodal tractions.

APPENDIX D

FINITE ELEMENT PLASTICITY PROCEDURES

D.1 INITIAL STRESS PROCEDURE

The finite element equations of equilibrium for linear and non-linear analysis were defined as follows:

$$\sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t \boldsymbol{\sigma} \, dx \, dy \, dz - \mathbf{F} = \mathbf{0} \quad (5.23)$$

1st Iteration:

A first approximation can be undertaken by running an analysis with :

$$\mathbf{K} \boldsymbol{\delta}_1 = \mathbf{F}$$

where $\mathbf{K}$ can either be the elastic stiffness matrix or the elastoplastic stiffness matrix as described in section 5.2.3. It is clear for elastoplastic situations that $\boldsymbol{\delta}_1$ can not fulfil the equilibrium equation (5.23) accurately.

Strain and stress results can be evaluated as follows:

$$\begin{aligned} \boldsymbol{\epsilon}_1 &= \mathbf{B} \boldsymbol{\delta}_1 \\ \boldsymbol{\sigma}'_1 &= \mathbf{D} \boldsymbol{\epsilon}_1 \end{aligned}$$

where $\boldsymbol{\sigma}'_1$ is the uncorrected stress vector. It is possible to obtain a more accurate stress-strain relationship by making use of the infinitesimal elastoplastic stress-strain relationship, which was presented in section 5.1:

$$d \boldsymbol{\sigma} = \mathbf{D}_{ep} d \boldsymbol{\epsilon} \quad (5.6)$$

A very good approximation for an accurate stress-strain relationship can be obtained by dividing the applied strains into a number of very small subincrements. For each of the subincrements an elastoplastic stress strain matrix $\mathbf{D}_{ep}$ may be evaluated with hardening and flow parameters which are averaged over the subincrements as shown in Guzelbey (1992). This leads to a corrected stress vector $\boldsymbol{\sigma}_1$, which corresponds accurately to the strain vector $\boldsymbol{\epsilon}_1$, which is shown for a uniaxial case in figure D.1.

Figure D.2 shows the stress-strain diagram obtained for the first iteration for a uniaxial case. It is generally clear that:

$$\boldsymbol{\sigma}'_1 = \boldsymbol{\sigma}_1 + (\Delta \boldsymbol{\sigma})_1$$

where $(\Delta \boldsymbol{\sigma})_1$ is the error introduced in the stress evaluation in the first iteration. Hence it is clear that the residual force error of this first iteration can be defined in terms of the "initial stress" $(\Delta \boldsymbol{\sigma})_1$ as follows:

$$\mathbf{R}_1^{(\sigma)} = \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t (\Delta \boldsymbol{\sigma})_1 dx dy dz$$

2nd Iteration:

Within the second iteration an approximate solution of the equilibrium equation formed by the previous equation can be found from:

$$\mathbf{K} \boldsymbol{\delta}_2 = \mathbf{R}_1^{(\sigma)}$$

where $\boldsymbol{\delta}_2$ can not fulfil the equilibrium requirements accurately. Strain and stress results can be evaluated as follows:

$$\boldsymbol{\epsilon}_2 = \mathbf{B} \boldsymbol{\delta}_2$$

$$\boldsymbol{\sigma}'_2 = \mathbf{D} \boldsymbol{\epsilon}_2$$

and a corrected stress vector σ_2 can be evaluated by dividing ϵ_2 into subincrements and using D_{ep} for each of the subincrements. Again it is clear that:

$$\sigma'_2 = \sigma_2 + (\Delta \sigma)_2$$

where $(\Delta \sigma)_1$ is the error introduced in the stress evaluation in the second iteration. Thus the residual force error of the second iteration can be defined as follows:

$$R_2^{(\sigma)} = \sum_{e=1}^m \iiint_{\text{element } e} B^t (\Delta \sigma)_2 dx dy dz$$

n-th Iteration:

Solve:

$$K \delta_n = R_{n-1}^{(\sigma)}$$

Evaluate

$$\epsilon_n = B \delta_n$$

$$\sigma'_n = D \epsilon_n$$

Find σ_n by dividing ϵ_n into subincrements and using D_{ep} for each of the subincrements. Then:

$$\sigma'_n = \sigma_n + (\Delta \sigma)_n$$

Evaluate:

$$R_n^{(\sigma)} = \sum_{e=1}^m \iiint_{\text{element } e} B^t (\Delta \sigma)_n dx dy dz$$

which is the residual error force (correction force) of the n -th iteration. If the obtained results are not accurate enough then run another iteration with $R_n^{(\sigma)}$.

D.2 INITIAL STRAIN PROCEDURE

The stress σ of the equilibrium equation (5.23) may be expressed in terms of displacements and strains as follows:

$$\sigma = D \epsilon_e = D (\epsilon - \epsilon_p) = D (B \delta_e - \epsilon_p)$$

Substituting the latter expression into the equations of equilibrium (5.23) yields:

$$\sum_{e=1}^m \iiint_{\text{element } e} B^t D (B \delta_e - \epsilon_p) dx dy dz - F = O$$

$$\text{i.e.} \quad \left[\sum_{e=1}^m \iiint_{\text{element } e} B^t D B dx dy dz \right] \delta = F + \sum_{e=1}^m \iiint_{\text{element } e} B^t D \epsilon_p dx dy dz$$

$$\text{or} \quad K \delta = F + \sum_{e=1}^m \iiint_{\text{element } e} B^t D \epsilon_p dx dy dz$$

1st Iteration

The equations of equilibrium are thoroughly satisfied if:

$$K \delta_1 = F + \sum_{e=1}^m \iiint_{\text{element } e} B^t D (\epsilon_p)_1 dx dy dz$$

In the first approximation it is assumed that $(\epsilon_p)_1 = 0$ and hence an approximate solution can be found by solving:

$$K \delta_1 = F$$

Corresponding stress and strain results can be evaluated as follows:

$$\epsilon'_1 = B \delta_1$$

$$\sigma_1 = D \epsilon'_1$$

where ϵ'_1 is the uncorrected strain which is equivalent to the elastic strain. It is then possible to evaluate $(\epsilon_p)_1$ by a similar subincrementation procedure as the one used for the initial stress procedure. For each of the defined subincrements, a plastic strain vector may be evaluated with hardening and flow parameters averaged over the subincrement (Guzelbey(1992)). Then it is clear that the corrected strain ϵ_1 may be expressed as follows:

$$\epsilon_1 = \epsilon'_1 + (\epsilon_p)_1$$

Notice that the obtained results were not accurate yet because they were based on δ_1 which was based on the initial assumption that $(\epsilon_p)_1 = 0$. Hence another iteration becomes necessary.

2nd iteration:

Defining accurate solutions δ and ϵ_p , which fulfil the equilibrium requirements exactly as follows:

$$\begin{aligned}\delta &= \delta_1 + \delta_2 \\ \epsilon_p &= (\epsilon_p)_1 + (\epsilon_p)_2\end{aligned}$$

Then the equation of equilibrium may be rewritten such that:

$$K (\delta_1 + \delta_2) = F + \sum_{e=1}^m \int \int \int_{\text{element } e} B^t D \left((\epsilon_p)_1 + (\epsilon_p)_2 \right) dx dy dz$$

In the first iteration it was already established that:

$$K \delta_1 = F$$

Moreover by assuming initially that $(\epsilon_p)_2 = 0$ it can be seen that:

$$K \delta_2 = R_1^{(\epsilon)}$$

where:

$$\mathbf{R}_1^{(\epsilon)} = \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t \mathbf{D} (\boldsymbol{\epsilon}_p)_1 dx dy dz$$

Thus a solution for $\boldsymbol{\delta}_2$ can be obtained and then:

$$\boldsymbol{\epsilon}'_2 = \mathbf{B} \boldsymbol{\delta}_2$$

$$\boldsymbol{\sigma}_2 = \mathbf{D} \boldsymbol{\epsilon}'_2$$

and again a plastic strain vector $(\boldsymbol{\epsilon}_p)_2$ which corresponds to the obtained stress and uncorrected strain vector can be evaluated by dividing the current increment into subincrements. Hence:

$$\boldsymbol{\epsilon}_2 = \boldsymbol{\epsilon}'_2 + (\boldsymbol{\epsilon}_p)_2$$

n-th iteration:

Solve:

$$\mathbf{K} \boldsymbol{\delta}_n = \mathbf{R}_{n-1}^{(\epsilon)}$$

where:

$$\mathbf{R}_{n-1}^{(\epsilon)} = \sum_{e=1}^m \iiint_{\text{element } e} \mathbf{B}^t \mathbf{D} (\boldsymbol{\epsilon}_p)_{n-1} dx dy dz$$

Calculate:

$$\boldsymbol{\epsilon}'_n = \mathbf{B} \boldsymbol{\delta}_n$$

$$\boldsymbol{\sigma}_n = \mathbf{D} \boldsymbol{\epsilon}'_n$$

Evaluate $(\boldsymbol{\epsilon}_p)_n$ by dividing the current increment into subincrements and then:

$$\boldsymbol{\epsilon}_n = \boldsymbol{\epsilon}'_n + (\boldsymbol{\epsilon}_p)_n$$

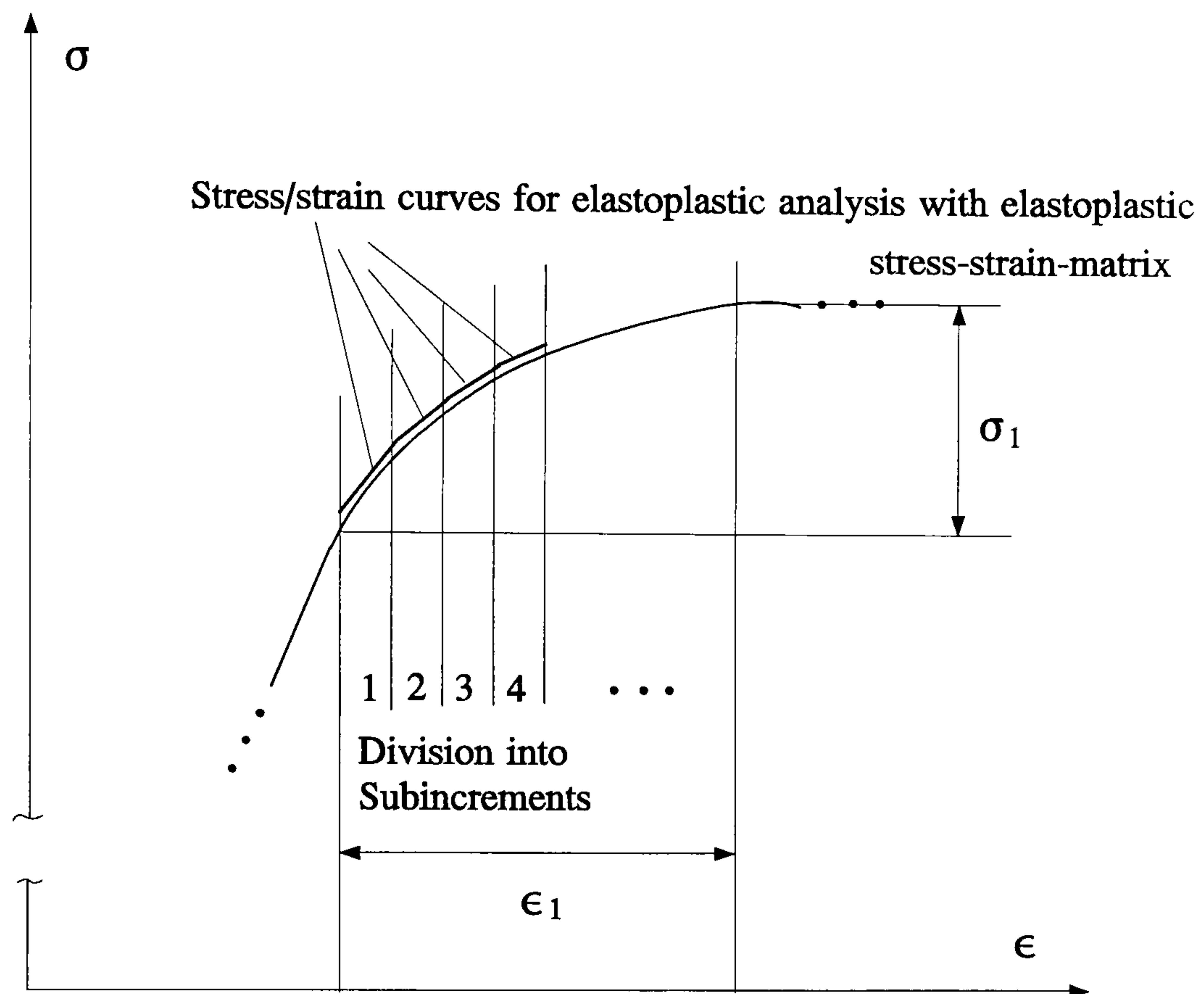


Figure D.1 Division into subincrements in order to obtain corrected stress value.

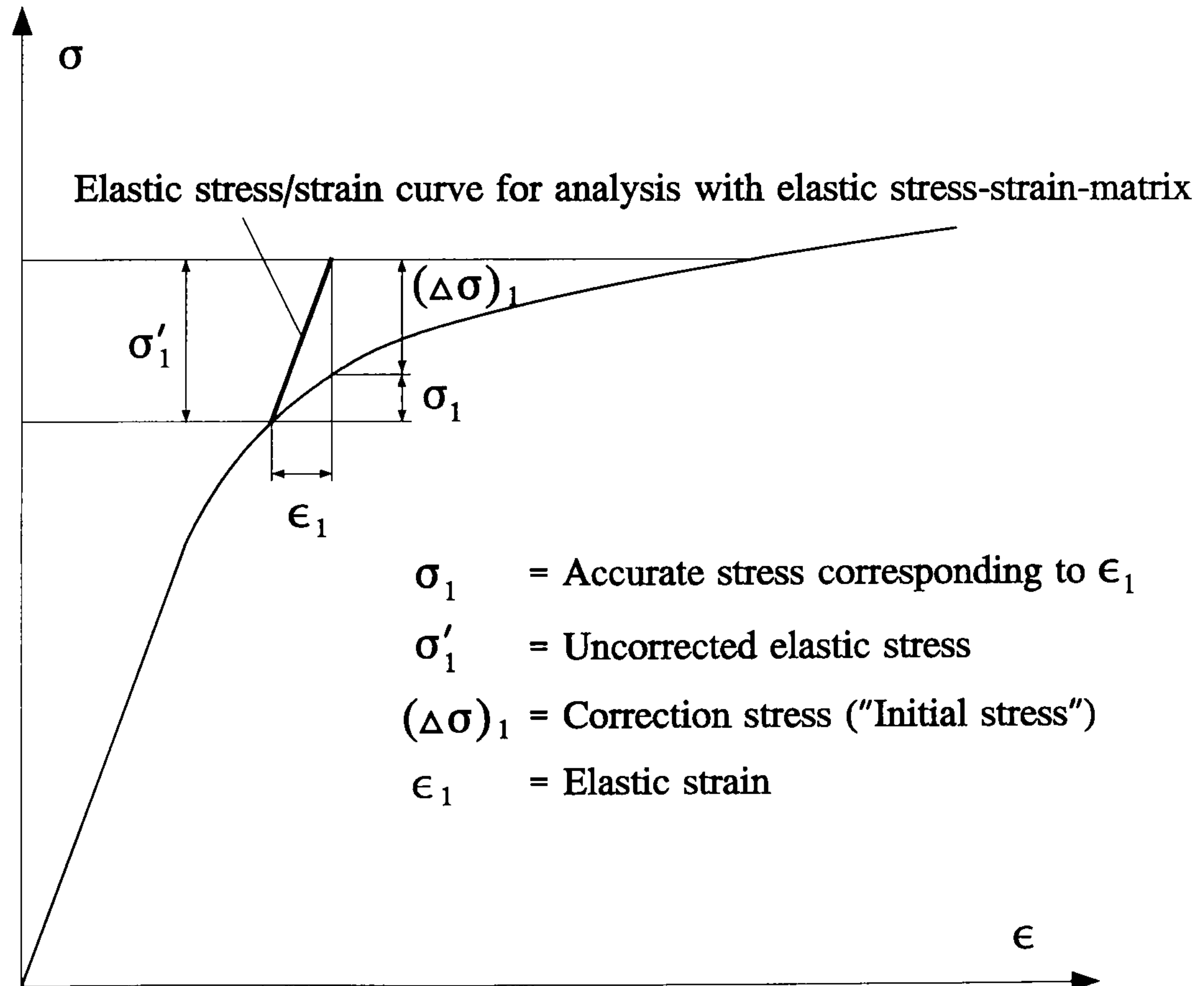


Figure D.2 Stress strain diagram for the first plasticity iteration.

APPENDIX E

GAP ELEMENT ROTATION

As shown in section 6.1.1, the gap element equations may be expressed in the following matrix form:

$$\mathbf{K}_{e_L} \boldsymbol{\delta}_{e_L} = \mathbf{F}_{e_L}$$

where the subscript L indicates that the matrix and vectors are defined with respect to the local coordinate system of the gap element, i.e. the normal and tangential directions of node I of the gap element.

In order to be able to assemble the gap element stiffness matrix into the subregional stiffness matrix it becomes necessary to transform the local gap element equations into equations formulated with respect to the global coordinate system, i.e. the x - and y -coordinates. If we define the local coordinate system as being rotated by the angle α about the global coordinate system as shown in figure E.1, it can be deduced that:

$$\begin{bmatrix} (u_n)_I \\ (u_t)_I \\ (u_n)_J \\ (u_t)_J \end{bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ -m & l & 0 & 0 \\ 0 & 0 & l & m \\ 0 & 0 & -m & l \end{bmatrix} \begin{bmatrix} (u)_I \\ (v)_I \\ (u)_J \\ (v)_J \end{bmatrix}$$

i.e.:

$$\boldsymbol{\delta}_{e_L} = \boldsymbol{\Theta} \boldsymbol{\delta}_{e_G}$$

where:

$$l = \cos \alpha$$

$$m = -\sin \alpha$$

and the subscript G indicates vectors and matrices with respect to the global coordinate system. Similarly it can be seen that:

$$\mathbf{F}_{e_G} = \boldsymbol{\Theta}^t \mathbf{F}_{e_L}$$

Multiplying both sides of the gap element equations by Θ^t :

$$\Theta^t K_{e_L} \delta_{e_L} = \Theta^t F_{e_L}$$

Substituting the two previous equations into each other:

$$\Theta^t K_{e_L} \delta_{e_L} = F_{e_G}$$

and substituting $\delta_{e_L} = \Theta \delta_{e_G}$ into the previous equation:

$$\Theta^t K_{e_L} \Theta \delta_{e_G} = F_{e_G}$$

or:

$$K_{e_G} \delta_{e_G} = F_{e_G}$$

where

$$K_{e_G} = \Theta^t K_{e_L} \Theta$$

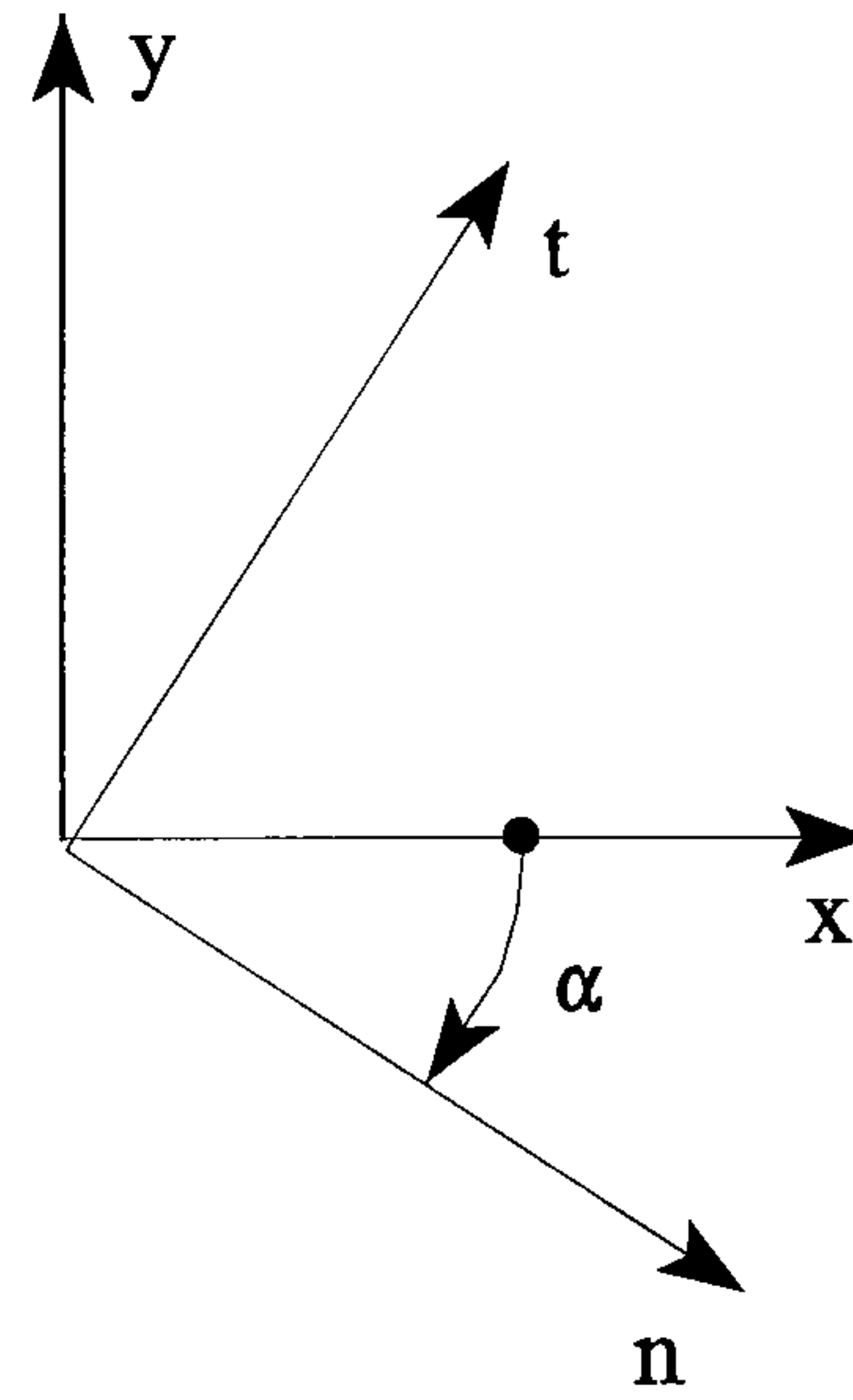


Figure E.1 Rotation of coordinate system.

APPENDIX F

ASSEMBLY OF THE SUBREGIONAL EQUATIONS INTO SOLVABLE STRUCTURAL EQUATIONS

There are many different possibilities of how gap element subregions, finite element subregions and boundary element subregions can be defined in order to model a problem accurately. It is possible to define any combination of these subregions if the matrices are partitioned accurately and the interface conditions are applied properly.

The partitioning and application of interface conditions were already presented for the case of the coupling of two BEM-subregions in section 3.3.2, for the coupling of an FEM subregion and a BEM subregion in section 4.3 and for the definition of an artificial gap element subregion between two FEM- or BEM-subregions in section 6.2. However there are plenty of combinations of subregions, which may occur in practical problems, such as for example the model of the elastoplastic punch case in figure 8.34 with two BEM-subregions, one FEM subregion and one artificial gap element subregion to be coupled.

The schematic procedure for the assembly of subregional equations into condensed solvable structural equations can be seen in figure F.1 for the example of an artificial gap element subregion γ between a BEM subregion α and a FEM subregion β (as shown in figure 6.2). Similarly to this example any other combination of subregions can be modelled by following the steps of figure 7.1 for the evaluation and partitioning of subregional FEM-, BEM- or gap element-equations. The displacement and traction components of any interface between two subregions need to be separated by matrix partitioning and then the appropriate interface boundary conditions can be applied.

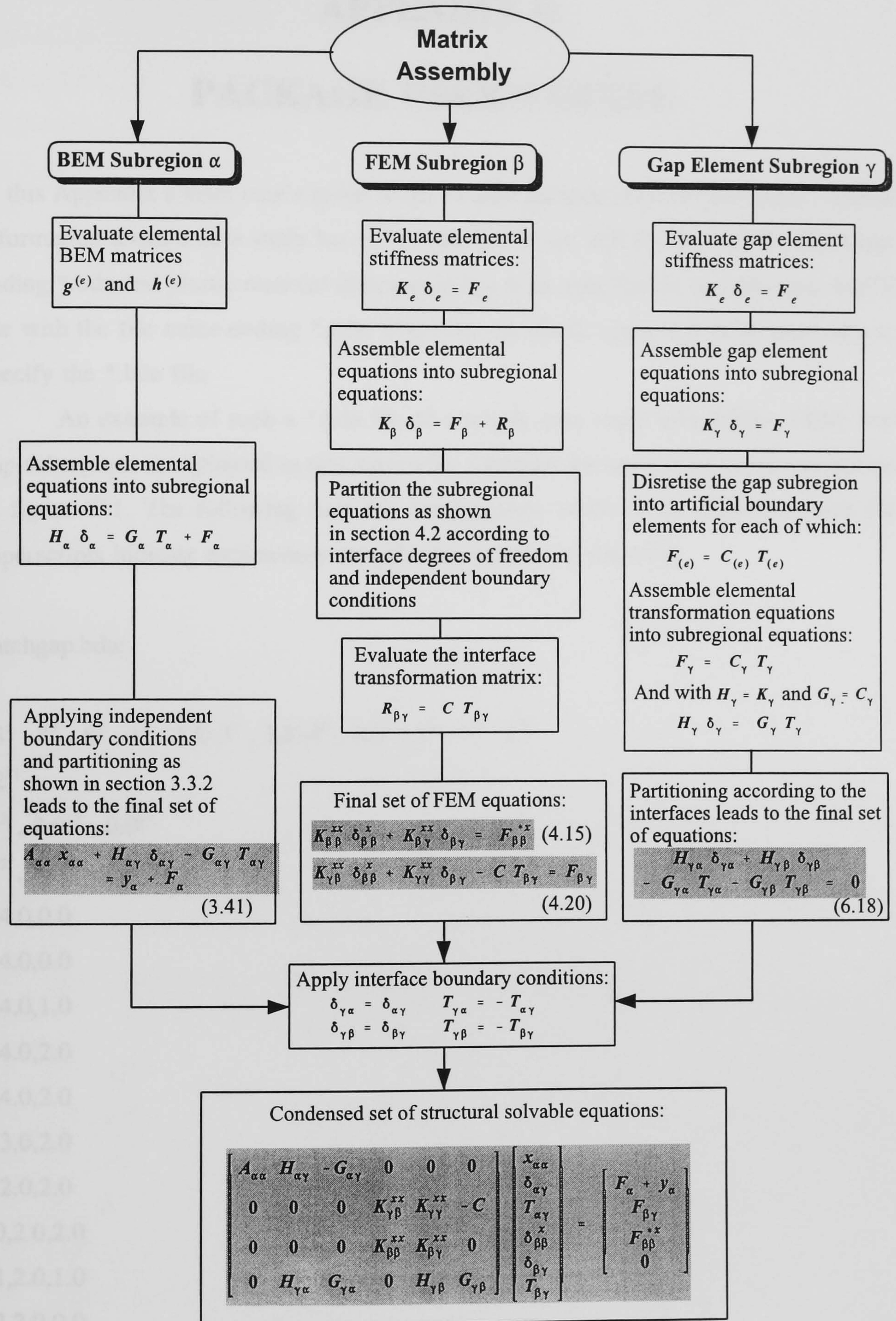


Figure F.1 Assembly of the subregional equations into structural equations.

APPENDIX G

PACKAGE USER'S GUIDE

In this Appendix a brief user's guide to the created package shall be presented. General information about a case study has to be specified in an ASCII-file with the file name ending *.bda and plastic material information has to be specified in an additional ASCII file with the file name ending *.pda. Therefore for elastic cases it is only necessary to specify the *.bda file.

An example of such a *.bda file of a simple case study with BEM-, FEM- and gap subregions is explained in this Appendix. Consider the case of the patch test shown in figure G.1. The following data file patchgap.bda needs to be defined (where the superscripts indicate explanatory remarks listed after the data file):

patchgap.bda:

18¹ , 8² , 8³ , 1⁴ , 1.E-3⁵ , 1.E-4⁶ , 0.0⁷ , 1⁸ , 1⁹ , 1¹⁰
12¹¹
1¹² , 2.0¹³ , 0.0¹⁴
2¹² , 3.0¹³ , 0.0¹⁴
3,4.0,0.0
4,4.0,0.0
5,4.0,1.0
6,4.0,2.0
7,4.0,2.0
8,3.0,2.0
9,2.0,2.0
10,2.0,2.0
11,2.0,1.0
12,2.0,0.0
6¹⁵
13¹⁶ , 2.0¹⁷ , 0.0¹⁸

14^{16} , 2.0^{17} , 1.0^{18}
 15,2.0,2.0
 16,2.0,2.0
 17,2.0,1.0
 18,2.0,0.0
 4^{19} , $1.E6^{20}$, 0^{21}
 1^{22} , 3^{23} , 1^{24} , 2^{24} , 3^{24}
 2^{22} , 3^{23} , 4^{24} , 5^{24} , 6^{24}
 3,3,7,8,9
 4,3,10,11,12
 $1.E-4^{25}$, 0^{26} , 0^{27}
 1^{28} , 1^{29} , 1^{30}
 $1.E-8^{31}$, $1.E13^{32}$, $1.E13^{33}$
 2^{34}
 1^{35} , 3^{36} , 13^{37} , 14^{37} , 15^{37}
 2^{35} , 3^{36} , 16^{37} , 17^{37} , 18^{37}
 0^{38} , 1^{39} , 3^{40}
 1^{41} , 0^{42} , 100.0^{43} , 1^{44} , 3^{45} , 4^{46} , 5^{46} , 6^{46}
 1^{47} , 10^{48} , 15^{49}
 2^{47} , 11^{48} , 14^{49}
 3,12,13,0
 1^{50} , 0.3^{51}
 1^{52} , 100^{53} , 0.01^{54} , 0.1^{55} , 1^{56}
 2^{57}
 1^{58} , $3.^{59}$, 0.5^{60}
 2^{58} , $3.^{59}$, $1.^{60}$
 0^{61} , 0^{62} , 0^{63}
 1^{64} , 0^{65} , 0^{66}
 1^{67} , 1.0^{68} (end of BE data)
 1^{69} , $1.E6^{70}$, 0.3^{71} , 0.0^{72} , $0.1E-2^{73}$
 8^{74} , 1^{75} , 0^{76}
 1^{77} , 0.0^{78} , 0.0^{79}

2^{77} , 1.0^{78} , 0.0^{79}

3,2.0,0.0

4,2.0,1.0

5,2.0,2.0

6,1.0,2.0

7,0.0,2.0

8,0.0,1.0

1^{80} , 3^{81} , 1.0^{82} , 8^{83} , 1^{84} , 2^{84} , 3^{84} , 4^{84} , 5^{84} , 6^{84} , 7^{84} , 8^{84}

0^{85} , 2^{86}

1^{87} , 1^{88} , 0.0^{89} , 1^{90} , 3^{91} , 1^{92} , 7^{92} , 8^{92}

2^{87} , 1^{88} , 0.0^{89} , 2^{90} , 1^{91} , 1^{92}

3^{93} , 1^{94} , 0^{95}

1^{96} , 1^{97} , 1^{98} , 3^{99} , 18^{100}

2^{96} , 1^{97} , 1^{98} , 4^{99} , 17^{100}

3,1,1,5,16

1^{101} , 1^{102} , 3^{103} , 18^{104} , 17^{104} , 16^{104}

0^{105}

0^{106}

0^{107}

0^{108}

0^{109}

0^{110}

BEM and gap element related data:

General information:

- 1 Total number of boundary nodes on BEM- and gap element subregions
- 2 Number of Gauss quadrature points for boundary elements
- 3 Number of singular integration points for boundary elements
- 4 Number of half of Finite Difference Method intervals (only relevant for body loads)

- 5 (Distance from boundary if source points are to be considered outside the BEM-boundary. Not relevant for this work. Always set to 0.0)
- 6 Maximum admissible error for most of the iterative procedures and result evaluations in the BEM package
- 7 (Max. distance under which source point is considered a boundary point. Not relevant to this work. Always set any positive value)
- 8 Number of BEM subregions
- 9 Number of gap element subregions
- 10 Number of FEM subregions

Nodal information

- 11 Number of nodes on BEM subregion (if there is more than one BEM subregion defined 11-14 have to be repeated for each subregion)
- 12 Node number
- 13 x-coordinate
- 14 y-coordinate
- 15 Number of nodes on gap element subregion (if there is more than one gap element subregion defined 11-14 have to be repeated for each subregion)
- 16 Node number (numbering of nodes on gap element subregion continues after BEM subregion numbering)
- 17 x-coordinate
- 18 y-coordinate

Elemental information

- 19 Number of boundary elements of BEM subregion (19-24 have to be repeated for each BEM subregion)
- 20 Young's modulus of material of the subregion
- 21 = 0 for manually generated mesh data
> 0 for automatic mesh generation
- 22 Element number

- 23 Number of nodes of element
- 24 Node numbers of the nodes of the boundary element in anticlockwise direction

Contact information

- 25 Maximum admissible error within the contact subincrement iterations
- 26 = 0 for frictionless contact
= 1 for frictional contact (for this case some more data than demonstrated in this example needs to be read.
- 27 = 0 if there are no gap element subincrement iterations desired.
= 1 for gap element subincrement iterations (e.g. for non-conforming and frictional contact)
- 28 Initial gap condition of the gap element with the lowest node number (i.e. here the gap element 13-18 because 13 is the lowest node number in the gap element subregion.
= 0 the gap element is opened (no contact)
= 1 the gap element is closed (contact)
- 29 Initial gap condition of the gap element with the second lowest node number
- 30 Initial gap condition of the gap element with the third lowest node number
- 31 Normal and tangential gap stiffness of an opened gap element (should be very low in order to allow rigid body displacements. Not relevant for this case study)
- 32 Normal gap stiffness of a closed gap element (should be very high)
- 33 Tangential gap stiffness of a closed gap element for frictionless contact (should be a very small number, but for this case study the relative tangential displacements are to be restrained)
- 34 Number of artificial boundary elements on gap subregion
- 35 Artificial boundary element number
- 36 Number of nodes on artificial boundary element
- 37 Global node numbers of the nodes of the artificial boundary element in anticlockwise direction

Boundary conditions and BEM/BEM (and BEM/gap) interface conditions

- 38 Number of boundary condition sets with only one boundary condition
- 39 Number of boundary condition sets with more than one boundary condition
- 40 Number of interface node pairs on BEM-BEM interfaces or BEM-Gap subregion interfaces
- 41 Number of boundary condition set
- 42 = 0 for prescribed traction
 = 1 for prescribed displacement
- 43 Value of prescribed boundary condition
- 44 = 1 for boundary condition in x-direction
 = 2 for boundary condition in y-direction
- 45 Number of nodes with the defined boundary condition
- 46 Structural node numbers of the nodes with the defined boundary condition
- 47 Node pair number
- 48 Node 1 of node pair
- 49 Node 2 of node pair

Other information

- 50 = 1 for plane stress
 = 2 for plane strain
- 51 Poisson's ratio
- 52 (This number has no meaning)
- 53 (Maximum number of ξ iterations)
- 54 Max. distance within which the point is considered on the boundary
- 55 Min. distance greater than which the point is considered an internal node
- 56 = positive for finite domain analysis
 = negative for infinite domain analysis
- 57 Number of internal points at which results are to be obtained
- 58 Internal point number
- 59 x-coordinate of the internal point

- 60 y-coordinate of the internal point
- 61 > 0 for body forces to be defined
- 62 = for no centrifugal loadings
- = 1 for centrifugal loading about z-axis
- = 2 for rotating inertia about an axis in the x-y plane
- 63 > 0 for cases with thermal loading
- 64 Number of main increments r
- 65 = 0 for manual incrementation
- = 1 for automatic division into r equal main increments
- 66 = 0 Directional cosines and coordinates are not updated between the increments
- = 1 Directional cosines are updated
- = 2 Coordinates are updated
- = 3 Directional cosines and coordinates are updated
- 67 For manual incrementation: Number of load increment
- 68 For manual incrementation: Fraction of total load

FEM related data:

General information

- 69 = 1 for plane stress
- = 2 for plane strain
- 70 Young's modulus
- 71 Poisson's ratio
- 72 (Coefficient of thermal expansion)
- 73 Material density
- 74 Number of nodes
- 75 Number of elements
- 76 Number of Gaussian integration points (= 0 for automatic selection)

Nodal information

- 77 Node number (starting from 1 for FEM nodes)
- 78 x-coordinate
- 79 y-coordinate

Elemental information

- 80 Element number
- 81 Element type
- 82 Thickness of the element
- 83 Number of nodes on element
- 84 Node number of the nodes on the element (in anticlockwise direction)

Boundary conditions

- 85 Number of boundary condition sets with only one boundary condition defined
- 86 Number of boundary condition sets with more than one boundary condition defined
- 87 Boundary condition set number
- 88 = 1 for prescribed displacement
- 89 Value of prescribed boundary condition
- 90 = 1 for boundary condition in x-direction
= 2 for boundary condition in y-direction
- 91 Number of boundary conditions in boundary condition set
- 92 Nodes of the boundary condition set

FEM/BEM (and FEM/gap) interface data

- 93 Number of node pairs on FEM/BEM- and FEM/Gap- interfaces
- 94 Number of boundary elements on FEM/BEM- and FEM/Gap- interfaces
- 95 Number of (external and internal) corner nodes on FEM/BEM- and FEM/Gap

interfaces

- 96 Node pair number
- 97 Number of boundary element or gap element subregion nodes paired with the FEM node (=1 for normal interface node pair, =2 for connection of an FEM node to two BEM corner nodes)
- 98 Interface node pair type
 - = 1 for a normal node-to-node type interface pair
 - = 2 for an interface corner inside the body (internal interface corner)
 - = 3 for an interface corner on the edge of the body (external interface corner)
- 99 FEM node number
- 100 BEM node number of the connected BEM node (to be followed by the second BEM node for an interface corner type node pair)
- 101 Number of interface boundary element
- 102 Interface boundary element type
 - = 1 for all interface elements except
 - = 3 for the case of an external corner node as described in section 4.5.1 *case a*.
(The boundary element of type 3 is not on the actual interface but on the boundary with the prescribed displacement)
- 103 Number of nodes on boundary element
- 104 BEM global node numbers of the boundary element nodes of the interface boundary element (in clockwise direction of the BEM subregion)

Information on the applied forces

- 105 Number of nodal forces on FEM nodes
- 106 Number of pressurised finite element sides
- 107
 - = 0 for cases without internal loading
 - =1 for centrifugal loading about z-axis
 - =2 for rotating inertia about an axis in the x-y plane
 - =3 for body forces