

PERFORMANCE OF HIGHER ORDER FINITE ELEMENTS

by

Ebru Harmandar

B.S. in C.E., Yıldız Technical University, 1999

Submitted to the Institute for Graduate Studies in
Science and Engineering in partial fulfillment of
the requirements for the degree of

Master of Science

in

Civil Engineering

Bogazici University Library



39001101671736

14

Boğaziçi University

2002

ACKNOWLEDGMENTS

I would like to express the very most thanks to my thesis supervisor, Prof. Dr. Semih S. Tezcan for his invaluable guidance, continuous support, helpful suggestions and everlasting encouragement. He has always been supportive for the preparation of this thesis. He and his mother, Müveddet Tezcan, always gave useful advice about life that I could never learn from anybody else.

My sincere gratitude is also due to my advisory committee Prof. Dr. Gülay Altay and Prof. Dr. Özal Yüzügüllü for their useful suggestions and comments. My thanks are also due to Dr. Hafez Keypour for his assistance in computational problems.

I would also like to thank to my parents for their understanding and loving care. Also, my thanks are due to Filiz and Gökhan Kobal for their devotion.

Heartfelt thanks are due to my husband Buğra Harmandar for his never ending help and understanding. My extreme thanks are also due to my sister Didem Turgut for her continuous support and endless love.

I would like to thank also Muhterem Sadunoğlu, Burcu Harmandar, Adalet Kobal and Bahar Kocaer for their encouragement and Ömer Faruk Tıǵlı for his kind assistance during the preparation of the thesis.

ABSTRACT

PERFORMANCE OF HIGHER ORDER FINITE ELEMENTS

This study is focused on the evaluation of the performance of the higher order finite elements for the solution of basic and complex problems. Performances of higher order finite elements are compared with those of the regular order elements. Results of this comparison revealed the efficiency and accuracy of higher order finite elements. Following a brief discussion of the finite element method; isoparametric elements, higher order elements, and shape functions are explained. For the purpose of illustration, four different case studies are investigated in a comparative fashion.

Firstly, a cantilever deep beam under a concentrated force at the tip is selected as a plane stress problem and is analyzed using five different types of finite elements and in each case the mesh size is gradually decreased. Second problem consists of a circular plate, which is analyzed using two different types of regular and higher order finite elements. The third and the fourth plate bending problems are the fixed boundary square plates, under uniform and central point loads, modeled by the same four different types of finite elements. The analyses of all these four problems are handled by the LUSAS package program.

The purpose of utilizing higher order finite elements is to obtain as accurate solutions as possible, with as little number of degrees of freedom as possible. It is demonstrated in all four case studies that the accuracies of higher order finite element were much superior than those of the regular finite elements.

ÖZET

YÜKSEK MERTEBELİ SONLU ELEMANLARIN PERFORMANSI

Bu çalışma, basit ve kompleks problemlerin çözümünde, yüksek mertebeli sonlu elemanların performanslarının değerlendirilmesini incelemiştir. Yüksek mertebeli sonlu elemanların performansları basit elemanlarıinkiyle karşılaştırılmıştır. Sonlu elemanlar metodunun kısaca anlatımının ardından, izoparametrik elemanlar, yüksek mertebeli elemanlar ve şekil fonksiyonları detaylı bir biçimde anlatılmıştır. Değerlendirme için dört değişik problem kıyaslanarak incelenmiştir.

İlk olarak, uç tekil yükü altındaki konsol kiriş, düzlem gerilme problemi olarak seçilmiştir ve beş farklı sonlu elemanın sayılarının kademeli olarak artırılmasıyla analizi yapılmıştır. İkinci problem; basit ve yüksek mertebeli sonlu elemanların iki değişik tipi için incelenen dairesel plaktan oluşmaktadır. Bu dört sonlu elemanla düzgün yayılı yük ve orta nokta tekil yükü altında üçüncü ve dördüncü problem kare plak olarak modellenmiştir. Bu dört problemin analizi LUSAS yazılımında yapılmıştır.

Yüksek mertebeli sonlu elemanların kullanılmasının amacı en az serbestlik derecesiyle en doğru çözümler elde edebilmektir. Dört problemde de gösterildiği gibi yüksek mertebeli sonlu elemanların doğruluğu basit sonlu elemanlarıkinden çok daha iyidir.

TABLE OF CONTENTS

ACKNOWLEDGMENTS	iii
ABSTRACT	iv
ÖZET	v
LIST OF FIGURES	ix
LIST OF TABLES	xviii
LIST OF SYMBOLS	xxi
1. THE FINITE ELEMENT METHOD	1
1.1. Basic Concept	1
1.2. Historical Background	2
1.3. Engineering Applications	3
2. FUNDAMENTALS OF THE FINITE ELEMENT	6
2.1. Virtual Work Principle	6
2.2. Geometric Shapes of Finite Elements	12
2.3. Automatic Mesh Generation	15
2.4. Convergence Criteria	17
3. SHAPE FUNCTIONS	18
3.1. Introduction	18
3.2. One Dimensional Linear Element	18
3.2.1. Shape Function of One Dimensional Linear Element	18
3.2.2. Stiffness Matrix of One Dimensional Linear Element	20
3.3. Triangular Element	21
3.3.1. Shape Function of a Constant Stress Triangle	21
3.3.2. Stiffness Matrix of the Constant Stress Triangle	23
3.4. Rectangular Element in Plane Stress	26
3.4.1. Shape Function of a Rectangular Element	26
3.3.2. Stiffness Matrix of the Rectangular Element	28
4. HIGHER ORDER FINITE ELEMENTS	31
4.1. Introduction	31
4.2. Pascal Polynomials	33

4.3. One Dimensional Higher Order Elements.....	36
4.4. One Dimensional Lagrange Polynomials.....	39
4.5. Higher Order Triangular Elements	41
4.6. Higher Order Rectangular Elements	42
4.7. Tetrahedral Element.....	45
4.8. Serendipity Hexahedral Element	47
5. CASE STUDY NO. 1 (Plane Stress Problem).....	49
5.1. Data for the Plane Stress Problem.....	49
5.2. The Exact Solutions	50
5.3. Finite Element Types used in the Analyses.....	52
5.4. Discussion of Results	54
6. CASE STUDY NO. 2 (Circular Plate Bending Problem)	67
6.1. Data for the Plate Bending Problem.....	67
6.2. The Exact Solutions	67
6.3. Finite Element Types used in the Analyses.....	69
6.4. Discussion of Results	71
7. CASE STUDY NO. 3 (Rectangular Plate Bending Problem under Central Load).....	78
7.1. Data for the Square Plate Bending Problem.....	78
7.2. The Exact Solutions	79
7.3. Finite Element Types used in the Analyses.....	81
7.4. Discussion of Results	82
8. CASE STUDY NO. 4 (Rectangular Plate Bending Problem under Uniform Load)...	88
8.1. Data for the Square Plate Bending Problem.....	88
8.2. The Exact Solutions	89
8.3. Finite Element Types used in the Analyses.....	90
8.4. Discussion of Results	90
9. CONCLUSIONS	97
APPENDIX A. ENHANCED STRAIN ELEMENTS	98
APPENDIX B. MESH SIZES OF PLANE STRESS PROBLEM	106
APPENDIX C. ISOFLEX THIN PLATE ELEMENT (QF4, TF3).....	182
APPENDIX D. ISOPARAMETRIC THICK MINDLIN PLATE (QTF8, TTF6)	186
APPENDIX E. MESH SIZES OF CIRCULAR PLATE BENDING PROBLEM	189

APPENDIX F. RECTANGULAR PLATE BENDING PROBLEM UNDER CENTRAL LOAD	194
APPENDIX G. MESH SIZES OF RECTANGULAR PLATE BENDING PROBLEM UNDER UNIFORM LOAD	201
REFERENCES.....	206
REFERENCES NOT CITED	209

LIST OF FIGURES

Figure 1.1. Representation of finite elements	2
Figure 2.1. Number of degrees of freedom per node	8
Figure 2.2. Two-dimensional finite elements	12
Figure 2.3. Three-dimensional finite elements	13
Figure 2.4. Axisymmetric elements	13
Figure 2.5. Curved finite elements	14
Figure 2.6. Uniform stress criterion	17
Figure 2.7. Rigid body criterion	17
Figure 3.1. One-dimensional linear element	19
Figure 3.2. Area coordinates of a triangle	21
Figure 3.3. Degrees of freedom of a triangle	24
Figure 3.4. Constant stress triangle	25
Figure 3.5. The shape functions of a rectangle	26
Figure 3.6. Rectangular element	28
Figure 4.1. Isoparametric, subparametric and superparametric elements	32

Figure 4.2. Pascal polynomials for two-dimensional problems.....	34
Figure 4.3. Pascal polynomials for three-dimensional problems.....	35
Figure 4.4. Quadratic line element	37
Figure 4.5. Location of nodes for one-dimensional higher order element	39
Figure 4.6. Shape function of quadratic linear element for node 1.....	39
Figure 4.7. Shape function of quadratic linear element for node 2.....	39
Figure 4.8. Nodal numbering in higher order triangles	41
Figure 4.9. Shape function of a linear rectangular element	42
Figure 4.10. Serendipity shape functions of a quadratic element	42
Figure 4.11. Lagrangian shape functions of a quadratic element	43
Figure 4.12. Serendipity shape functions of a cubic element	44
Figure 4.13. Lagrangian shape functions of a cubic element	44
Figure 4.14. Nodal numbering in a tetrahedron	45
Figure 4.15. Natural coordinates for tetrahedron	46
Figure 4.16. Natural coordinates for hexahedron.....	47
Figure 4.17. Nodal numbering in a Serendipity hexahedron.....	48
Figure 5.1. Cantilever beam with a point load at the tip	49

Figure 5.2. Maximum normal stress σ_x , at Section II-II'	64
Figure 5.3. Maximum normal stress σ_y , at Section II-II'	65
Figure 5.2. Maximum shear stress τ_{xy} , at Section II-II'	66
Figure 6.1. Clamped circular plate	67
Figure 6.2. Vertical deflection w , at the center of the plate.....	74
Figure 6.3. Moment M_r , at the boundary of the plate.....	75
Figure 6.4. Moment M_t , at the boundary of the plate.....	76
Figure 6.5. Moment (M_r , M_t), at the boundary of the plate	77
Figure 7.1. Square plate under central load	78
Figure 7.2. Bending moment at the center of the plate	80
Figure 7.3. Vertical deflection w , at the center of the plate.....	85
Figure 7.4. Moment M_y , at the boundary of the plate	86
Figure 7.5. Moment (M_x , M_y), at the center of the plate	87
Figure 8.1. Square plate under central load	88
Figure 8.2. Vertical deflection w , at the center of the plate.....	94
Figure 8.3. Moment (M_x , M_y), at the boundary of the plate	95
Figure 8.4. Moment (M_x , M_y), at the center of the plate	96

Figure B.1. Mesh sizes of plane stress elements (QPM4, QPM4M)	107
Figure B.2. Mesh sizes of plane stress element (QPM8).....	108
Figure B.3. Mesh sizes of plane stress element (TPM3).....	109
Figure B.4. Mesh sizes of plane stress element (TPM6).....	110
Figure B.5. Comparison of normal stress (σ_x) at section II-II' (2/4 mesh size)	111
Figure B.6. Comparison of normal stress (σ_x) at section II-II' (4/8 mesh size)	112
Figure B.7. Comparison of normal stress (σ_x) at section II-II' (16/32 mesh size)	113
Figure B.8. Comparison of normal stress (σ_x) at section II-II' (40/80 mesh size)	114
Figure B.9. Maximum normal stress (σ_y) at section II-II'	115
Figure B.10. Comparison of normal stress (σ_y) at section II-II' (2/4 mesh size)	116
Figure B.11. Comparison of normal stress (σ_y) at section II-II' (4/8 mesh size)	117
Figure B.12. Comparison of normal stress (σ_y) at section II-II' (16/32 mesh size).....	118
Figure B.13. Comparison of normal stress (σ_y) at section II-II' (40/80 mesh size).....	119
Figure B.14. Maximum shear stress (τ_{xy}) at section II-II'	120
Figure B.15. Comparison of shear stress (τ_{xy}) at section II-II' (2/4 mesh size).....	121
Figure B.16. Comparison of shear stress (τ_{xy}) at section II-II' (4/8 mesh size).....	122

Figure B.17. Comparison of shear stress (τ_{xy}) at section II-II' (16/32 mesh size).....	123
Figure B.18. Comparison of shear stress (τ_{xy}) at section II-II' (40/80 mesh size).....	124
Figure B.19. Maximum normal stress (σ_x) at section III-III'	125
Figure B.20. Comparison of normal stress (σ_x) at section III-III' (2/4 mesh size).....	126
Figure B.21. Comparison of normal stress (σ_x) at section III-III' (4/8 mesh size).....	127
Figure B.22. Comparison of normal stress (σ_x) at section III-III' (16/32 mesh size).....	128
Figure B.23. Comparison of normal stress (σ_x) at section III-III' (40/80 mesh size).....	129
Figure B.24. Maximum normal stress (σ_y) at section III-III'	130
Figure B.25. Comparison of normal stress (σ_y) at section III-III' (2/4 mesh size).....	131
Figure B.26. Comparison of normal stress (σ_y) at section III-III' (4/8 mesh size).....	132
Figure B.27. Comparison of normal stress (σ_y) at section III-III' (16/32 mesh size).....	133
Figure B.28. Comparison of normal stress (σ_y) at section III-III' (40/80 mesh size).....	134
Figure B.29. Maximum shear stress (τ_{xy}) at section III-III'	135
Figure B.30. Comparison of shear stress (τ_{xy}) at section III-III' (2/4 mesh size).....	136
Figure B.31. Comparison of shear stress (τ_{xy}) at section III-III' (4/8 mesh size).....	137
Figure B.32. Comparison of shear stress (τ_{xy}) at section III-III' (16/32 mesh size).....	138

Figure B.33. Comparison of shear stress (τ_{xy}) at section III-III' (40/80 mesh size)	139
Figure B.34. Maximum strain (ϵ_x) at section II-II'	140
Figure B.35. Comparison of strain (ϵ_x) at section II-II' (2/4 mesh size).....	141
Figure B.36. Comparison of strain (ϵ_x) at section II-II' (4/8 mesh size).....	142
Figure B.37. Comparison of strain (ϵ_x) at section II-II' (16/32 mesh size).....	143
Figure B.38. Comparison of strain (ϵ_x) at section II-II' (40/80 mesh size).....	144
Figure B.39. Maximum strain (ϵ_y) at section II-II'	145
Figure B.40. Comparison of strain (ϵ_y) at section II-II' (2/4 mesh size).....	146
Figure B.41. Comparison of strain (ϵ_y) at section II-II' (4/8 mesh size).....	147
Figure B.42. Comparison of strain (ϵ_y) at section II-II' (16/32 mesh size).....	148
Figure B.43. Comparison of strain (ϵ_y) at section II-II' (40/80 mesh size).....	149
Figure B.44. Shear strain (γ_{xy}) at section II-II'	150
Figure B.45. Comparison of shear strain (γ_{xy}) at section II-II' (2/4 mesh size).....	151
Figure B.46. Comparison of shear strain (γ_{xy}) at section II-II' (4/8 mesh size).....	152
Figure B.47. Comparison of shear strain (γ_{xy}) at section II-II' (16/32 mesh size).....	153
Figure B.48. Comparison of shear strain (γ_{xy}) at section II-II' (40/80 mesh size).....	154

Figure B.49. Maximum strain (ϵ_x) at section III-III'	155
Figure B.50. Comparison of strain (ϵ_x) at section III-III' (2/4 mesh size)	156
Figure B.51. Comparison of strain (ϵ_x) at section III-III' (4/8 mesh size)	157
Figure B.52. Comparison of strain (ϵ_x) at section III-III' (16/32 mesh size)	158
Figure B.53. Comparison of strain (ϵ_x) at section III-III' (40/80 mesh size)	159
Figure B.54. Maximum strain (ϵ_y) at section III-III'	160
Figure B.55. Comparison of strain (ϵ_y) at section III-III' (2/4 mesh size)	161
Figure B.56. Comparison of strain (ϵ_y) at section III-III' (4/8 mesh size)	162
Figure B.57. Comparison of strain (ϵ_y) at section III-III' (16/32 mesh size)	163
Figure B.58. Comparison of strain (ϵ_y) at section III-III' (40/80 mesh size)	164
Figure B.59. Shear strain (γ_{xy}) at section III-III'	165
Figure B.60. Comparison of shear strain (γ_{xy}) at section III-III' (2/4 mesh size)	166
Figure B.61. Comparison of shear strain (γ_{xy}) at section III-III' (4/8 mesh size)	167
Figure B.62. Comparison of shear strain (γ_{xy}) at section III-III' (16/32 mesh size)	168
Figure B.63. Comparison of shear strain (γ_{xy}) at section III-III' (40/80 mesh size)	169
Figure B.64. Horizontal deflection (u) at section I-I'	170

Figure B.65. Comparison of horizontal deflection (u) at section I-I' (2/4 mesh size)	171
Figure B.66. Comparison of horizontal deflection (u) at section I-I' (4/8 mesh size)	172
Figure B.67. Comparison of horizontal deflection (u) at section I-I' (16/32 mesh size) .	173
Figure B.68. Comparison of horizontal deflection (u) at section I-I' (40/80 mesh size) .	174
Figure B.69. Horizontal deflection (u) at section II-II'	175
Figure B.70. Comparison of horizontal deflection (u) at section II-II' (2/4 mesh size)...	176
Figure B.71. Comparison of horizontal deflection (u) at section II-II' (4/8 mesh size)...	177
Figure B.72. Comparison of horizontal deflection (u) at section II-II' (16/32 mesh size)	178
Figure B.73. Comparison of horizontal deflection (u) at section II-II' (40/80 mesh size)	179
Figure B.74. Vertical deflection (u) at section I-I'	180
Figure B.75. Vertical deflection (u) at section II-II'	181
Figure E.1. Mesh sizes of plate bending element (TF3) (Case Study No.2)	190
Figure E.2. Mesh sizes of plate bending element (TTF6) (Case Study No.2).....	191
Figure E.3. Mesh sizes of plate bending element (QF4) (Case Study No.2).....	192
Figure E.4. Mesh sizes of plate bending element (QTF8) (Case Study No.2)	193
Figure F.1. Mesh sizes of plate bending element (TF3) (Case Study No.3)	195

Figure F.2. Mesh sizes of plate bending element (TTF6) (Case Study No.3).....	196
Figure F.3. Mesh sizes of plate bending element (QF4) (Case Study No.3).....	197
Figure F.4. Mesh sizes of plate bending element (QTF8) (Case Study No.3).....	198
Figure G.1. Mesh sizes of plate bending element (TF3) (Case Study No.4).....	202
Figure G.2. Mesh sizes of plate bending element (TTF6) (Case Study No.4)	203
Figure G.3. Mesh sizes of plate bending element (QF4) (Case Study No.4)	204
Figure G.4. Mesh sizes of plate bending element (QTF8) (Case Study No.4).....	205

LIST OF TABLES

Table 1.1. Field of finite Element Method.....	5
Table 3.1. Stiffness matrix of a constant stress triangle.....	27
Table 3.2. Stiffness matrix of a rectangular plane stress element.....	30
Table 4.1. Lagrange shape functions for one-dimensional elements.....	40
Table 5.1. Stresses and strains at Section II-II' and III-III'.....	52
Table 5.2. Deflections at Section I-I' and II-II'.....	52
Table 5.3. Plane stress elements used in the solution.....	53
Table 5.4. Mesh sizes of plane stress elements.....	54
Table 5.5. Comparison of maximum normal stress σ_x , at II-II' (kN/m^2).....	55
Table 5.6. Comparison of stress σ_y , at II-II' (kN/m^2).....	56
Table 5.7. Comparison of shear stress τ_{xy} , at II-II' (kN/m^2).....	56
Table 5.8. Comparison of maximum normal stress σ_x , at III-III' (kN/m^2).....	57
Table 5.9. Comparison of stress σ_y , at III-III' (kN/m^2).....	57
Table 5.10. Comparison of shear stress τ_{xy} , at III-III' (kN/m^2).....	58
Table 5.11. Comparison of strain ϵ_x , at II-II'.....	58

Table 5.12. Comparison of strain ϵ_y , at II-II'	59
Table 5.13. Comparison of shear strain τ_{xy} , at II-II'	59
Table 5.14. Comparison of strain ϵ_x , at III-III'	60
Table 5.15. Comparison of strain ϵ_y , at III-III'	60
Table 5.16. Comparison of shear strain τ_{xy} , at III-III'	61
Table 5.17. Comparison of maximum horizontal deflection at I-I' (m).....	61
Table 5.18. Comparison of maximum vertical deflection at I-I' (m).....	62
Table 5.19. Comparison of maximum horizontal deflection at II-II' (m)	62
Table 5.20. Comparison of maximum vertical deflection at II-II' (m)	63
Table 6.1. Exact values of the vertical deflection and bending moments	68
Table 6.2. Plate bending elements used in the solution.....	70
Table 6.3. Mesh sizes of plate bending elements.....	70
Table 6.4. Comparison of vertical deflection w , at the center (m).....	72
Table 6.5. Comparison of moment M_r , at the boundary (kNm).....	72
Table 6.6. Comparison of moment M_t , at the boundary (kNm).....	73
Table 6.7. Comparison of moment (M_r , M_t), at the boundary (kNm).....	73
Table 7.1. Vertical deflection and bending moments.....	81

Table 7.2. Mesh sizes of plate bending elements..... 81

Table 7.3. Comparison of vertical deflection w , at the center (m)..... 83

Table 7.4. Comparison of moment M_y , at the boundary (kNm) 83

Table 7.5. Comparison of moment (M_x, M_y), at the center (kNm) 84

Table 8.1. Vertical deflection and bending moments..... 89

Table 8.2. Mesh sizes of plate bending elements..... 90

Table 8.3. Comparison of vertical deflection w , at the center (m)..... 92

Table 8.4. Comparison of moment (M_x, M_y), at the boundary (kNm)..... 92

Table 8.5. Comparison of moment (M_x, M_y), at the center (kNm) 93

LIST OF SYMBOLS

c	Coefficient of viscosity (<i>Force-second / Length</i>)
$[C]$	Consistent damping matrix
$\{d\}$	Nodal displacement vector
$\{\dot{d}\}$	Nodal velocity vector
$\{\ddot{d}\}$	Nodal acceleration vector
$[D]$	Material elasticity matrix
$\{D\}_T$	Temperature elasticity matrix
E	Modulus of elasticity
$[E]$	Material elasticity matrix
$\{f\}_s$	Edge loads at nodes
$\{f\}_T$	Temperature change loads
$\{f\}_X$	Body forces at nodes
$\{f\}_{\epsilon_0}$	Nodal loads due to initial strains
$\{f\}_{\sigma_0}$	Nodal loads due to initial stresses
G	Modulus of rigidity
h	Height of the beam
I	Moment of inertia
$[k]$	Stiffness matrix
L	Length of the beam
$[m]$	Consistent mass matrix
M_r	Moment in radial direction
M_t	Moment in circumferential direction
M_x	Moment in x-direction
M_y	Moment in y-direction
N_i	Shape function for d_i
$[N]$	Shape function matrix
$\{p\}_s$	External loads acting along the boundary

P	Concentrated load
q	Uniform load
$[S]$	Stress matrix
t	Thickness
$\{u\}$	Generic displacement vector
$\{\dot{u}\}$	Generic velocity vector
$\{\ddot{u}\}$	Generic acceleration vector
$\{u\}_s$	External displacement vector
U_e	Strain Energy
∂U_e	Increase in strain energy
$\{X\}$	Body force vector
W_e	External work
∂W_e	Increase in external work
α_i	Coefficient of thermal expansion
γ	Shear strain
$\{\underline{\Delta}\}$	Operator matrix to differentiate the shape functions
ΔT	Change in temperature
ε	Strain
ε_c	Compatible strain
ε_e	Enhanced strain
$\{\varepsilon\}$	Strain vector
$\{\varepsilon\}_o$	Initial strain vector
ν	Poisson's ratio
Π_e	Total potential energy
$\partial \Pi_e$	Variation in potential energy
ρ	Mass density per unit volume
σ	Stress
$\{\sigma\}$	Stress vector
$\{\sigma\}_o$	Initial stress vector
τ	Shear stress

1. THE FINITE ELEMENT METHOD

1.1. Basic Concept

The finite element method is a numerical method which can be used for the accurate solution of complex engineering problems. By the development of the method, the potentialities of the method for the solution of different types of applied science and engineering problems were recognized. Over the years, the finite element technique has been so well established. The method becomes popular in different fields of engineering because it can be used for the solution of any problem simply by changing the input data.

The basic idea in the finite element method is to find the solution of a complicated problem by replacing it with a simpler one. Since the actual problem is substituted for a simpler one in finding the solution, it is possible to find only an approximate solution rather than the exact solution. Also, it will often be possible to improve or refine the estimated solution by computational effort. In the finite element method, the solution region is considered as built up of many small subregions named finite elements. These elements are considered to be interconnected at specified joints which are called nodes or nodal points. The nodes usually lie on the element boundaries where adjacent elements are considered to be connected. Since the actual variation of the field variable (like displacement, stress, temperature, pressure or velocity) inside the continuum is not known, it is assumed that the variation of the field variable inside a finite element can be approximated by a simple function. The approximating functions (also named interpolation models) are defined in terms of the values of the field variables at the nodes. When field equations (like equilibrium equations) for the whole continuum are written, the new unknowns will be the nodal values of the field variables. By solving the field equations, which are generally in the form of matrix equations, the nodal values of the field variable will be known. Once there are known, the approximating functions define the field variable throughout the assemblage of the elements.

An example of how a finite element model represents a complex geometrical shape is shown in Figure 1.1. It is very difficult to find the exact response (stresses and displacements) of the machine under any specified loading condition, this structure is approximated as composed of several pieces as shown in Figure 1.1 in the finite element method. In each piece or element, a convenient approximate solution is assumed and the conditions of overall equilibrium of the structure derived. The satisfaction of these conditions will yield an approximate solution for the displacements and stresses [1].

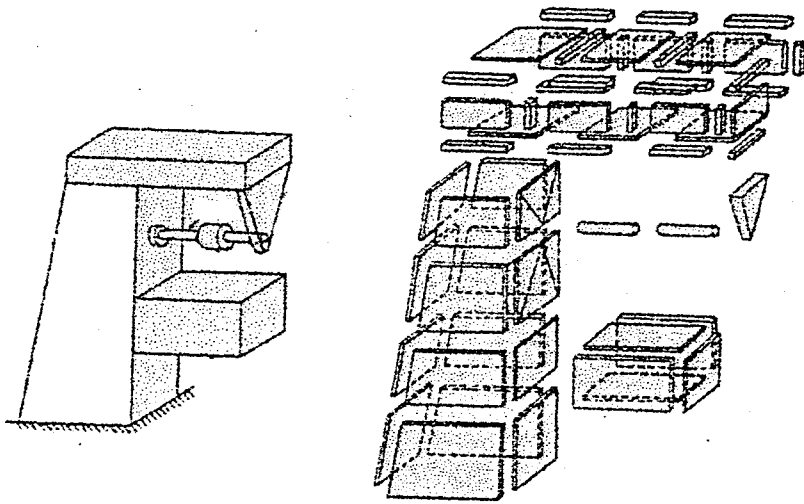


Figure 1.1. Representation of finite elements

1.2. Historical Background

Basic ideas of the finite element method originated from advances in aircraft structural analysis as it is mentioned before [2], [3]. In 1941, Hrenikoff presented a solution of elasticity problems using the frame work method [4]. Courant's paper, which used piecewise polynomial interpolation over triangular subregions to model torsion problems, appeared in 1943 [5]. Turner et al. derived stiffness matrices for truss, beam, and other elements and presented their findings in 1956 [6]. The term finite element was first coined and used by Clough in 1960 [7].

In the early 1960s, engineers used the method for approximate solution of problems in stress analysis, fluid, flow, heat transfer, and other areas. A book by Argyris and Kelsey in 1965 on energy theorems and matrix methods laid a foundation for further developments in finite element studies [8]. The first book on finite elements by Zienkiewicz and Chung was published in 1967 [9]. In the late 1960s and early 1970s, finite element analysis was applied to nonlinear problems and large deformations [2], [3]. Oden's book on nonlinear continua appeared in 1972 [10].

The finite element method is rapidly becoming an essential and integral part for the solution of Medical problems, such as orthopedics, dentistry, etc. Finite element technology consist of a library of element models, a process for combining these models into a mathematical model of an engineering system, and a set of algorithms for numerical solution of equations. This method supported by computer software and by knowledge based on application experiences. The real application of finite element methods requires not only the superiority of the theory but also a significant computer programming effort.

1.3. Engineering Applications

As stated earlier, the finite element method was developed for the analysis of aircraft structures. This method is applicable to a wide variety of boundary value problem in engineering. A boundary value problem is one in which a solution required in the domain of a body subject to the satisfaction of prescribed boundary conditions on the dependent variables or their derivatives. A general list of fields of applications of finite element method is given in Table 1.1 [1]. For each field of application the problem is formulated in anyone of the following types:

- Equilibrium problems
- Eigenvalue problems
- Propagation or transient problems

In an equilibrium problem, it is necessary to find the steady state displacement or stress distribution if it is a solid mechanic problem, temperature or heat flux distribution if

it is a heat transfer problem, and pressure or velocity distribution if it is a fluid mechanics problem.

Eigenvalue problems are sometimes called characteristic value problems and occur in the analysis of homogeneous differential equations [11]. Time will not appear explicitly in eigenvalue problems. These problems can be considered as extensions of equilibrium problems in which critical values of certain constraints are to be determined in addition to the corresponding steady state configurations. The natural frequencies or buckling loads and mode shapes are determined for the solid mechanics or structures problem, stability of laminar flows for the fluid mechanics problem, and resonance characteristics for the electrical circuit problem [12], [13].

The propagation or transient problems are time dependent. The response of a body under time varying force in the area of solid mechanics and under sudden heating or cooling in the field of heat transfer comprise this type of problems.

Table 1.1. Field of Finite Element Method

Area of Study	Equilibrium Problems	Eigenvalue Problems	Propagation Problems
Civil Engineering Structures	• Static analysis of trusses, frames, folded plates, shell roofs, shear walls, bridges and prestressed concrete structures	• Natural frequencies and modes of structures • Stability of structures	• Propagation of stress waves • Response of structures to aperiodic loads
Aircraft Structures	• Static analysis of aircraft wings, fuselages, fins, rockets, spacecraft and missile structures	• Natural frequencies, flutters, and stability of aircraft, rocket, spacecraft and missile structures	• Response of aircraft structures to random loads, dynamic response of aircraft and spacecraft to aperiodic loads
Heat Conduction	• Steady state temperature distribution in solids and fluids	-	• Transient heat flow in rocket nozzles, internal combustion engines, turbine blades, fins and building structures
Geomechanics	• Analysis of excavations, underground openings, rock joints and soil structure interaction problems. • Stress analysis in soils, dams, layered piles and machine foundations	• Natural frequencies and modes of dam reservoir systems and soil-structure interaction problems	• Time-dependent soil-structure interaction problems. • Transient seepage in soils and rocks. • Stress wave propagation in soils and rocks
Hydrodynamics	• Analysis of hydraulic structures and dams	• Sloshing of liquids in rigid and flexible containers	• Rarefied gas dynamics • Magneto hydrodynamic flows
Biomedical Engineering	• Stress analysis of eyeballs, bones and teeth • Mechanics of heart valves	-	• Impact analysis of skull • Dynamics of anatomical structures
Nuclear Engineering	• Analysis of nuclear pressure vessels and containment structures • Steady state temperature distribution in reactor components	• Natural frequencies and stability of containment structures • Neutron flux distribution	• Response of reactor containment structures to dynamic loads • Unsteady temperature distribution in reactor components

2. FUNDAMENTALS OF THE FINITE ELEMENT METHOD

2.1. Virtual Work Principle

Using the virtual work principle one can express the stiffness properties as well as the full equilibrium equations of a finite element. “ *Virtual work principle may be described as when a relatively small set of arbitrary displacements, compatible with the kinematic and geometric boundary conditions, are given to a system in equilibrium under external loads, the increase in work done by the external loads over these small displacements, equals the increase in the strain energy of the system*” [14].

This principle can be expressed as

$$\partial U_e = \partial W_e \quad (2.1)$$

The strain energy is given by

$$U_e = \int_V \frac{1}{2} (\epsilon_x \sigma_x + \epsilon_y \sigma_y + \epsilon_z \sigma_z + \gamma_{xy} \tau_{xy} + \gamma_{xz} \tau_{xz} + \gamma_{yz} \tau_{yz}) dV \quad (2.2)$$

$$U_e = \frac{1}{2} \int_V \{\epsilon\}^T \{\sigma\} dV \quad (2.3)$$

in which the volume integration is to be taken over the volume of the finite element. The stress and strain vectors can be defined by

$$\{\sigma\} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} \quad \{\epsilon\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \quad (2.4)$$

Considering Hooke's Law, stress-strain relationship for a two-dimensional problem can be generalized as

$$\varepsilon_x = \frac{\sigma_x}{E} - \mu \frac{\sigma_y}{E} \quad \varepsilon_y = -\mu \frac{\sigma_x}{E} + \frac{\sigma_y}{E} \quad \gamma_{xy} = \frac{\tau_{xy}}{G} \quad (2.5)$$

from the Equation (2.5), strains can be defined by

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\mu & 0 \\ -\mu & 1 & 0 \\ 0 & 0 & \frac{E}{G} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} \quad (2.6)$$

when stresses are taken into account;

$$\{\sigma\} = [D] \{\varepsilon\} \quad (2.7)$$

Substituting material matrix, $[D]$ into equation (2.7)

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & -\mu & 0 \\ -\mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (2.8)$$

The stress vector may be written as;

$$\{\sigma\} = [D] (\{\varepsilon\} - \{\varepsilon\}_o) + \{\sigma\}_o - \alpha_t (\Delta T) [D] \{D\}_T \quad (2.9)$$

in which, $[D]$, α_T , ΔT and $\{D\}_T$ are material elasticity matrix, coefficient of thermal expansion, change in temperature and temperature elasticity vector, respectively.

The initial strain, ϵ_0 , may arise from thermal action, initial lack of fit in a structural member, or other causes. The initial stress, σ_0 , arises when expansion is prohibited or when the expanded or overlength member is forced back to some initial reference size [15].

For the derivation of U and W , a plane stress element will be considered, subjected to internal forces. Each element has four nodes with two degrees of freedom per node, as shown in Figure 2.1, that is translations u and v in the x and y -directions, respectively. The purpose of the analysis is to determine the stress components at the center of each element.

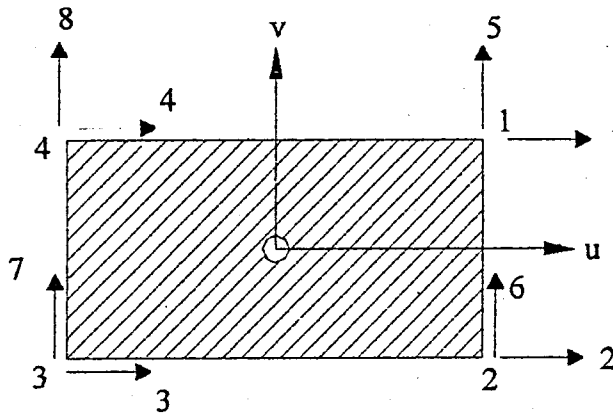


Figure 2.1. Number of degrees of freedom per node

u and v can be written as

$$u = N_1 d_1 + N_2 d_2 + N_3 d_3 + N_4 d_4 + N_5 d_5 + N_6 d_6 + N_7 d_7 + N_8 d_8 \quad (2.10)$$

$$v = N'_1 d_1 + N'_2 d_2 + N'_3 d_3 + N'_4 d_4 + N'_5 d_5 + N'_6 d_6 + N'_7 d_7 + N'_8 d_8$$

the equation (2.10) can be generalized as

The strains may be defined by

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}}_{[\underline{\Delta}]} \begin{Bmatrix} u \\ v \end{Bmatrix} \quad (2.12)$$

and strains, generic displacements and $[\underline{\Delta}][N]$ can be expressed as

$$\{\varepsilon\} = [\underline{\Delta}]\{u\} \quad \{u\} = [N]\{d\} \quad [\underline{\Delta}][N] = [G] \quad (2.13)$$

in which $[\underline{\Delta}]$ is the operator matrix to differentiate the shape functions.

Substituting (2.13) into (2.12) and using (2.8);

$$\{\varepsilon\} = [G]\{d\} \quad \{\sigma\} = \underbrace{[D][G]}_{[S]}\{d\} \quad (2.14)$$

in which $[G]$, $[D]$ and $[S]$ are the strain matrix, elasticity matrix and stress matrix, respectively. By replacing strains in (2.14) with strains in (2.9), and substitute them into strain energy expressed in (2.3), we obtain

$$\begin{aligned} U_e = & \frac{1}{2} \int_V \{d\}^T [G]^T [D][G] \{d\} dV - \frac{1}{2} \int_V \{d\}^T [G]^T [D] \{\varepsilon\}_0 dV \\ & + \frac{1}{2} \int_V \{d\}^T [G]^T \{\sigma\}_0 dV - \frac{1}{2} \alpha_T \Delta T \int_V \{d\}^T [G]^T \{D\}_T dV \end{aligned} \quad (2.15)$$

The external work for a finite element may be expressed as follows:

$$\begin{aligned}
 W_e = & \frac{1}{2} \{d\}^T \{p\} + \frac{1}{2} \int_v \{u\}^T \{X\} dV - \frac{1}{2} \int_v \{u\}^T (c\{\dot{u}\}) dV - \frac{1}{2} \int_v \{u\}^T (\rho\{\ddot{u}\}) dV \\
 & + \frac{1}{2} \int_s \{u\}_s^T \{p\}_s ds
 \end{aligned} \tag{2.16}$$

in which, $\{X\}$ = body force vector in x, y, z- directions, $\{u\}$ = generic displacement vector, $\{\dot{u}\}$ = generic velocity vector, $\{\ddot{u}\}$ = generic acceleration vector, ρ = mass density per unit volume, $\{u\}_s$ = external displacement and $\{p\}_s$ = external loads acting along the boundary.

Employing consistent shape functions for the generic velocities and accelerations to be the same shape functions used for the nodal displacements, we write;

$$\{u\} = [N]\{d\} \quad \{\dot{u}\} = [N]\{\dot{d}\} \quad \{\ddot{u}\} = [N]\{\ddot{d}\} \tag{2.17}$$

the external work then becomes

$$\begin{aligned}
 W_e = & \frac{1}{2} \{d\}^T \{p\} + \frac{1}{2} \{d\}^T \int_v [N]^T \{X\} dV - \frac{1}{2} \{d\}^T c \left(\int_v [N]^T [N] dV \right) \{\dot{d}\} \\
 & - \frac{1}{2} \{d\}^T \rho \left(\int_v [N]^T [N] dV \right) \{\ddot{d}\} + \frac{1}{2} \{d\}^T \int_s [N]^T \{p\}_s ds
 \end{aligned} \tag{2.18}$$

The strain energy U_e and the external work W_e for a particular element can be combined for determining the potential energy, stating that, with a small variation of ∂u in the generic displacements or ∂d in the displacements, the variation in the potential energy $\partial \Pi_e$ will be zero [14]. That is:

$$\begin{aligned}
\partial \Pi_e = & \underbrace{\left(\int_v [G]^T [D] [G] dV \right)}_{[k]} \{d\} - \underbrace{\left(\int_v [G]^T dV \right)}_{-\{f\}_{\epsilon_0}} [D] \{\epsilon\}_0 + \underbrace{\left(\int_v [G]^T dV \right)}_{\{f\}_{\sigma_0}} [\sigma]_0 \\
& - \underbrace{\alpha_T \Delta T \left(\int_v [G]^T dV \right)}_{-\{f\}_T} \{D\}_T - \{p\} + \underbrace{\int_v [N]^T \{X\} dV}_{-\{f\}_X} \\
& - \underbrace{\left(\int_v c [N]^T [N] dV \right)}_{[c]} \{\dot{d}\} - \underbrace{\left(\rho \int_v [N]^T [N] dV \right)}_{[m]} \{\ddot{d}\} + \underbrace{\int_s [N]^T \{p\}_s ds}_{-\{f\}_s} = 0
\end{aligned} \tag{2.19}$$

in which; $[k]$ = stiffness matrix, $\{f\}_{\epsilon_0}$ = nodal loads due to initial strains, $\{f\}_{\sigma_0}$ = nodal loads due to initial stresses, $\{f\}_T$ = temperature change loads, $\{f\}_X$ = body forces at nodes, $[c]$ = consistent damping matrix, $[m]$ = consistent mass matrix, $\{f\}_s$ = edge loads at nodes.

Note that, the common multipliers $(\frac{1}{2})$ in front of the terms have been cancelled before expressing $\partial \Pi_e$. Similarly, all of the common multiplier $\partial \{d\}^T$, have been also cancelled, since the potential energy will be zero.

The initial deformations can be easily placed in (2.20). Since the deformations are known, the forces, which will create those deformations, can be added external loads.

$$\{f\}_{\text{known}} = [k] \{d\}_{\text{known}} \tag{2.20}$$

As a result of (2.19); the following formula can be obtained:

$$[k] \{d\} + [c] \{\dot{d}\} + [m] \{\ddot{d}\} = \{p\} - \left[\{f\}_X + \{f\}_s + \{f\}_{\epsilon_0} + \{f\}_{\sigma_0} + \{f\}_T + \{f\}_{\text{known}} \right] \tag{2.21}$$

2.2. Geometric Shapes of Finite Elements

The first step in a finite element analysis is to select the type of element and the corresponding finite element mesh. For a given type of element the accuracy increases with decreasing element size and, in general, one will use small elements in regions where the unknown function varies rapidly.

The process of discretization is essentially an exercise of engineering judgement. The shapes, sizes, number and configurations of the elements have to be chosen carefully for a good accuracy.

Discretization of an domain into a number of nodes and elements is an essential requirement for finite element analysis of the object. The continuous object has transformed into a mesh comprising of discrete nodes and elements. However, the user interaction required for discretization proved so burdensome that is necessitated the development automatic mesh generation schemes. An automatic mesh generation scheme should require minimum user input and should generate a mesh of well-shaped elements in a time efficient manner.

The choice of the type of element is generally dictated by the geometry of the body and the number of coordinates necessary to describe the system. When the configuration and other details of the problem can be described in terms of two independent spatial coordinates, two-dimensional elements can be used. The basic element for two-dimensional analysis is the triangular elements. And the other ones are shown in Figure 2.2. In some cases the use of quadrilateral element proves to be advantageous.

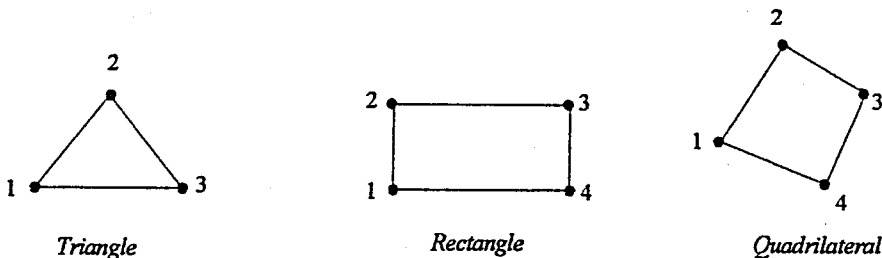
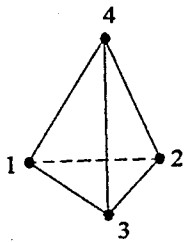
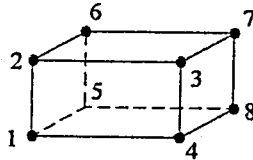


Figure 2.2. Two-dimensional finite elements

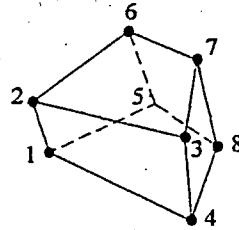
If the geometry, material properties other parameters of the body can be described by the independent spatial coordinates, the body can be idealized by using three-dimensional elements as illustrated in Figure 2.3.



Tetrahedron



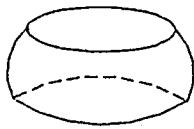
Rectangular Prism



Hexahedron

Figure 2.3. Three-dimensional finite elements

Some problems, which are three dimensional, can be explained with only one or two independent coordinates. Such problems can be idealized by using an axisymmetric element shown in Figure 2.4.



One-dimensional



Two-dimensional

Figure 2.4. Axisymmetric elements

For the discretization of curved problems, the elements with curved boundaries are acceptable exposed in Figure 2.5.

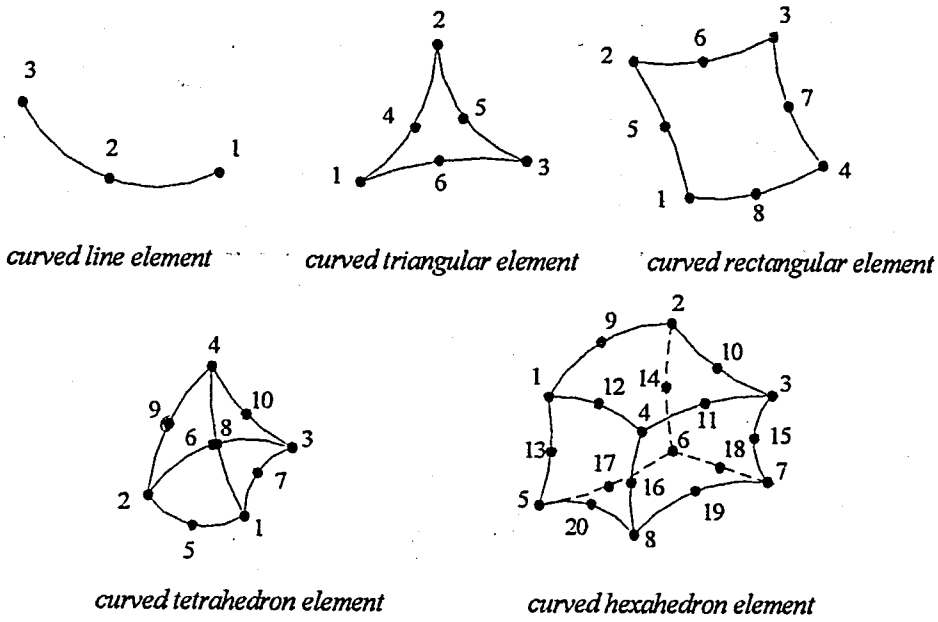


Figure 2.5. Curved finite elements

The ability to model curved boundaries has been achieved by the addition of midside nodes. Finite elements with straight sides are known as linear elements; those with curved sides are called higher-order elements [1].

The size of the finite element is as important as its shapes. The size of the element influences the convergence of the solution directly. If the size of the element is small, the solution is supposed to be more accurate. However, the smaller the size, the more is the computational time. Also, the aspect ratio, the shape of the element in the assemblage of elements, affects the accuracy of the finite element solution. For instance, in a rectangle, the ratio of larger and smaller dimension is called the aspect ratio and in a good finite element mesh, it should be as close to unity as possible. The maximum value of the aspect ratio should be smaller than 1.5.

The number of elements to be chosen for the idealization is related to the accuracy desired, size of elements, and the number of degrees of freedom involved. An increase in the number of elements generally means more accurate results.

In addition, the body should not possess any unexpected changes in geometry, material properties and external conditions. The body should be divided possibly into equal

parts. Hence, the spacing of the nodes may be uniform. In contrast, if there are any discontinuities in the problem, nodes have to be set up at these continuities. Also, finite element method is the most popular and reliable tool for the computation of the stress concentration. For improving the accuracy the general practice is to refine locally the mesh. However, when the geometry is defined by four-sided surfaces which are meshed separately with quadrilaterals, the local mesh refinement around the stress concentration such as a hole, will usually lead to an unnecessary increase in the number of elements in the adjacent surfaces to insure proper element connectivity and displacement compatibility [16].

2.3. Automatic Mesh Generation

The objective of the design of any adaptive finite element analysis procedure is to achieve economically the maximum rate of the convergence of the finite element approximation. This objective can be realized by designing an automatic mesh refinement process which enables user-specified accuracy to be achieved on an optimal mesh. Any posteriori error estimator and an automatic mesh refinement strategy play vital roles in developing fully automatic adaptive analysis procedure [16], [17].

The bandwidth of the overall system matrix depends on the manner in which the nodes are numbered. For simple systems, it is easy to label the nodes. But for large systems, it is impossible to label. Therefore, automatic mesh generation algorithms, capable of discretizing any geometry into an efficient finite element mesh without user intervention. The availability of geometric modeling systems has greatly increased the efficiency of these algorithms. The minimum requirement of an automatic mesh generator is that the topology representation of the original geometry is maintained in the resulting finite element discretization [1].

In the process of mesh generation a background mesh, which completely covers the domain to be gridded is used to define the mesh density and element orientation. The mesh generation process starts by generating nodes along the boundary segments according to the specified mesh size distribution. The boundary of the domain will be discretised and transformed to a polygon with an even number of sides. The techniques

used for the element generations are an extension of the ones used in the triangular element generation [18]. Two triangles which have a common side are first generated according to the prescribed mesh size distribution; these triangles are then combined to form a quadrilateral. The generation front is updated once a quadrilateral is generated. New sides are added to the generation front and used sides are deleted. The boundary of the domain being gridded is always formed by the generation front. The generation front will be divided into subfronts once it contains more than one closed loop of sides. Each subfront contains an even number of sides and has one set of closed loop of sides. The element generation continues successively or in parallel in each subfront. The element generation process is terminated when every generation subfront is empty. The domain of interest is then covered entirely by quadrilaterals [19], [20].

Two-dimensional mesh generation methods called node connection methods comprise of two phases; namely node generation and element generation [16]. Nodes are first generated along and within the boundaries of the body. Elements are generated next by connecting the generated nodes.

Node generation technique can be either random or non-random. A random node generation method suffers from several disadvantages. The nodes are not well spaced. The generation of a node is an iterative process. The number of iterations is usually limited and there is no time efficient. On the contrary, a non-random node generation technique results in a set of well-spaced nodes. It is reliable and time efficient.

The set of nodes generated are completely enclosed by a triangle defined by three arbitrary points in triangular element generation. The circumcenter and circumradius of this enclosing triangle are computed. The generated nodes are inserted into this triangle one after the other. When a node is inserted, all triangles whose circumcircles enclose the inserted point are identified. The nodes of these triangles form an adjacent set of points. The adjacent set of points form a polygon within which the inserted node lies. The sides of this polygon are connected to the inserted nodes resulting in a new triangulation. At this stage the original set of triangles whose circumcircles enclose the inserted node are discarded. The process is repeated for each of the generated nodes inserted [21], [22].

2.4. Convergence Criteria

In order to ensure convergence to the correct result certain simple requirements have to be satisfied. The assumed shape functions limit the infinite degrees of freedom of the system, and the true minimum of the energy may never be reached, irrespective of the fineness of subdivision. The displacement function should be able to signify the true displacement distribution as closely as possible [22]. The displacement function should be such that it does not permit straining of an element to occur when the nodal displacements are caused by a rigid body displacement. The approximation of the displacement function must be able to represent an arbitrary constant rigid body motion and also an arbitrary constant strain state [2]. Nodal deformations conforming to a uniform shape change should produce uniform strains throughout the element, as shown in Figure 2.6 [14]. When nodal deformations correspond to a rigid body motion, stresses or strains should not develop within the element, as presented in Figure 2.7.

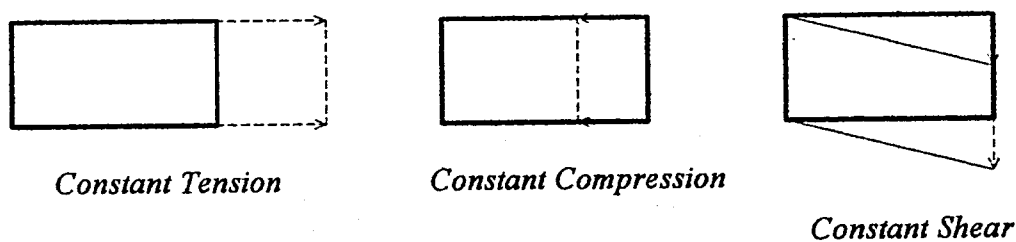


Figure 2.6. Uniform Stress Criterion

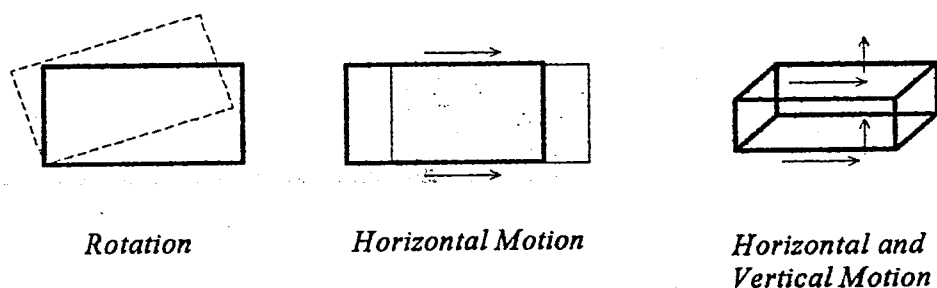


Figure 2.7. Rigid Body Criterion

A displacement function should not contribute any value, at a point along an edge, which is not in the immediate neighbourhood of the joint in the domain. Also it should pass from the zero value at all joints, other than the joint for which it is specified [14]. Displacement functions should be a linear function of nodal displacements.

3. SHAPE FUNCTIONS

3.1. Introduction

Shape functions for commonly used plane and three-dimensional finite elements were established in 1970's. Over this time, considerable advances have been made in the design of computer algebra packages so that new elements can in principle be formulated quite quickly.

The shape function is usually denoted by the letter N and is usually the coefficient that appears in the interpolation polynomial. In other words, an approximation polynomial which is equal to the function it approximates at a number of specified stations or points is called a shape function [1]. A shape function is written for each individual node of a finite element and has the property that its magnitude is 1 at that node and 0 for all other nodes in that element. The terminology is often interchanged between interpolation polynomial and shape function.

In the finite element analysis, the values of the field variable at the nodes of the element are treated as unknowns and the value of the field variable at any point inside the element is then specified as a function of the nodal unknowns and the coordinates of the point. Thus the specified function can be considered as an shape function since it matches with the nodal unknowns at various nodes of the element. All is known relating to useful element shape function families and that Lagrangian and what has become known as Serendipity interpolation will suffice for all time [23].

3.2. One Dimensional Linear Element

3.2.1. Shape Function for One Dimensional Linear Element

The shape function of an element may be expressed basically by the help of the Lagrange polynomials. The Lagrange polynomials are defined by [14]

$$y(x) = \frac{(x-x_1)(x-x_2)\dots(x-x_n)}{(x_i-x_1)(x_i-x_2)\dots(x_i-x_n)} = \prod_{\substack{j=1 \\ (i \neq j)}}^{j=n} \frac{(x-x_j)}{(x_i-x_j)} \quad (3.1)$$

The Lagrange polynomials may be generated for as many nodes as possible, considering the origin on the left-hand side or on the mid-point. In the equation (3.1) $(x-x_i)$ as well as (x_i-x_i) will be skipped. The Lagrange shape functions of a linear element in terms of the Cartesian coordinates may be obtained as follows:

$$y(x) = \frac{(\xi-\xi_1)(\xi-\xi_2)\dots(\xi-\xi_n)}{(\xi_i-\xi_1)(\xi_i-\xi_2)\dots(\xi_i-\xi_n)} = \prod_{\substack{j=1 \\ (i \neq j)}}^{j=n} \frac{(\xi-\xi_j)}{(\xi_i-\xi_j)} \quad (3.2)$$

in which, $\xi = \frac{x}{a}$.

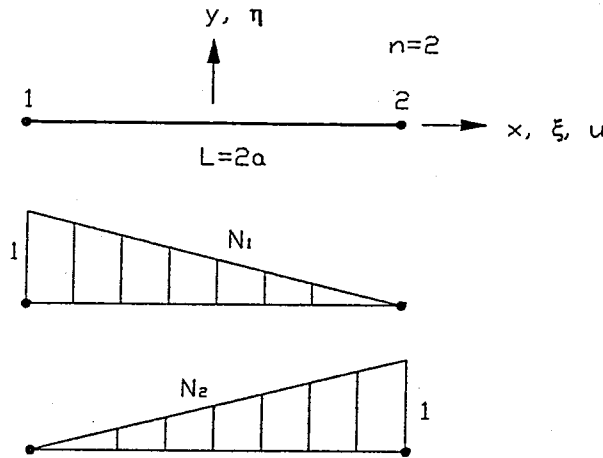


Figure 3.1. One-dimensional Linear Element

The shape functions of this linear element are

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

$$N_2 = \frac{1}{2}(1+\xi) \quad (3.3)$$

3.2.2. Stiffness Matrix of One Dimensional Linear Element

In Section 2.1, the development of general form of the stiffness formulation is explained by using virtual work principle. From the equation (2.12), we obtained

$$[G] = [\underline{\Delta}][N] = \left[\frac{\partial}{\partial x} \right] \begin{bmatrix} \frac{1}{2}(1-\xi) & \frac{1}{2}(1+\xi) \end{bmatrix} \quad (3.4)$$

$$[G] = \begin{bmatrix} -\frac{1}{2a} & \frac{1}{2a} \end{bmatrix}$$

where $\xi = \frac{x}{a}$. For the differentiation, the chain rule is used. This rule is illustrated as

$$\frac{\partial N}{\partial x} = \frac{\partial N}{\partial \xi} \frac{\partial \xi}{\partial x} = \frac{1}{a} \frac{\partial N}{\partial \xi} \quad (3.5)$$

From the equation (2.19), the stiffness matrix of one-dimensional element is

$$[k] = \int_V [G]^T [D] [G] dV = \underbrace{[G]^T [D] [G]}_{[H]} A \int_{-a}^a dx \quad (3.6)$$

$$[k] = (2a) [H] A$$

in which; A = the cross sectional area of the bar element, $[D] = E$.

The material elasticity matrix, $[D]$ is equal to E_x for the bar elements. Substituting $[G]$ into equation (3.6), and writing $L = 2a$, the stiffness matrix becomes

In Figure 3.2, for a point P within the domain of the triangle [7], the area coordinates are

$$P(\xi, \eta, \zeta) \quad (3.8)$$

where

$$\left. \begin{aligned} \xi &= \frac{A_1}{A} = \frac{h_1}{H_1} \\ \eta &= \frac{A_2}{A} \\ \zeta &= \frac{A_3}{A} \end{aligned} \right\} \xi + \eta + \zeta = 1 \quad (3.9)$$

The relationship between the Cartesian and natural coordinates may be expressed in a linear fashion as follows:

$$\begin{aligned} x &= a_1\xi + a_2\eta + a_3\zeta \\ y &= b_1\xi + b_2\eta + b_3\zeta \end{aligned} \quad (3.10)$$

Reanalyzing and satisfying that at corners 1, 2 and 3 the area coordinates are $(\xi = 1, 0, 0)$, $(0, \eta = 1, 0)$ and $(0, 0, \zeta = 1)$, respectively the coordinates of the corners 1, 2 and 3 at Equation (3.10); we obtain

$$\left. \begin{aligned} a_1 &= x_1 & b_1 &= y_1 \\ a_2 &= x_2 & b_2 &= y_2 \\ a_3 &= x_3 & b_3 &= y_3 \end{aligned} \right\} \quad (3.11)$$

Therefore, we may write the relationship as;

$$\left. \begin{aligned} 1 &= \xi + \eta + \zeta \\ x &= a_1\xi + a_2\eta + a_3\zeta \\ y &= b_1\xi + b_2\eta + b_3\zeta \end{aligned} \right\} \quad (3.12)$$

or, in matrix form

$$\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} = \underbrace{\begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix}}_{[B]} \begin{Bmatrix} \xi \\ \eta \\ \zeta \end{Bmatrix} \quad (3.13)$$

In order to obtain (ξ, η, ζ) in terms of (x, y) , the matrix $[B]$ must be inverted. We obtain

$$\begin{Bmatrix} \xi \\ \eta \\ \zeta \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} (x_2y_3 - x_3y_2) & (y_2 - y_3) & (x_3 - x_2) \\ (x_3y_1 - x_1y_3) & (y_3 - y_1) & (x_1 - x_3) \\ (x_1y_2 - x_2y_1) & (y_1 - y_2) & (x_2 - x_1) \end{bmatrix} \begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} \quad (3.14)$$

For a triangle with three nodes, the linear interpolation model for $\{u\}$ is expressed in equation (2.12). Shape functions of a linear interpolation model will be

$$N_1 = \xi \quad N_2 = \eta \quad N_3 = \zeta \quad (3.15)$$

3.3.2. Stiffness Matrix of the Constant Stress Triangle

The triangular element is considered to lie within the x-y plane as shown in Figure 3.3.

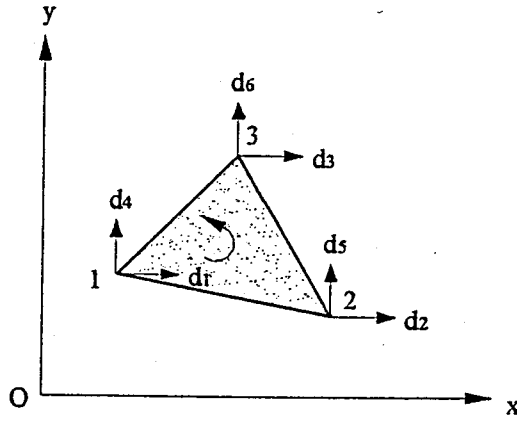


Figure 3.3. Degrees of freedom of a triangle

The generic displacements u and v may be expressed by Equation 2.13 in matrix form as follows:

$$\{u\} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \xi & \eta & \zeta & 0 & 0 & 0 \\ 0 & 0 & 0 & \xi & \eta & \zeta \end{bmatrix} \begin{Bmatrix} d_1 \\ \vdots \\ d_6 \end{Bmatrix} \quad (3.15)$$

Substituting the shape functions into equation (2.11), the strain matrix is obtained as

$$\{\epsilon\} = [\underline{\Delta}] \{u\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \underbrace{\begin{bmatrix} b_1 & b_2 & b_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & a_1 & a_2 & a_3 \\ a_1 & a_2 & a_3 & b_1 & b_2 & b_3 \end{bmatrix}}_{[G]} \frac{1}{2A} \{d\} \quad (3.16)$$

where a_1 , a_2 and a_3 are the x -projections of the side-1, side-2 and side-3, respectively. Similarly, b_1 , b_2 and b_3 are the y -projections of the side-1, side-2 and side-3, with opposite sign, respectively, as shown in Figure 3.4.

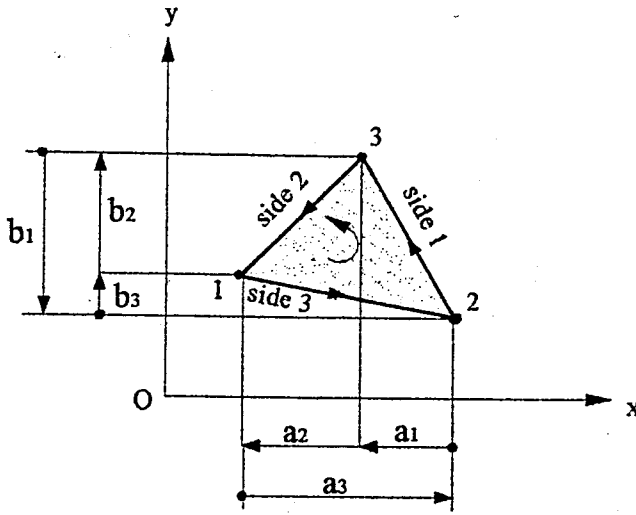


Figure 3.4. Constant stress triangle

The stress matrix is defined by Equation (2.8) as

$$[S] = [D][G] \quad (3.17)$$

Rearranging equation (3.17), the stress matrix of the constant triangle becomes

$$[S] = \begin{bmatrix} b_1 D_{11} & b_2 D_{11} & b_3 D_{11} & a_1 D_{12} & a_2 D_{12} & a_3 D_{12} \\ b_1 D_{12} & b_2 D_{12} & b_3 D_{12} & a_1 D_{22} & a_2 D_{22} & a_3 D_{22} \\ a_1 D_{33} & a_2 D_{33} & a_3 D_{33} & b_1 D_{33} & b_2 D_{33} & b_3 D_{33} \end{bmatrix} \frac{1}{2A} \quad (3.18)$$

in which for a two-dimensional isotropic domain, $D_{11} = \frac{E}{1-\mu^2}$, $D_{12} = \mu D_{11}$,

$$D_{33} = \frac{E}{2(1+\mu)} \text{ and } D_{22} = D_{11}.$$

The stiffness matrix is obtained by substituting the stress matrix in equation (2.19). The explicit form of the stiffness matrix is given in Table 3.1, after executing the following multiplications and performing the necessary integrations.

$$[k] = \int_V [G]^T \underbrace{[D][G]}_{[S]} dV = \int_V [G]^T [S] t dA \quad (3.19)$$

in which, t = thickness of the triangular element, $[S]$ = stress matrix.

3.4. Rectangular Element in Plane Stress

3.4.1. Shape Functions of a Rectangular Element

The shape functions of one-dimensional linear element were presented earlier in Equation (3.3). These shape functions are used as the shape functions of the linear rectangular element in each direction. The configuration of the Lagrange shape functions for a rectangular element is illustrated in Figure 3.5.

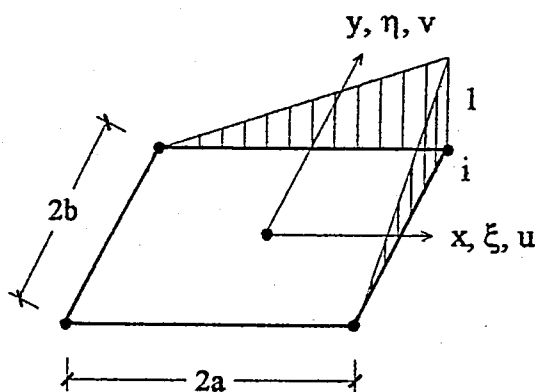


Figure 3.5. The shape functions of a rectangle

The Lagrangian shape functions of any i^{th} node of the element are defined as

$$N_i = \frac{1}{2}(1 + \xi \xi_i) \frac{1}{2}(1 + \eta \eta_i) \quad (3.20)$$

in which; $\xi = \frac{x}{a}$, $\eta = \frac{y}{b}$, ξ_i and η_i are either +1 or -1 depending of the location of the node.

Rearranging the equation (3.20) for all nodes as shown in Figure 3.6, the shape functions become

$$\begin{aligned}
 N_1 = N_5 &= \frac{1}{4}(1+\xi)(1+\eta) \\
 N_2 = N_6 &= \frac{1}{4}(1+\xi)(1-\eta) \\
 N_3 = N_7 &= \frac{1}{4}(1-\xi)(1+\eta) \\
 N_4 = N_8 &= \frac{1}{4}(1-\xi)(1-\eta)
 \end{aligned} \tag{3.21}$$

3.4.2. Stiffness Matrix of the Rectangular Element

Plane stress is defined as a state of stress where the only non-zero stresses are σ_{xx} , σ_{yy} and σ_{xy} . Degrees of freedom are shown in Figure 3.6.

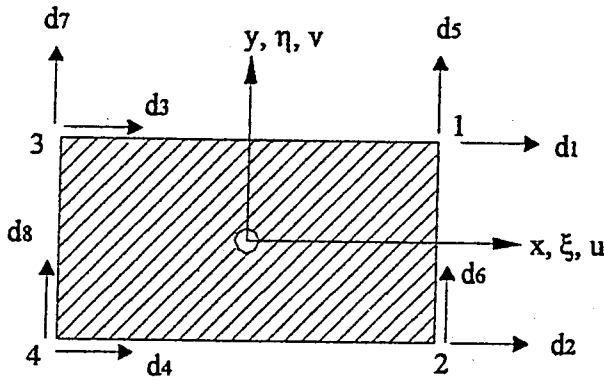


Figure 3.6. Rectangular element

The generic displacements u and v may be expressed as

$$u = N_1 d_1 + N_2 d_2 + N_3 d_3 + N_4 d_4 + 0 d_5 + 0 d_6 + 0 d_7 + 0 d_8 \tag{3.22}$$

$$v = 0 d_1 + 0 d_2 + 0 d_3 + 0 d_4 + N_5 d_5 + N_6 d_6 + N_7 d_7 + N_8 d_8 \tag{3.23}$$

or, in matrix notation

$$\{u\} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & N_5 & N_6 & N_7 & N_8 \end{bmatrix} \begin{Bmatrix} d_1 \\ \vdots \\ d_8 \end{Bmatrix} \quad (3.24)$$

When Eqs. (2.11) and (2.12) are considered and the shape functions defined in (3.21) are substituted into (2.11), we obtain the strain matrix $[G]$ as follows

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \underbrace{\begin{bmatrix} b(1+\eta) & b(1-\eta) & -b(1+\eta) & -b(1-\eta) & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & a(1+\xi) & -a(1+\xi) & a(1-\xi) & -a(1-\xi) \\ a(1+\xi) & -a(1+\xi) & a(1-\xi) & -a(1-\xi) & b(1+\eta) & b(1-\eta) & -b(1+\eta) & -b(1-\eta) \end{bmatrix}}_{[G]} \frac{1}{4ab} \{d\}$$

Substituting $[G]$ into (2.18), the stress matrix $[S]$ becomes

$$[S] = \frac{1}{4ab} \begin{bmatrix} D_{11}b(1+\eta) & D_{11}b(1-\eta) & -D_{11}b(1+\eta) & -D_{11}b(1-\eta) & D_{12}a(1+\xi) & -D_{12}a(1+\xi) & D_{12}a(1-\xi) & -D_{12}a(1-\xi) \\ D_{12}b(1+\eta) & D_{12}b(1-\eta) & -D_{12}b(1+\eta) & -D_{12}b(1-\eta) & D_{22}a(1+\xi) & -D_{22}a(1+\xi) & D_{22}a(1-\xi) & -D_{22}a(1-\xi) \\ D_{33}a(1+\xi) & -D_{33}a(1+\xi) & D_{33}a(1-\xi) & -D_{33}a(1-\xi) & D_{33}b(1+\eta) & D_{33}b(1-\eta) & -D_{33}b(1+\eta) & -D_{33}b(1-\eta) \end{bmatrix}$$

In order to obtain the stiffness matrix $[k]$, equation (3.19) may be used

$$[k] = \int \underbrace{[G]^T [D] [G]}_{[G]} dV \quad (3.25)$$

After the necessary operations the stiffness matrix is obtained as given in Table 3.2., where

$\beta = \frac{b}{a}$ is the aspect ratio, which is recommended to be within $0.5 < \beta < 2$.

4. HIGHER ORDER FINITE ELEMENTS

4.1. Introduction

A finite element becomes a higher order element if additional nodes are defined along its boundaries. In higher order elements, some additional midside and/or interior nodes are introduced in order to increase the accuracy of solutions. Certainly, the number of generalized coordinates and degrees of shape functions are selected to match the number of nodal degrees of freedom. Higher order elements are basically needed to achieve as close results as possible to the exact solutions. Although the higher order elements do not reduce the computational time significantly, the reduction in the number of elements generally reduces the effort needed in the preparation of the data and the chances of errors in the input data. The higher order elements are especially useful in those cases where the gradient of the field variable is expected to vary rapidly. In these cases the simplex elements, which approximate the gradient by the set of constant values, do not yield good results. The combination of greater accuracy and the reduction in the data preparation effort have resulted in the widespread use of higher order elements in several practical applications.

Two independent sets of nodes may exist in a higher order element. In one set of nodes only the coordinate transformations (element shapes) are defined and in the second set of nodes the interpolating polynomials for the physical unknowns are defined. There are three distinct possibilities regarding the number of nodes in these two sets. Whenever the number of nodes used to define the element shape is less than the number of nodes used to define the physical interpolation function, these are called as *subparametric*. If however, the number of nodes used to define the element shape is equal to the number of nodes used to define the physical parameters, these elements are called as the *isoparametric* elements. Finally, if the number of nodes used to define the element geometry are greater than the number of nodes used to define the physical parameters, these elements are called *superparametric* elements as shown in Figure 4.1.

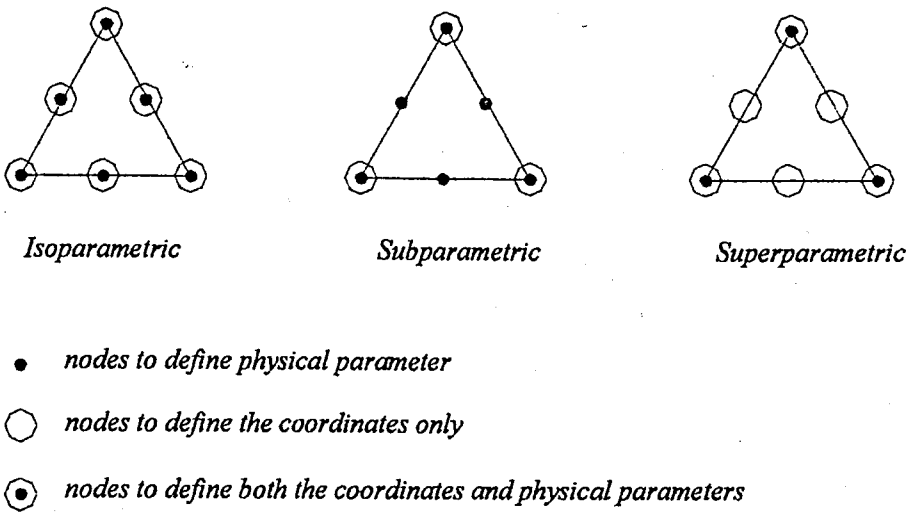


Figure 4.1. Isoparametric, subparametric and superparametric elements

Problems involved curved boundaries cannot be modeled satisfactorily by using straight-sided elements. The family of elements known as isoparametric elements has been developed for this purpose. The basic idea underlying the isoparametric elements is to use the same interpolation functions to define the element shape or geometry as well as the field variable within the element. The isoparametric formulation makes it possible to have nonrectangular elements, elements with curved sides, infinite elements for unbounded media, and singularity elements for fracture mechanics [24]. To derive the isoparametric element equations, a local and natural coordinate system is introduced for each element shape. Then the interpolation or shape functions will have to be expressed in terms of natural coordinates. The representation of geometry in terms of nonlinear shape functions can be considered as a mapping procedure which transforms a regular shape like a straight sided triangle or rectangle in local coordinate system into a distorted shape like a curved sided triangle or rectangle in the global Cartesian coordinate system. This concept may be used in representing problems with curved boundaries by the help of curved sided isoparametric elements. Also, the isoparametric elements are extensively used in three-dimensional and shell analysis problems.

There are two types of shape functions for the higher order elements as will be discussed below: a) The Pascal interpolation polynomials, b) The Lagrange shape functions.

4.2. Pascal Polynomials

The finite element method is based upon the concept of approximating a continuous function (temperature, pressure, displacement, etc.) by a discrete model that is composed of a set of piecewise continuous functions which are defined over a finite number of subdomains called element. The most popular form of the element function is Pascal polynomial. The order of the polynomial depends on the number of items known about the continuous function at each element node.

The basic idea of the finite element method is to consider a body as composed of several elements which are connected at specified nodal points. The unknown solution inside any finite element is assumed to be given by a simple function in terms of the nodal values of that element. The nodal values of the solution, also known as nodal degrees of freedom, are treated as unknowns in formulating the system or overall equations. Once the nodal degrees of freedom are known, the solution within any finite element will also be known.

Finite elements may be classified into three groups according to the order of the element interpolation polynomial. These groupings are simplex, complex and multiplex. The simplex elements have an approximating polynomial that consists of the constant term plus the linear terms. The number of coefficients in the polynomial is equal to the dimension of the coordinate space plus one. The polynomial

$$u = a_1 + a_2 x + a_3 y \quad (4.1)$$

is the simplex function for the two-dimensional triangular element. The polynomial is linear in x and y and contains three coefficients because the triangle has three nodes.

The complex elements use a polynomial function consisting of the constant and linear terms plus second, third and higher-order terms, as they are needed. The complex elements can have the same shapes as the simplex elements, but they have additional boundary or midside nodes and may also have internal nodes. The primary difference between a simplex and a complex element is that the number of nodes in a complex

element is greater than one plus the dimension of the coordinate space. An interpolating polynomial for a two-dimensional complex triangular element as shown in Figure 2.5, is

$$u = a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 x y + a_6 y^2 \tag{4.2}$$

This equation has six coefficients; therefore, the element must have six nodes. It is also complete through the second order terms.

The multiplex elements also use polynomials containing higher-order terms, but the element boundaries must be parallel to the respective coordinate axes in order to achieve inter-element continuity. The element boundaries of the simplex and complex elements are not subjected to this restriction [20].

The pattern of variation of the field variable resulting from the polynomial model should be independent of the local coordinate system. This property is called geometric isotropy [1]. In order to achieve this isotropy, the Pascal polynomial should contain terms which should not violate the symmetry with respect to x and y and their powers. The Pascal polynomials for two and three-dimensional elements are shown in Figure 4.2 and Figure 4.3.

	<u>Order</u>
1	0
x.....y	1
x ² xy.....y ²	2
x ³ x ² y.....xy ² y ³	3
x ⁴ x ³ y.....x ² y ² xy ³ y ⁴	4
x ⁵ x ⁴ y.....x ³ y ² x ² y ³ xy ⁴ y ⁵	5

Figure 4.2. Pascal polynomials for two-dimensional problems

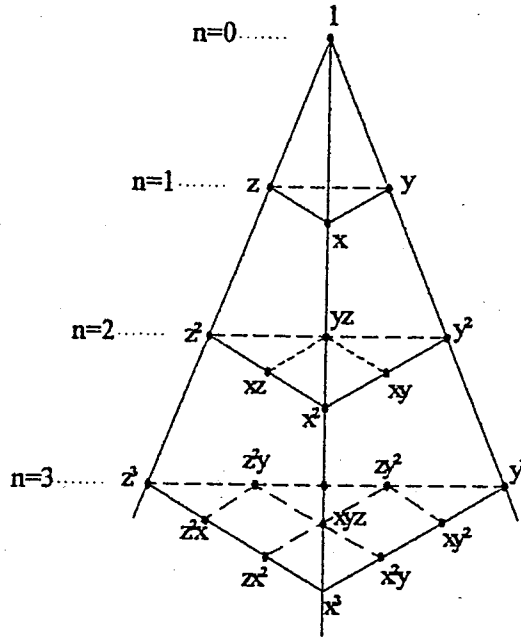


Figure 4.3. Pascal polynomials for three-dimensional problems

The Pascal interpolation polynomials for a triangle may be expressed in matrix notation as;

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} F_1(x, y) \\ F_2(x, y) \end{bmatrix} = \begin{bmatrix} F(x, y) \end{bmatrix} \begin{Bmatrix} a_1 \\ \vdots \\ a_6 \end{Bmatrix} \quad (4.3)$$

$[c]\{d\}$

After the nodal coordinates x_1, x_2, x_3 and y_1, y_2, y_3 are substituted the general displacements become the nodal displacements. Therefore, Equation (4.3) becomes

$$\begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \end{Bmatrix} = \underbrace{\begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix}}_{[F]_b} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{Bmatrix} \quad (4.4)$$

$$\{d\} = [F]_b \{a\}$$

The coefficients a_1 and a_6 are obtained from equation (4.4) by inversion as follows:

$$\{a\} = [F]_b^{-1} \{d\} = [C] \{d\} \quad (4.5)$$

Therefore the generic displacements may be expressed as

$$\{u\} = \underbrace{[F(x, y)]}_{[N]} [C] \{d\} \quad (4.6)$$

in which $[N] = [F(x, y)] [C]$.

4.3. One Dimensional Higher Order Elements

The quadratic Pascal polynomial model for one-dimensional element is expressed as

$$u = a_1 + a_2 x + a_3 x^2 \quad (4.7)$$

Since there are three constants a_1 , a_2 and a_3 in equation (4.7) the element is assumed to have three degrees of freedom, one at each of three nodes as shown in Figure 4.3.

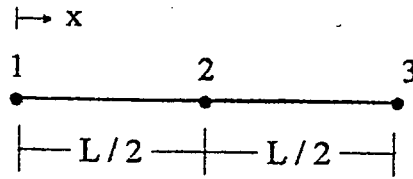


Figure 4.4. Quadratic Line Element

By the help of (4.3), the nodal displacements become

$$\begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 1 & \frac{L}{2} & \frac{L^2}{4} \\ 1 & L & L^2 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = [F]_o \{a\} \quad (4.8)$$

$$\left. \begin{aligned} d_1 &= a_1 & (x=0) \\ d_2 &= a_1 + a_2 \frac{L}{2} + a_3 \frac{L^2}{4} & (x=\frac{L}{2}) \\ d_3 &= a_1 + a_2 L + a_3 L^2 & (x=L) \end{aligned} \right\} \quad (4.9)$$

By investing the $[F]_o$ matrix, we obtain the $[C]$ matrix as

$$[C] = [F]_o^{-1} = \begin{Bmatrix} 1 & 0 & 0 \\ -\frac{3}{L} & \frac{4}{L} & -\frac{1}{L} \\ \frac{2}{L^2} & -\frac{4}{L^2} & \frac{2}{L^2} \end{Bmatrix} \quad (4.10)$$

Finally, the shape functions may be written as

$$[N] = [F(x, y)] [C] \quad (4.11)$$

$$[N] = \begin{bmatrix} 1 & x & x^2 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 \\ -\frac{3}{L} & \frac{4}{L} & -\frac{1}{L} \\ \frac{2}{L} & -\frac{4}{L} & \frac{2}{L} \end{bmatrix} = [N_1 \quad N_2 \quad N_3] \quad (4.12)$$

which yields

$$\left. \begin{aligned} N_1 &= 1 - \frac{3x}{L} + \frac{2x^2}{L^2} \\ N_2 &= \frac{4x}{L} - \frac{4x^2}{L^2} \\ N_3 &= -\frac{x}{L} + \frac{2x^2}{L^2} \end{aligned} \right\} \quad (4.13)$$

and also using $\xi = \frac{x}{L}$, the shape functions become

$$\begin{aligned} N_1 &= 1 - 3\xi + 2\xi^2 \\ N_2 &= 4\xi (1 - \xi) \\ N_3 &= -\xi (1 - 2\xi) \end{aligned} \quad (4.14)$$

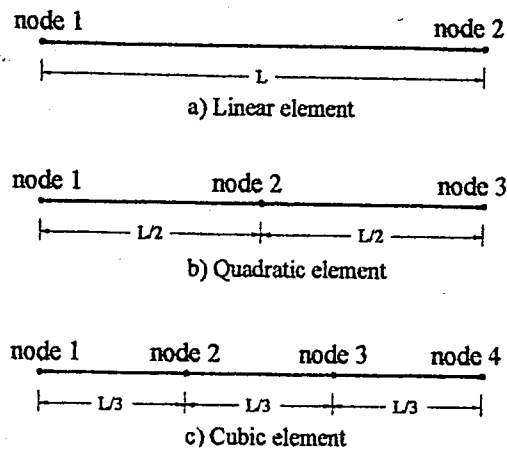


Figure 4.5. Location of nodes for one-dimensional higher order element

4.4. One Dimensional Lagrange Polynomials

The Lagrange polynomials are defined in equation (3.1). The shape functions of the linear, quadratic as well as cubic form of the linear elements as shown in Figure 4.6 and 4.7 may be obtained as follows:

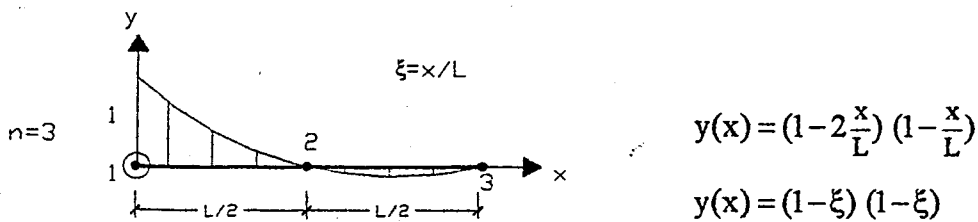


Figure 4.6. Shape function of quadratic linear element for node 1

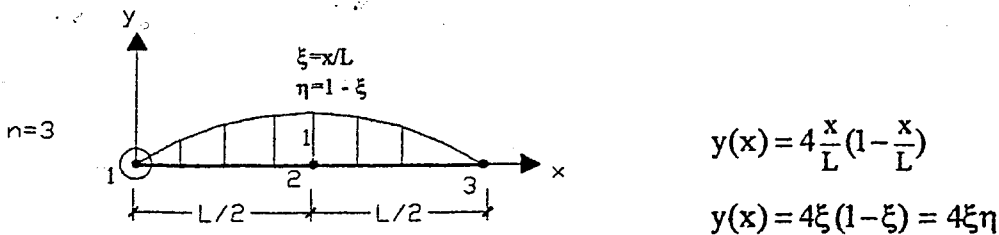
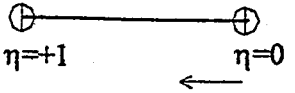
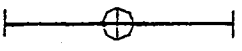
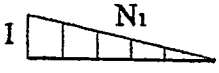
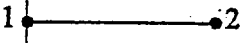
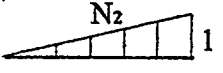
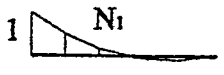
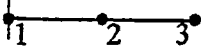
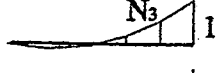
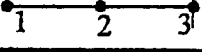
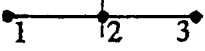
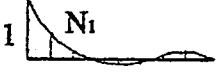
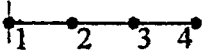
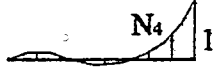
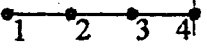
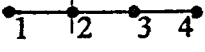
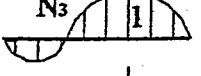
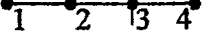


Figure 4.7. Shape function of quadratic linear element for node 2

The Lagrange shape functions in terms of Cartesian coordinates of a variety of linear elements are listed in Table 4.1.

Table 4.1. Lagrange Shape Functions for one-dimensional elements

Shape of the Function	Origin at the ends $\xi=0$ $\xi=+1$ $\eta=+1$ $\eta=0$ 	Origin at midpoint $\xi=-1$ $\xi=0$ $\xi=+1$ 
   	$N_1 = (1-\xi) = \eta$ $N_2 = (1-\eta) = \xi$	$N_1 = \frac{1}{2}(1-\xi\xi_i)$ $N_2 = \frac{1}{2}(1+\xi)$
   	$N_1 = (1-\xi)(1-2\xi) = -\eta(1-2\eta)$ $N_3 = -\xi(1-2\xi) = (1-\eta)(1-2\eta)$	$N_1 = \frac{1}{2}(1+\xi\xi_i)\xi\xi_i$ $N_3 = \frac{1}{2}(1+\xi)\xi$
 	$N_2 = 4\xi(1-\xi) = 4\xi\eta$	$N_2 = \frac{1}{2}(1-\xi^2)$
   	$N_1 = \frac{1}{2}(1-\xi)(2-3\xi)(1-3\xi) = \frac{1}{2}\eta(1-3\eta)(2-3\eta)$ $N_4 = \frac{1}{2}\xi(1-3\xi)(2-3\xi) = \frac{1}{2}(1-\eta)(2-3\eta)(1-3\eta)$	$N_1 = \frac{1}{16}(1+\xi\xi_i)(9\xi^2-1)$ $N_4 = \frac{1}{16}(1+\xi)(9\xi^2-1)$
   	$N_2 = \frac{9}{2}\xi(1-\xi)(2-3\xi) = -\frac{9}{2}\eta(1-\eta)(1-3\eta)$ $N_3 = -\frac{9}{2}\xi(1-\xi)(1-3\xi) = \frac{9}{2}\eta(1-\eta)(2-3\eta)$	$N_2 = \frac{9}{16}(1-\xi^2)(1-3\xi)$ $N_3 = \frac{9}{16}(1-\xi^2)(1+3\xi)$

4.5. Higher Order Triangular Elements

The shape functions of a linear triangle are discussed earlier in Section 3.3.. The nodal numbering of higher order triangles is shown in Figure 4.8. For the linear triangle the shape functions are simply the area coordinates. Thus,

$$N_1 = \xi \quad N_2 = \eta \quad N_3 = \zeta \quad (4.15)$$

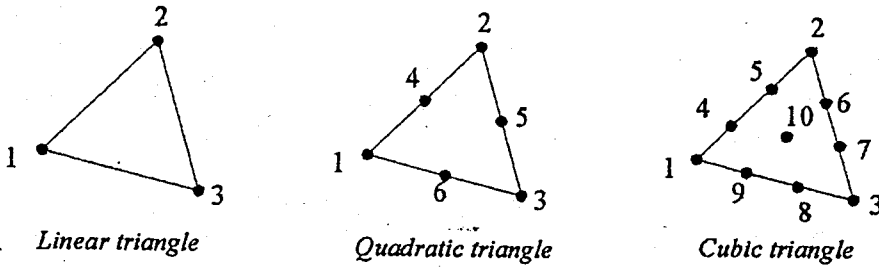


Figure 4.8. Nodal numbering in higher order triangles

The Lagrangian shape functions of a quadratic triangular element, from Table 4.1, are

$$\begin{aligned}
 N_1 &= \xi(2\xi - 1) & N_4 &= 4\xi\eta \\
 N_2 &= \eta(2\eta - 1) & N_5 &= 4\eta\zeta \\
 N_3 &= \zeta(2\zeta - 1) & N_6 &= 4\xi\zeta
 \end{aligned} \quad (4.16)$$

The Lagrangian shape functions of a cubic triangle, shown in Figure 4.8, are obtained from Table 4.1 in the same manner. The nodes 4 and 5 are located at one-third of the edge 12 with similar locations for the nodes 6 and 7, and 8 and 9 at the edges 23 and 31, respectively. The node 10 is at the centroid of the triangle. The general form of the Lagrangian shape function for a cubic triangular element for the corner, midside and interior nodes are

$$\left. \begin{aligned}
 N_1 &= \frac{1}{2} \xi (3\xi - 1)(3\xi - 2) & \text{corner node} \\
 N_4 &= \frac{9}{2} \xi \eta (3\xi - 1) & \text{midside node} \\
 N_{10} &= 27\xi\eta\zeta & \text{interior node}
 \end{aligned} \right\} \quad (4.17)$$

4.6. Higher Order Rectangular Element

The Lagrange interpolation polynomials for one-dimensional problems may be used repeatedly in order to construct shape functions for two or three- dimensional problems. For instance, in two dimensions, the product of Lagrange interpolation polynomials in x and y directions may be used to represent the shape functions of a rectangular element.

In the two-dimensional problems, we also consider Serendipity family. Universal Serendipity elements are defined as isoparametric elements having linear, quadratic and cubic node configurations at their edges in an arbitrary manner [26]. The general expression for Serendipity family is that there is no node inside of the element. The functions are similar to the Lagrange shape functions.

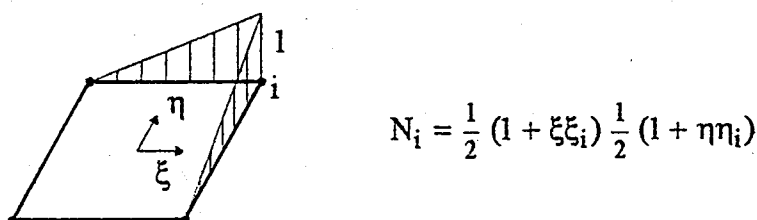


Figure 4.9. Shape function of a linear rectangular element

For a linear rectangular element, the shape functions are the same for both the Serendipity and Lagrange family as shown in Figure 4.9.

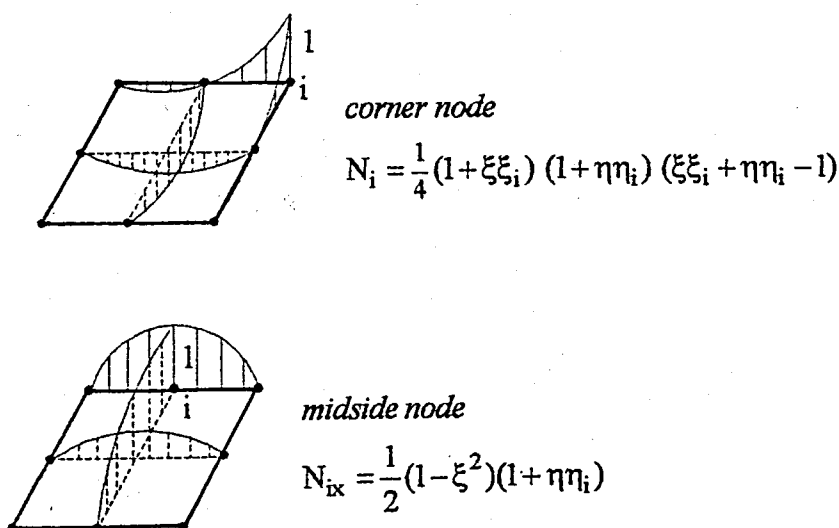


Figure 4.10. Serendipity shape functions of a quadratic element

The Serendipity shape functions of a quadratic rectangular element with eight nodes are shown in Figure 4.10. The Lagrangian shape functions of a quadratic rectangular element are shown in Figure 4.11. Similarly, the Serendipity and Lagrangian shape functions for a cubic rectangular element are shown in Figure 4.12, and 4.13, respectively.

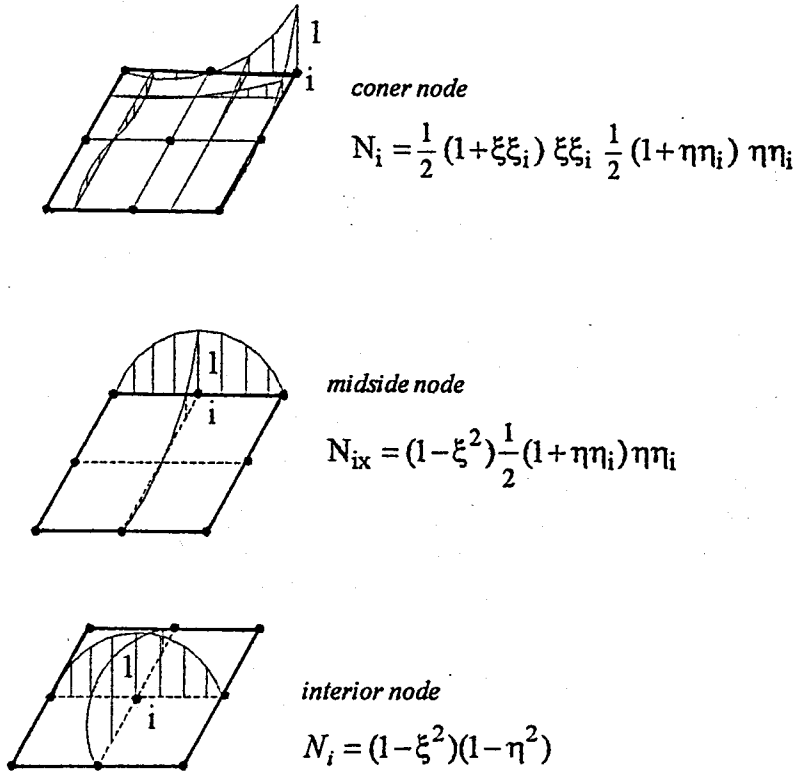


Figure 4.11. Lagrangian shape functions for a quadratic element

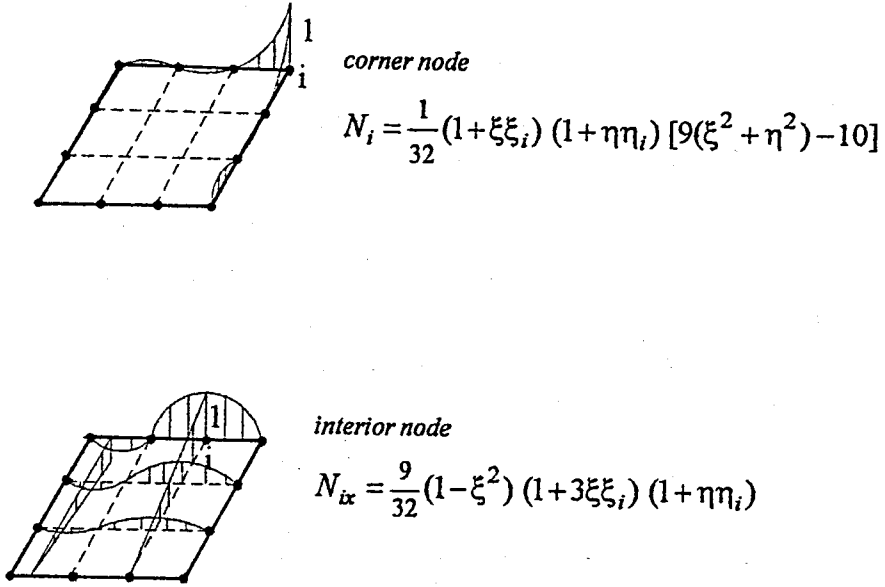


Figure 4.12. Serendipity shape functions for a cubic element

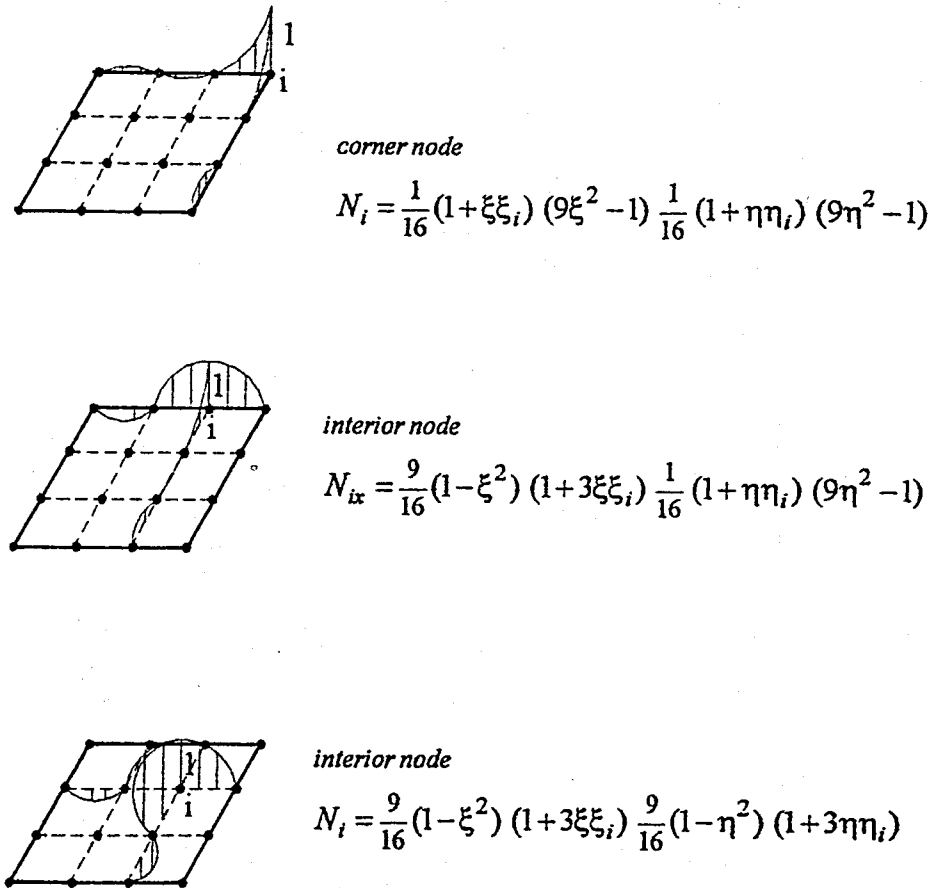


Figure 4.13. Lagrangian shape functions for a cubic element

4.7. Tetrahedron Element

The three-dimensional counterpart of an isoparametric triangular element is a tetrahedral element. The tetrahedral element is four-sided and may be classified as linear (4 nodes), quadratic (10 nodes), or cubic (20 nodes) as shown in Figure 4.14.

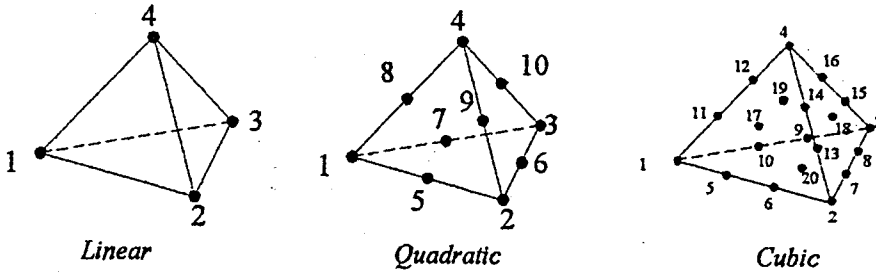


Figure 4.14. Nodal numbering in a tetrahedron

The natural or tetrahedral coordinates L_1 , L_2 , L_3 , and L_4 of a tetrahedron are related to the Cartesian coordinates

$$\begin{Bmatrix} 1 \\ x \\ y \\ z \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ z_1 & z_2 & z_3 & z_4 \end{bmatrix} \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \\ L_4 \end{Bmatrix} \quad (4.18)$$

For a linear tetrahedron the shape functions are

$$\begin{aligned} N_1 &= L_1 \\ N_2 &= L_2 \\ N_3 &= L_3 \\ N_4 &= L_4 \end{aligned} \quad (4.19)$$

For a quadratic tetrahedron, there will be 30 nodal unknowns, three at each of ten nodes indicated in Figure 4.14(b). The shape functions of a quadratic tetrahedron at the corner, interior and midside nodes from Table 4.1 are

$$\left. \begin{aligned} N_1 &= L_1(2L_1 - 1) && \text{corner node} \\ N_5 &= L_1 L_2 && \text{midside node} \end{aligned} \right\} \quad (4.20)$$

The cubic interpolation model involves twenty nodal unknowns and the nodal shape functions for corner nodes are

$$N_1 = \frac{1}{2} L_1 (3L_1 - 1)(3L_1 - 2) \quad (4.21)$$

and the shape function for the midside node 5 is

$$N_5 = \frac{9}{2} L_1 L_2 (3L_1 - 1) \quad (4.22)$$

and for the interior node 17

$$N_{17} = 27 L_1 L_2 L_4 \quad (4.23)$$

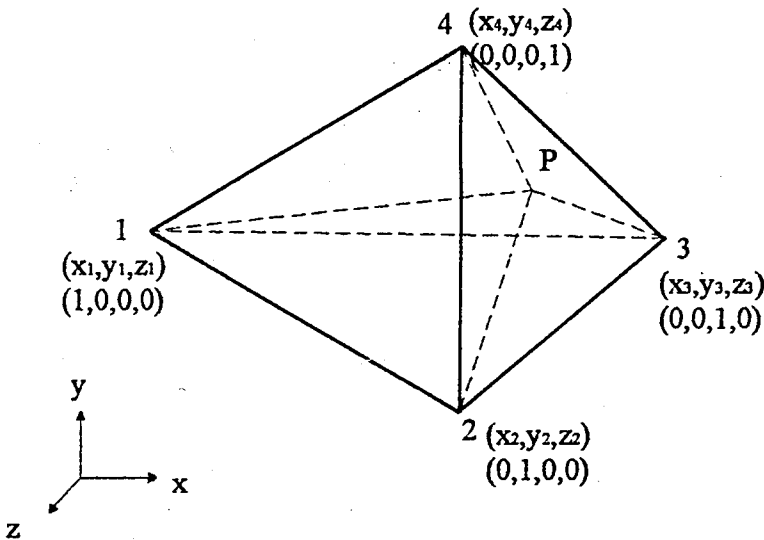


Figure 4.15. Natural coordinates for a tetrahedron

4.8. Serendipity Hexahedral Element

In the hexahedral elements, a consistent node-numbering scheme must be followed for defining the connectivity. For an eight-node hexahedron, the mapping is considered onto a cube of 2 unit sides placed symmetrically with ξ , η , ζ coordinates as shown in Figure 4.16.

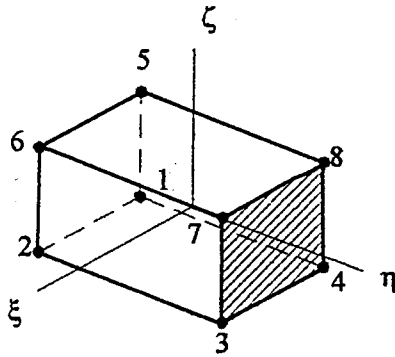


Figure 4.16. Natural coordinates of hexahedron

Natural coordinates for a hexahedral element can be established analogous to the case of a quadrilateral element. Thus the origin of the natural coordinates system is taken at the intersection of the lines joining the mid-points of opposite faces of the hexahedron and define the sides by $\xi = \pm 1$, $\eta = \pm 1$, $\zeta = \pm 1$. The relation between natural and Cartesian coordinates is given by [1]

$$x = \sum_{i=1}^8 N_i x_i \quad y = \sum_{i=1}^8 N_i y_i \quad z = \sum_{i=1}^8 N_i z_i \quad (4.24)$$

where

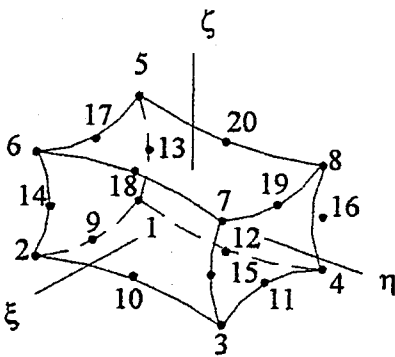
$$N_i = \frac{1}{8} (1 + \xi \xi_i) (1 + \eta \eta_i) (1 + \zeta \zeta_i) \quad (4.25)$$

The equation (4.25) is the shape function of the linear interpolation model for a hexahedron. A Serendipity quadric hexahedron element involves twenty nodes which are the sum of eight-corner nodes and twelve midside nodes as shown in Figure 4.17. The Serendipity shape functions for the corner nodes are

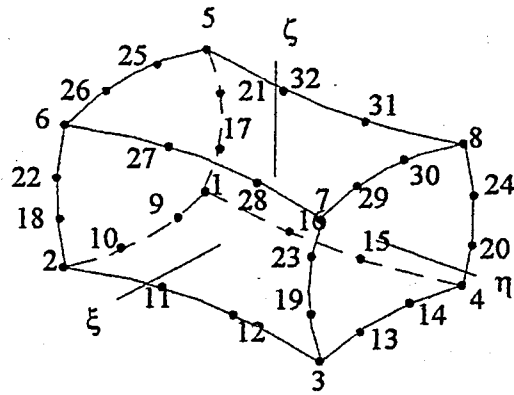
$$N_i = \frac{1}{8} (1 + \xi \xi_i) (1 + \eta \eta_i) (1 + \zeta \zeta_i) (\xi \xi_i + \eta \eta_i + \zeta \zeta_i - 2) \quad (4.26)$$

For mid-side nodes

$$N_9 = \frac{1}{4} (1 - \xi^2) (1 + \eta \eta_i) (1 + \zeta \zeta_i) \quad (4.27)$$



Quadratic hexahedron



Cubic hexahedron

Figure 4.17. Nodal numbering in Serendipity hexahedron

The shape functions of a Serendipity cubic hexahedron, which has 32 nodes with eight corner nodes plus twenty-four nodes at one-third points along the edges, are for the specific nodes are [1]

$$N_1 = \frac{1}{64} (1 + \xi \xi_i) (1 + \eta \eta_i) (1 + \zeta \zeta_i) (9\xi^2 + 9\eta^2 + 9\zeta^2 - 19)$$

$$N_9 = \frac{9}{64} (1 - \xi^2) (1 + 9\xi \xi_i) (1 + \eta \eta_i) (1 + \zeta \zeta_i)$$

(4.30)

$$N_{11} = \frac{9}{64} (1 - \eta^2) (1 + \xi \xi_i) (1 + 9\eta \eta_i) (1 + \zeta \zeta_i)$$

$$N_{17} = \frac{9}{64} (1 - \zeta^2) (1 + \xi \xi_i) (1 + \eta \eta_i) (1 + 9\zeta \zeta_i)$$

5. CASE STUDY NO. 1

(Plane Stress Problem)

5.1. Data for the Plane Stress Problem

The purpose of this section is to show how higher order finite elements can provide relatively more accurate predictions of stresses, strains and displacements. The plane stress problem illustrated in Figure 5.2 is used to test the plane stress element. It models an isotropic rectangular plate with uniform thickness under a force P applied at the end. The length, thickness, height, Young's modulus, and Poisson's ratio of the problem are L , t , h , E and ν , respectively. This plane stress problem has the following properties:

Beam Dimension: $8 \times 2 \times 0.20$ m

Force: $P = 40$ kN

Young's Modulus: $E = 30.10^6$ kN/m²

Poisson's ratio: $\nu = 0.3$

Thickness: $t = 0.20$ m

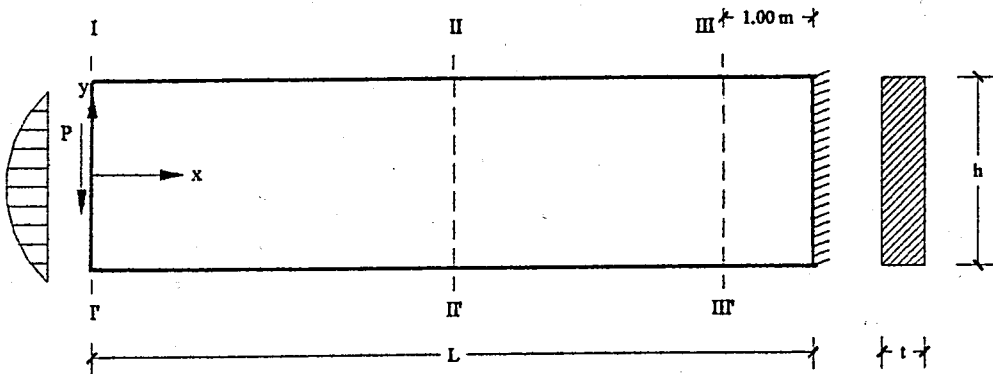


Figure 5.1. Cantilever beam with a point load at the tip

The force P , will be distributed parabolically along the section I-I'. The values of the distribution for nodes are obtained by the following equation [29]

$$P = \int_{-\frac{h}{2}}^{+\frac{h}{2}} \frac{2P}{h^3} \left(3y^2 - \frac{3}{4}h^2 \right) dy \quad (5.1)$$

When we substitute different values of height into equation (5.1), we obtain nodal forces for special points at section I-I', expressed as

$$\begin{aligned} P_{(y=0)} &= \int_{-0.125}^{0.125} 10(3y^2 - 3) dy = -7.49094 \text{ kN} \\ P_{(y=\pm 0.25)} &= \int_{0.125}^{0.375} 10(3y^2 - 3) dy = -6.99219 \text{ kN} \\ P_{(y=\pm 0.5)} &= \int_{0.375}^{0.625} 10(3y^2 - 3) dy = -5.58594 \text{ kN} \\ P_{(y=\pm 0.75)} &= \int_{0.625}^{0.875} 10(3y^2 - 3) dy = -3.24219 \text{ kN} \\ P_{(y=\pm 1)} &= \int_{0.875}^{1.00} 10(3y^2 - 3) dy = -0.44922 \text{ kN} \end{aligned} \quad (5.2)$$

5.2. The Exact Solutions

The exact values of stresses and strains for sections II-II' and III-III', also the displacements for section I-I' and II-II' are determined according to the following formulas [29]

for stress

$$\sigma_x = -\frac{Pxy}{I_{zz}}, \quad \sigma_y = 0, \quad \tau_{xy} = -\frac{P}{2I_{zz}} \left(\frac{h^2}{4} - y^2 \right) \quad (5.3)$$

for strain

$$\varepsilon_x = -\frac{Pxy}{EI_{zz}}, \quad \varepsilon_y = -\nu\varepsilon_x, \quad \gamma_{xy} = -\frac{P}{4GI_{zz}}\left(\frac{h^2}{4} - y^2\right) \quad (5.4)$$

for deflections

$$u = -\frac{Pyx^2}{2EI_{zz}} - \frac{\nu Py^3}{6EI_{zz}} + \frac{Py^3}{6GI_{zz}} + y\left(\frac{PL^2}{2EI_{zz}} - \frac{Ph^2}{8GI_{zz}}\right) \quad (5.5)$$

$$v = \frac{\nu Px^2}{2EI_{zz}} + \frac{Px^3}{6EI_{zz}} - \frac{PL^2x}{2EI_{zz}} + \frac{PL^3}{3EI_{zz}} + \frac{PL}{A_{1.2}G}$$

where the moment of inertia is

$$I = \frac{th^3}{12} = 0.13333 \text{ m}^4 \quad (5.6)$$

and shear modulus is

$$G = \frac{E}{2(1+\nu)} = 125.10^5 \text{ kN/m}^2 \quad (5.7)$$

The numerical values of stresses, strains, and deflections are all summarized in Table 5.1 and 5.2

Table 5.1. Stresses and strains at section II-II' and III-III'

	x (m)	y (m)	σ_x (kN/m ²)	σ_y (kN/m ²)	τ_{xy} (kN/m ²)	ϵ_x	ϵ_y	γ_{xy}
II-II'	4	1	1200	0	0	0,00004	-0,000008	0
	4	0,75	900	0	65,625	0,00003	-0,000006	0,000002625
	4	0,5	600	0	112,5	0,00002	-0,000004	0,0000045
	4	0,25	300	0	140,625	0,00001	-0,000002	0,000005625
	4	0	0	0	150	0	0	0,000006
III-III'	7	1	2100	0	0	0,00007	-0,000014	0
	7	0,75	1575	0	65,625	0,0000525	-0,0000105	0,000002625
	7	0,5	1050	0	112,5	0,000035	-0,000007	0,0000045
	7	0,25	525	0	140,625	0,0000175	-0,0000035	0,000005625
	7	0	0	0	150	0	0	0,000006

Table 5.2. Deflections at section I-I' and II-II'

	x (m)	y (m)	u (m)	v (m)
II-II'	0	1	0,0003117	0,001783467
	0	0,75	0,0002325	0,001783467
	0	0,5	0,0001545	0,001783467
	0	0,25	0,0000771	0,001783467
	0	0	0	0,001783467
III-III'	4	1	0,0002317	0,000579733
	4	0,75	0,0001725	0,000576233
	4	0,5	0,0001145	0,000573733
	4	0,25	0,0000571	0,000572233
	4	0	0	0,000571733

5.3. Finite Element Types used in the Analyses

The plane stress elements TPM3, TPM6, QPM4 and QPM8 in the LUSAS¹ package program are formulated by assuming that the variation of plane direct stress and shear stresses is negligible [30].

¹ LUSAS, London University Structural Analysis System, Finite Element Analysis Ltd., Forge House, 66 High Street, Kingston Upon Thames, Surrey, KT1 1HN, England.

The plane stress elements are suitable for analyzing structures which are thin in the out of plane direction. Types of the elements used in this example are illustrated in Table 5.3. These elements are a family of 2-dimensional isoparametric elements with higher order models capable of modeling the curved boundaries. The element formulations are based on the standard isoparametric approach. The variation of the stresses within an element can be regarded as constant for lower order elements, and linear for the higher models [31]. 2-dimensional plane stress continuum element with enhanced strains is an isoparametric element with an assumed strain field. This mixed assumed strain element demonstrates a superior performance to QPM4. The strain field for this element consists of two parts: the compatible strains derived from an assumed displacement field and the assumed enhanced strains. Appendix A expresses these enhanced strains [30].

The cantilever beam firstly subdivided into a coarse mesh of 2 by 4. Step by step, the mesh sizes are increased, until the short side of the problem is subdivided into forty parts and the long side into eighty parts. Other mesh sizes are also obtained by subdividing the short and the long sides into 4 and 8 parts and also 16 and 32 parts, respectively. Mesh sizes are shown in Table 5.4.

Table 5.3. Plane stress elements used in the solutions

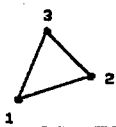
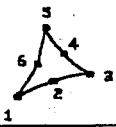
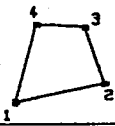
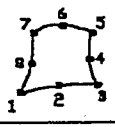
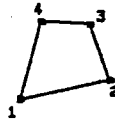
Name	Geometry	Title	Node Number	Freedom s
TPM 3		PLANE STRESS continuum element in 2D	3	U,V
TPM 6		PLANE STRESS continuum element in 2D	6	U,V
QPM 4		PLANE STRESS continuum element in 2D	4	U,V
QPM 8		PLANE STRESS continuum element in 2D	8	U,V
QPM4M		PLANE STRESS continuum element in 2D with enhanced strains	4	U,V

Table 5.4. Mesh sizes of Plane Stress Elements

Division of Short Side	Division of Long Side	Total Number of Elements	
		TPM3, TPM6	QPM4, QPM4M, QPM8
2	4	16	8
4	8	64	32
16	32	1024	512
40	80	6400	3200

5.4. Discussion of Results

The plane stress problem is analyzed by utilizing the package program, LUSAS. The required solutions are obtained easily by running this program. As it is mentioned earlier, the coarse and fine mesh sizes are used to obtain the relative accuracy of results and the numbers of elements are shown in Table 5.4. An example for this mesh size of the cantilever beam, and also the comparison tables and figures are shown in Appendix B.

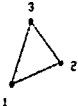
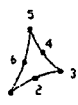
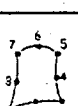
The stresses, strains are determined for sections II-II' and III-III', displacements are for sections I-I' and II-II'. The stress, strain and deflections are shown in Table 5.5 to Table 5.20. These results are plotted in two types of figures, in order to investigate the relative accuracies of different mesh sizes.. The first type is designed by showing the mesh sizes in horizontal axis and the physical parameters in the vertical axis. The second type is designed by showing the physical values horizontal axis and the coordinates in the vertical axis, for each mesh size.

The stress components σ_x , σ_y and τ_{xy} are determined for sections II-II' and the results are illustrated in Figure 5.2 to Figure 5.4. As it is seen in Table 5.5, the exact value of the maximum stress, σ_x , at section II-II' is 1200 kN/m^2 . The result using the linear triangular element (TPM3) for mesh size $2 / 4$ is 237.50 kN/m^2 . The result for the same mesh size using the quadratic higher order rectangular element (QPM8) is 1200.95 kN/m^2 , which is very close to the exact value. Thus, very accurate results may be obtained by using higher order elements even with a very coarse mesh size.

The figures of the comparisons of strains ε_x , ε_y and γ_{xy} at sections II-II' and III-III' are included in Figure B.34 to Figure B.63. When Figure B.19 is investigated, it is seen that all results approach the exact solution monotonically, as the mesh size becomes finer. Only the QPM8 and TPM6 elements yield the nearest values to the exact solution for a coarse mesh.

Figure B.64 to B.75 presented in Appendix B refer to the horizontal and vertical deflections at sections I-I' and II-II'. All figures demonstrate that the quadratic higher order triangular and higher order rectangular elements, and linear rectangular element with enhanced strains approach the exact solutions much faster than the linear elements.

Table 5.5. Comparison of maximum normal stress (σ_x) at II-II' (kN/m^2)

LUSAS 13. 4						Exact Solutions
Name	Geometry	2 / 4 *	4 / 8	16 / 32	40 / 80	
TPM3		237,498	678,713	1109,750	1170,660	1200
TPM6		1175,490	1185,850	1200,000	1199,860	1200
QPM4		757,528	1095,460	1195,150	1199,970	1200
QPM8		1200,950	1200,010	1200	1200	1200
QPM4M		1275,570	1198,590	1200	1200	1200

* Dividing the short side of the problem in 2 parts and the long side in 4 parts

Table 5.6. Comparison of stress (σ_y) at II-II' (kN/m^2)

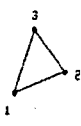
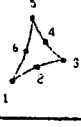
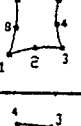
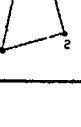
LUSAS 13. 4						Exact Solutions
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-31,31310	16,61380	-1,42056	-0,23247	0
TPM6		5,77268	-0,12707	0,00013	0,00003	0
QPM4		0	0	0	0	0
QPM8		0	0	0	0	0
QPM4M		0	0	0	0	0

Table 5.7. Compariosn of shear stress (τ_{xy}) at II-II' (kN/m^2)

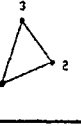
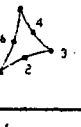
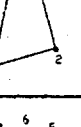
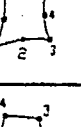
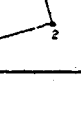
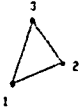
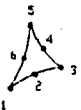
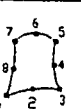
LUSAS 13. 4						Exact Solutions
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		82,729	119,070	147,284	149,555	150
TPM6		176,756	155,960	150,391	150,062	150
QPM4		-31,134	106,978	147,049	149,525	150
QPM8		171,495	155,794	150,391	150,062	150
QPM4M		100	137,896	149,219	149,875	150

Table 5.8. Comparison of maximum normal stress (σ_x) at III-III' (kN/m^2)

LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4 *	4 / 8	16 / 32	40 / 80	
TPM3		543,56	1201,50	1939,30	2044,41	2100
TPM6		1944,60	2055,50	2091,92	2092,39	2100
QPM4		1445,75	1903,49	2082,19	2092,08	2100
QPM8		2106,52	2062,18	2092,56	2092,48	2100
QPM4M		1944,73	2107,33	2089,57	2092,24	2100

* Dividing the short side of the problem in 2 parts and the long side in 4 parts

Table 5.9. Comparison of stress (σ_y) at III-III' (kN/m^2)

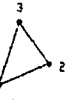
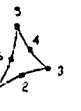
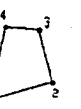
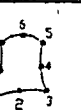
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		26,090	4,318	-1,760	-0,470	0
TPM6		-22,157	-16,325	-0,286	-0,041	0
QPM4		0	0	0	0	0
QPM8		0	0	0	0	0
QPM4M		0	0	0	0	0

Table 5.10. Comparison of shear stress (τ_{xy}) at III-III' (kN/m^2)

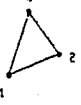
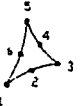
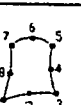
Name	Geometry	14				Exact Solutions
		2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		96,222	108,193	148,700	151,804	150
TPM6		176,811	168,956	153,038	152,478	150
QPM4		146,949	104,459	149,792	151,959	150
QPM8		214,644	184,002	153,107	152,482	150
QPM4M		100,000	128,623	152,343	152,356	150

Table 5.11. Comparison of strain (ϵ_x) at II-II'

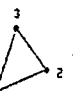
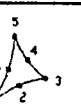
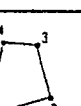
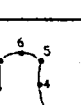
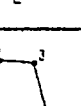
Name	Geometry	LUSAS 13. 4				Exact Solution
		2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		8,019E-06	2,259E-05	3,695E-05	3,900E-05	4,E-05
TPM6		3,921E-05	3,953E-05	3,997E-05	4,000E-05	4,E-05
QPM4		2,465E-05	3,614E-05	3,973E-05	3,996E-05	4,E-05
QPM8		3,997E-05	4,000E-05	4,000E-05	4,000E-05	4,E-05
QPM4M		4,249E-05	3,995E-05	4,000E-05	4,000E-05	4,E-05

Table 5.12. Comparison of strain (ε_y) at II-II'

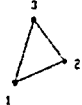
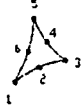
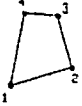
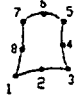
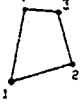
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-2,096E-06	-4,346E-06	-7,192E-06	-7,708E-06	-8,E-06
TPM6		-7,971E-06	-7,914E-06	-7,995E-06	-7,999E-06	-8,E-06
QPM4		-2,052E-06	-5,410E-06	-7,450E-06	-7,792E-06	-8,E-06
QPM8		-7,689E-06	-7,991E-06	-8,000E-06	-8,000E-06	-8,E-06
QPM4M		-8,338E-06	-7,991E-06	-8,000E-06	-8,000E-06	-8,E-06

Table 5.13. Comparison of Shear strain (γ_{xy}) at II-II'

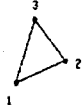
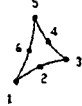
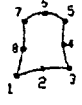
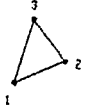
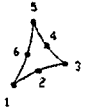
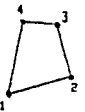
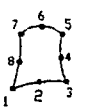
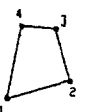
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		1,611E-06	6,404E-07	-9,494E-08	-6,533E-08	0
TPM6		2,679E-06	5,452E-07	3,421E-08	5,473E-09	0
QPM4		-3,024E-06	2,688E-06	1,209E-06	5,242E-07	0
QPM8		2,280E-06	4,875E-07	3,125E-08	5,000E-09	0
QPM4M		8,000E-06	4,968E-06	1,438E-06	5,900E-07	0

Table 5.14. Comparison of strain (ϵ_x) at III-III'

LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4 *	4 / 8	16 / 32	40 / 80	
TPM3		1,782E-05	3,991E-05	6,458E-05	6,812E-05	7,E-05
TPM6		6,483E-05	6,856E-05	6,973E-05	6,975E-05	7,E-05
QPM4		4,721E-05	6,321E-05	6,923E-05	6,966E-05	7,E-05
QPM8		7,059E-05	6,927E-05	6,975E-05	6,975E-05	7,E-05
QPM4M		6,502E-05	7,075E-05	6,966E-05	6,974E-05	7,E-05

* Dividing the short side of the problem in 2 parts and the long side in 4 parts

Table 5.15. Comparison of strain (ϵ_y) at III-III'

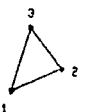
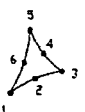
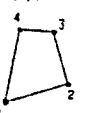
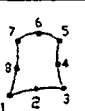
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-2,126E-06	-7,321E-06	-1,260E-05	-1,348E-05	-1,40E-05
TPM6		-1,302E-05	-1,390E-05	-1,395E-05	-1,395E-05	-1,40E-05
QPM4		-4,755E-06	-1,147E-05	-1,299E-05	-1,359E-05	-1,40E-05
QPM8		-1,588E-05	-1,641E-05	-1,394E-05	-1,395E-05	-1,40E-05
QPM4M		-1,393E-05	-1,659E-05	-1,395E-05	-1,395E-05	-1,40E-05

Table 5.16. Comparison of shear strain (γ_{xy}) at III-III'

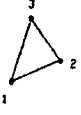
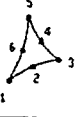
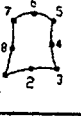
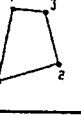
Name	Geometry	Lusas 13.4				Exact Solution
		2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		1,445E-06	-2,495E-06	-1,440E-06	-6,042E-07	0
TPM6		-6,244E-07	-6,619E-07	1,178E-08	1,379E-09	0
QPM4		1,368E-05	3,433E-06	1,144E-06	4,876E-07	0
QPM8		-1,172E-06	-4,195E-06	4,184E-08	2,835E-09	0
QPM4M		8,000E-06	5,710E-06	1,286E-06	5,423E-07	0

Table 5.17. Comparison of maximum horizontal deflection at I-I' (m) (10^{-4})

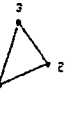
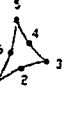
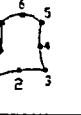
Name	Geometry	Lusas 13.4				Exact Solutions
		2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-1,084	-2,218	-3,121	-3,195	-3,117
TPM6		-3,204	-3,207	-3,210	-3,210	-3,117
QPM4		-2,070	-2,891	-3,187	-3,206	-3,117
QPM8		-3,205	-3,208	-3,210	-3,210	-3,117
QPM4M		-3,191	-3,200	-3,209	-3,210	-3,117

Table 5.18. Comparison of maximum vertical deflection at I-I' (m) (10⁻⁴)

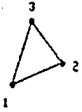
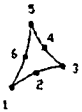
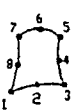
Lusas 13. 4						Exact Solutions
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-0,649	-1,239	-1,730	-1,771	-1,783
TPM6		-1,764	-1,776	-1,779	-1,779	-1,783
QPM4		-1,167	-1,600	-1,766	-1,777	-1,783
QPM8		-1,775	-1,777	-1,779	-1,779	-1,783
QPM4M		-1,727	-1,765	-1,778	-1,779	-1,783

Table 5.19. Comparison of maximum horizontal deflection at II-II' (m) (10⁻⁴)

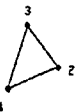
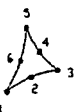
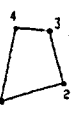
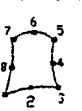
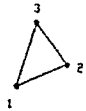
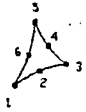
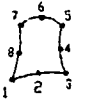
LUSAS 13.4						Exact Solutions
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-0,789	-1,662	-2,343	-2,399	-2,317
TPM6		-2,407	-2,408	-2,410	-2,410	-2,317
QPM4		-1,511	-2,168	-2,393	-2,407	-2,317
QPM8		-2,406	-2,408	-2,410	-2,410	-2,317
QPM4M		-2,391	-2,400	-2,409	-2,410	-2,317

Table 5.20. Comparison of maximum vertical deflection at II-II' (m) (10⁻⁴)

LUSAS 13.4						Exact Solutions
Name	Geometry	2 / 4	4 / 8	16 / 32	40 / 80	
TPM3		-2,318	-4,004	-5,564	-5,696	-5,797
TPM6		-5,615	-5,706	-5,722	-5,723	-5,797
QPM4		-3,919	-5,133	-5,680	-5,716	-5,797
QPM8		-5,698	-5,714	-5,723	-5,724	-5,797
QPM4M		-5,404	-5,651	-5,717	-5,722	-5,797

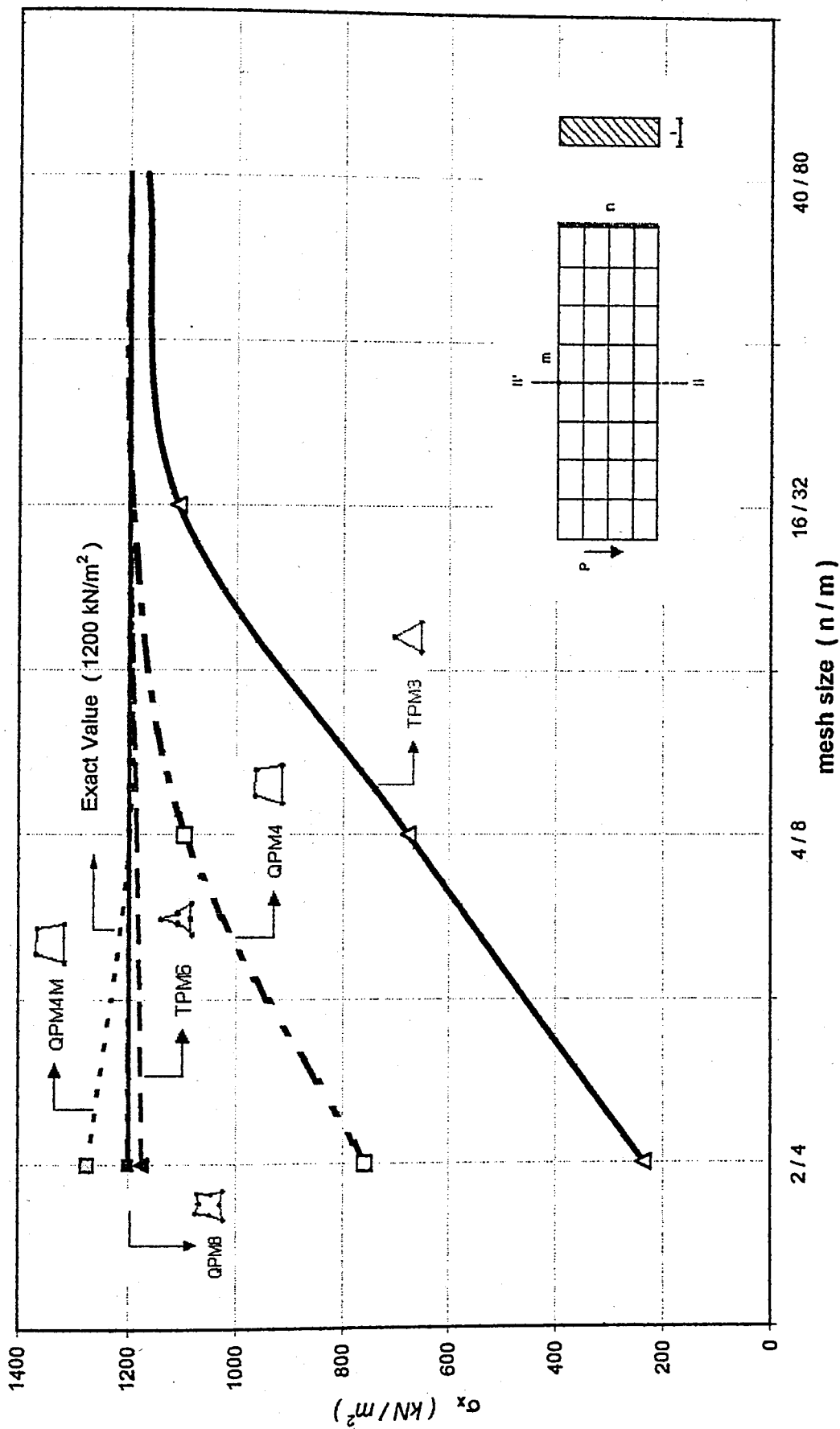


Figure 5.2. Maximum normal stress (σ_x) at section II-II'

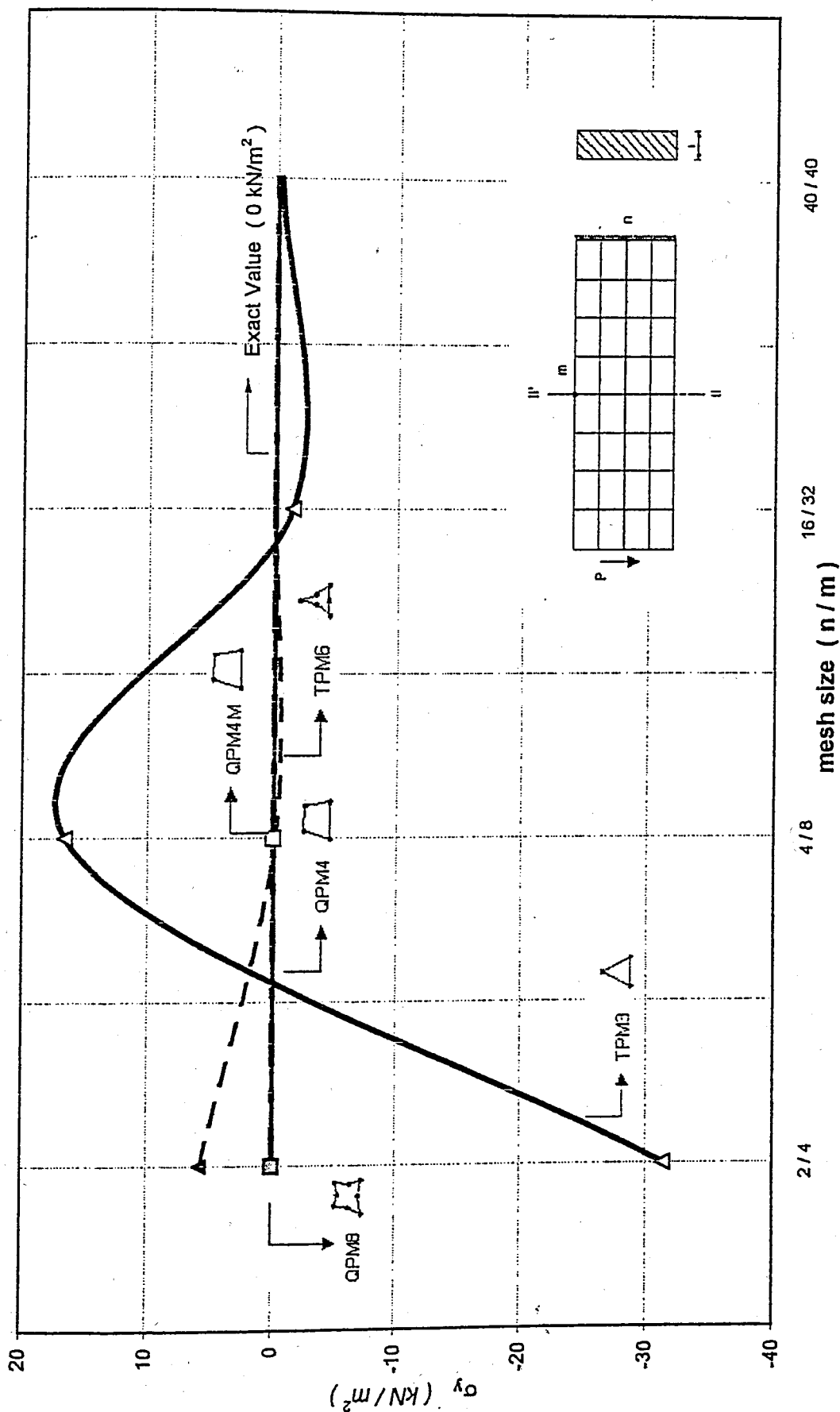


Figure 5.3. Maximum normal stress (σ_y) at section II-II'

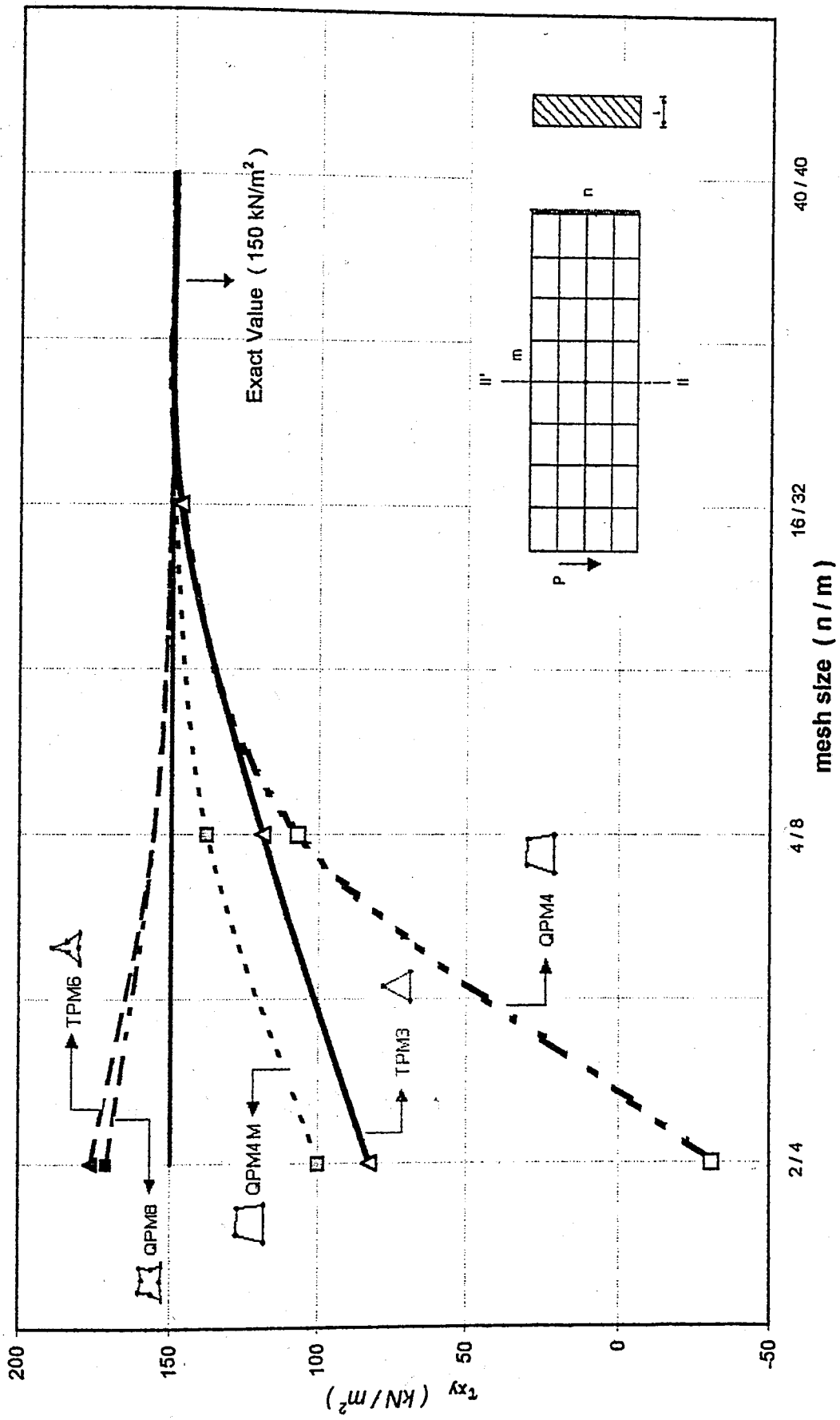


Figure 5.4. Maximum shear stress (τ_{xy}) at section II-II'

6. CASE STUDY NO. 2

(Circular Plate Bending Problem)

6.1. Data for Circular Plate Bending Problem

A circular plate with clamped edges all around is considered under a uniform loading as shown in Figure 6.1. The modulus of elasticity used is the same as used in the plane stress problem. The uniform load, thickness, radius are q , t and a , respectively. The properties of the problem are

Uniform Load: $q = 50 \text{ kN/m}^2$

Thickness: $t = 0.20 \text{ m}$

Radius of the plate: $a = 5 \text{ m}$

Young's modulus: $E = 30 \cdot 10^6 \text{ kN/m}^2$

Poisson's ratio: $\nu = 0.3$

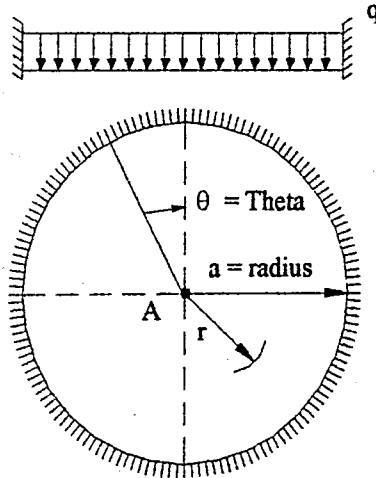


Figure 6.1. Clamped circular plate

6.2. The Exact Solutions

If a circular plate of radius a carries a load of intensity q uniformly distributed over the entire surface of the plate, the magnitude of the maximum deflection at the center of the plate is equal to [32]

$$w_{\max} = \frac{q a^4}{64 D} \quad (6.1)$$

where the flexural rigidity of the plate, D , is

$$D = \frac{E t^3}{12(1-\nu^2)} = 21\,978 \text{ kNm} \quad (6.2)$$

Bending moments for both radial and circumferential directions at the boundary of the plate is expressed as [32]

$$(M_r)_{r=a} = -\frac{q a^2}{8} \quad (M_t)_{r=a} = -\frac{\nu q a^2}{8} \quad (6.3)$$

At the center of the plate where $r=0$

$$M_r = M_t = \frac{q a^2}{16} (1+\nu) \quad (6.4)$$

The numerical results of vertical deflection and the bending moments are given in Table 6.1.

Table 6.1. Exact values of the vertical deflection and bending moments

LOCATION	w (m)	M_r (kNm)	M_t (kNm)
AT CENTER	-0,0222168	-101,5625	-101,5625
AT BOUNDARY	0	156,25	46,875

6.3. Finite Element Types used in the Analyses

The problem is solved by utilizing the package program LUSAS 13.4. In this section, the plate elements, TF3, QF4, TTF6 and QTF8 are used, as shown in Table 6.2 [31]. Linear triangular and linear quadrilateral elements, which are TF3 and QF4, respectively, are a family of thin plate flexure elements in 2-dimensions of which, the higher models are capable of modeling the curved boundaries. As is the case by the thin plate theory, the transverse shearing effects are excluded [31]. The isoflex family of thin plate bending elements is formed by applying the Krichoff constraints within the elements formulated using Mindlin plate bending assumptions [30]. Expanded information about the isoflex thin plates is expressed in Appendix C.

The quadratic higher order triangle and higher order quadrilateral elements, TTF6 and QTF8, are a family of thick plate flexure elements based on the Mindlin plate formulation. Transverse shear deformations are included [31]. The element formulations are based on an isoparametric approach. The isoparametric thick Mindlin plates are presented in Appendix D.

Four different mesh sizes are used for this circular plate-bending problem. The mesh sizes are shown in Table 6.3. Firstly, radial and circumferential directions are divided into 2 and 8 parts, respectively. Secondly, the mesh sizes are increased, gradually. The results are arranged from coarse to fine mesh sizes, and the comparisons are illustrated in the respective figures in this order. Mesh types and nodal numbers of the circular plate are presented in Appendix E.

Table 6.2. Plate bending elements used in the solutions

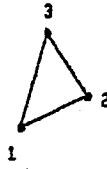
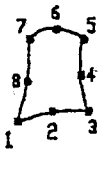
Name	Geometry	Title	Freedom
QF4		ISOFLEX thin plate flexure element in 2D (<i>Quadrilateral</i>)	w, θ_x, θ_y
TF3		ISOFLEX thin plate flexure element in 2D (<i>Triangular</i>)	w, θ_x, θ_y
TTF6		MINDLIN thick plate flexure element in 2D (<i>Triangular</i>)	w, θ_x, θ_y
QTF8		MINDLIN thick plate flexure element in 2D (<i>Quadrilateral</i>)	w, θ_x, θ_y

Table 6.3. Mesh sizes of plate bending elements

Division in Radial Direction	Division in Theta Angle	Total Number of Elements	
		TF3, TTF6	QF4, QTF8
2	8	32	16
4	16	128	64
7	28	392	196
25	100	5000	2500

6.4. Discussion of Results

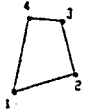
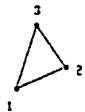
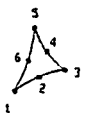
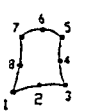
The vertical deflection and the moments, M_r , M_t , at the center as well as the boundary of the circular plate are calculated for the different mesh sizes by the help of LUSAS package program. The results are compared in Table 6.4 to Table 6.7 and these comparisons are also illustrated in Figure 6.2 to Figure 6.5.

The exact value of the vertical deflection at the center of the plate is -0.0222 m. The linear quadrilateral element, QF4 appropriate this value faster than the others. TF3 has a lower bound approach as shown in Figure 6.2. Other elements approach upper bound.

The results of the bending moments at the center and boundary vary due to the mesh sizes, are shown in Table 6.5 to Table 6.7. All elements result relatively in poor results with coarse mesh, but by refining the mesh size, more accurate solutions are obtained.

It is seen that the results approach closer and closer to the exact values as the mesh sizes become finer. But, the quadratic higher order triangular and higher order quadrilateral elements do not display satisfactory solutions. The reason for this phenomenon may be explained that these higher order elements, TTF6 and QTF8, are within a family of thick plate flexure elements based on the Mindlin plate formulation in which the transverse shear deformations are duly included in the formulations. Therefore, the exact values of deflections and moments for this problem are the solutions given by the higher order elements.

Table 6.4. Comparison of vertical deflection w , at the center (m)

LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 8 *	4 / 16	7 / 28	25 / 100	
QF4		-0,019813	-0,022060	-0,022222	-0,022220	-0,022217
TF3		-0,019494	-0,021670	-0,022051	-0,022204	-0,022217
TTF6		-0,017648	-0,022328	-0,022396	-0,022388	-0,022217
QTF8		-0,019005	-0,022744	-0,022627	-0,022402	-0,022217

* Division of the plate 2 parts in radial direction and 8 parts in theta direction

Table 6.5. Comparison of moment M_r , at the boundary (kNm)

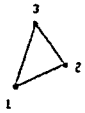
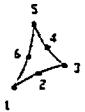
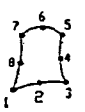
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 8	4 / 16	7 / 28	25 / 100	
QF4		138,531	153,209	153,470	156,207	156,25
TF3		134,685	150,916	154,306	155,934	156,25
TTF6		104,095	136,041	147,418	156,631	156,25
QTF8		164,486	159,189	157,293	156,331	156,25

Table 6.6. Comparison of moment M_b at the boundary (kNm)

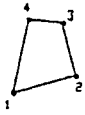
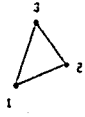
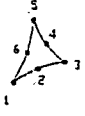
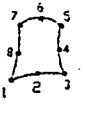
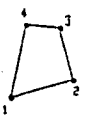
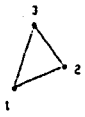
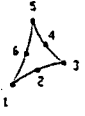
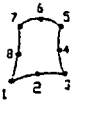
Name	Geometry	LUSAS 13. 4				Exact Solution
		2 / 8	4 / 16	7 / 28	25 / 100	
QF4		40,778	44,143	45,731	46,762	46,875
TF3		36,114	44,297	46,337	47,033	46,875
TTF6		26,492	38,355	43,290	46,606	46,875
QTF8		32,370	43,636	45,791	47,178	46,875

Table 6.7. Comparison of moment (M_r , M_t) at the center (kNm)

Name	Geometry	LUSAS 13. 4				Exact Solution
		2 / 8	4 / 16	7 / 28	25 / 100	
QF4		-117,413	-111,058	-106,203	-102,203	-101,5625
TF3		-115,519	-108,797	-104,884	-101,995	-101,5625
TTF6		-65,053	-107,404	-104,814	-101,819	-101,5625
QTF8		-42,930	-94,869	-102,760	-101,895	-101,5625

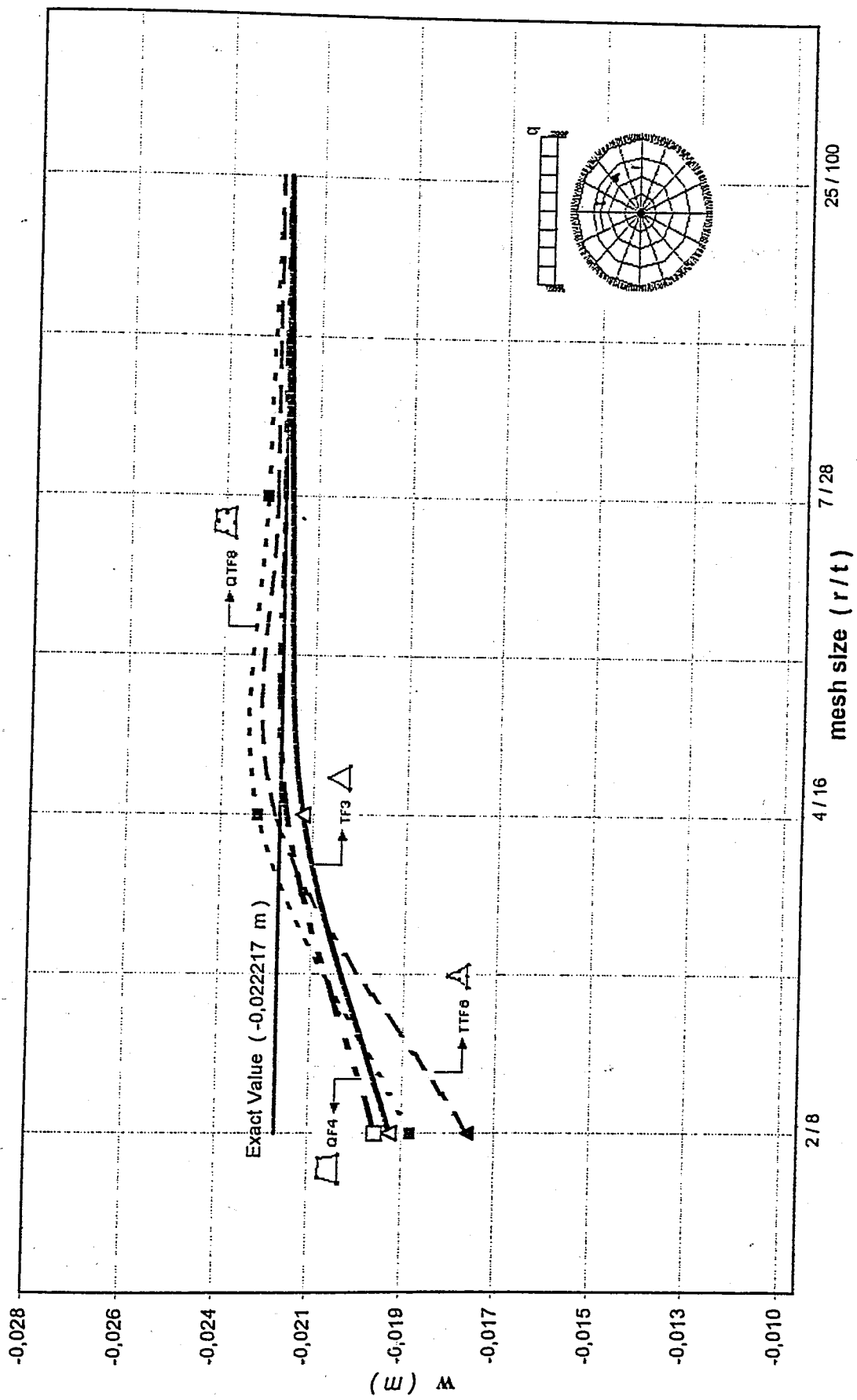


Figure 6.2. Vertical deflection w , at the center of the plate

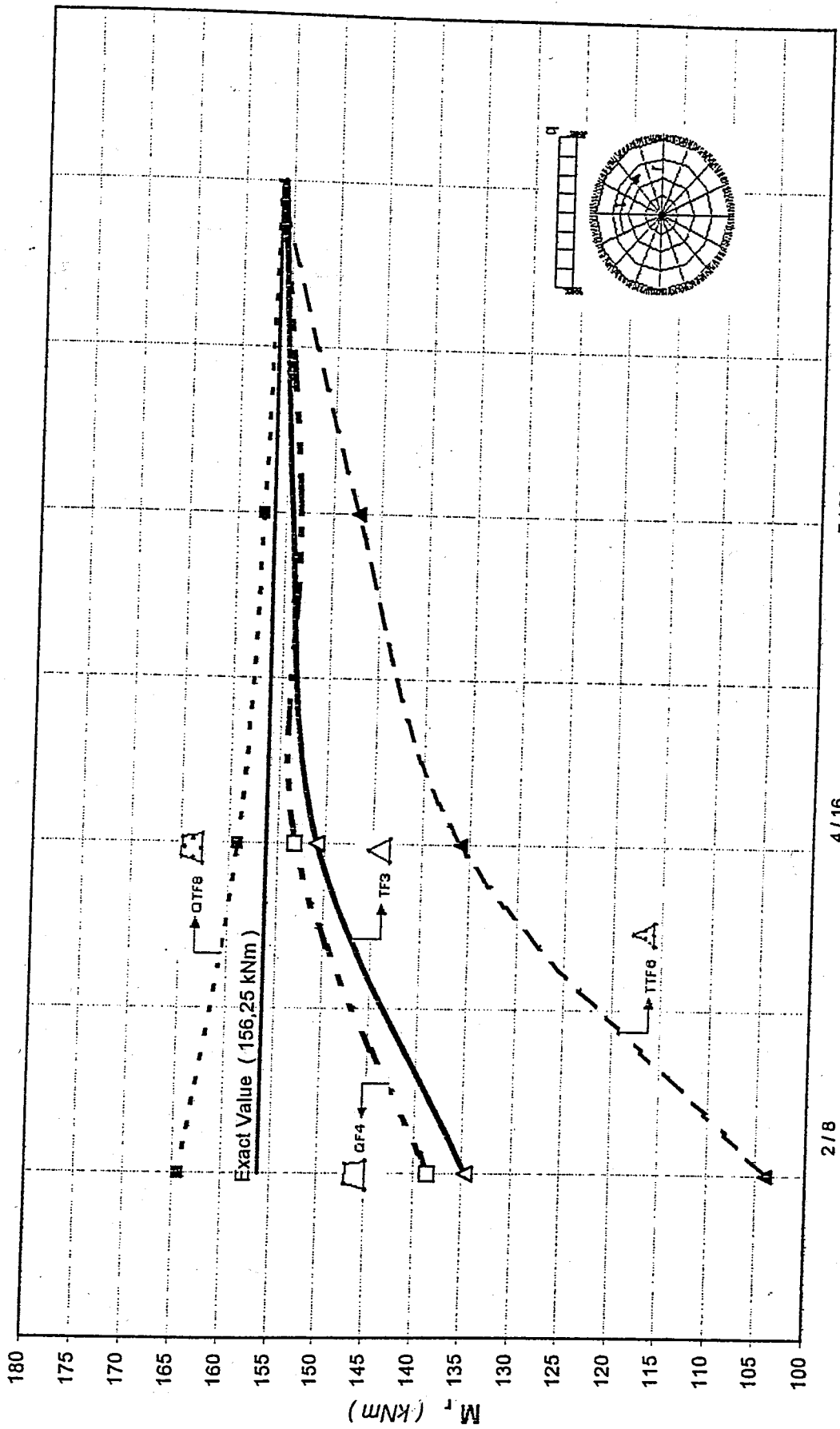


Figure 6.3. Moment M_r , at the boundary of the plate

2/8

4/16

mesh size (r/t)

7/28

25/100

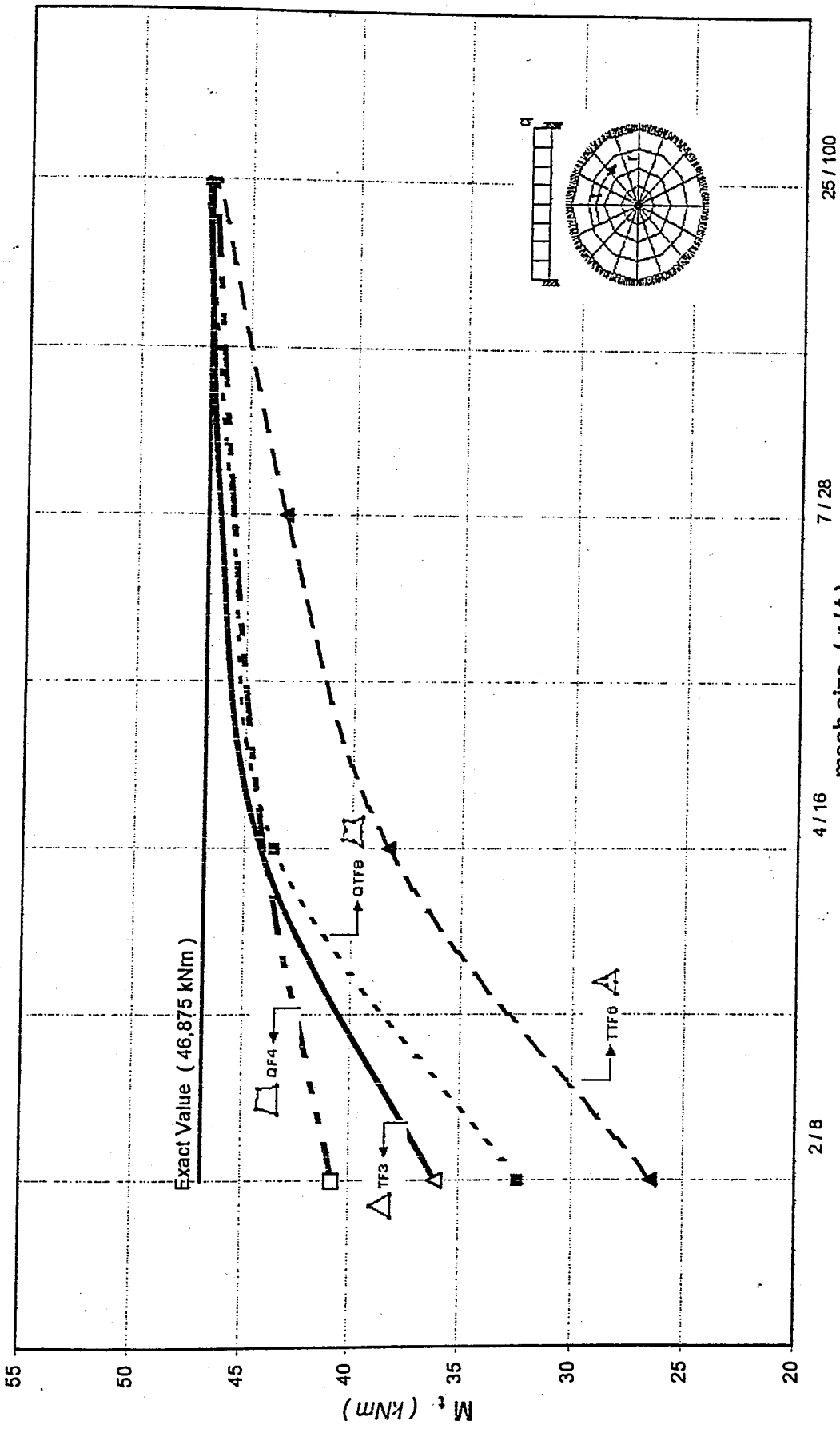


Figure 6.4. Moment M_t , at the boundary of the plate

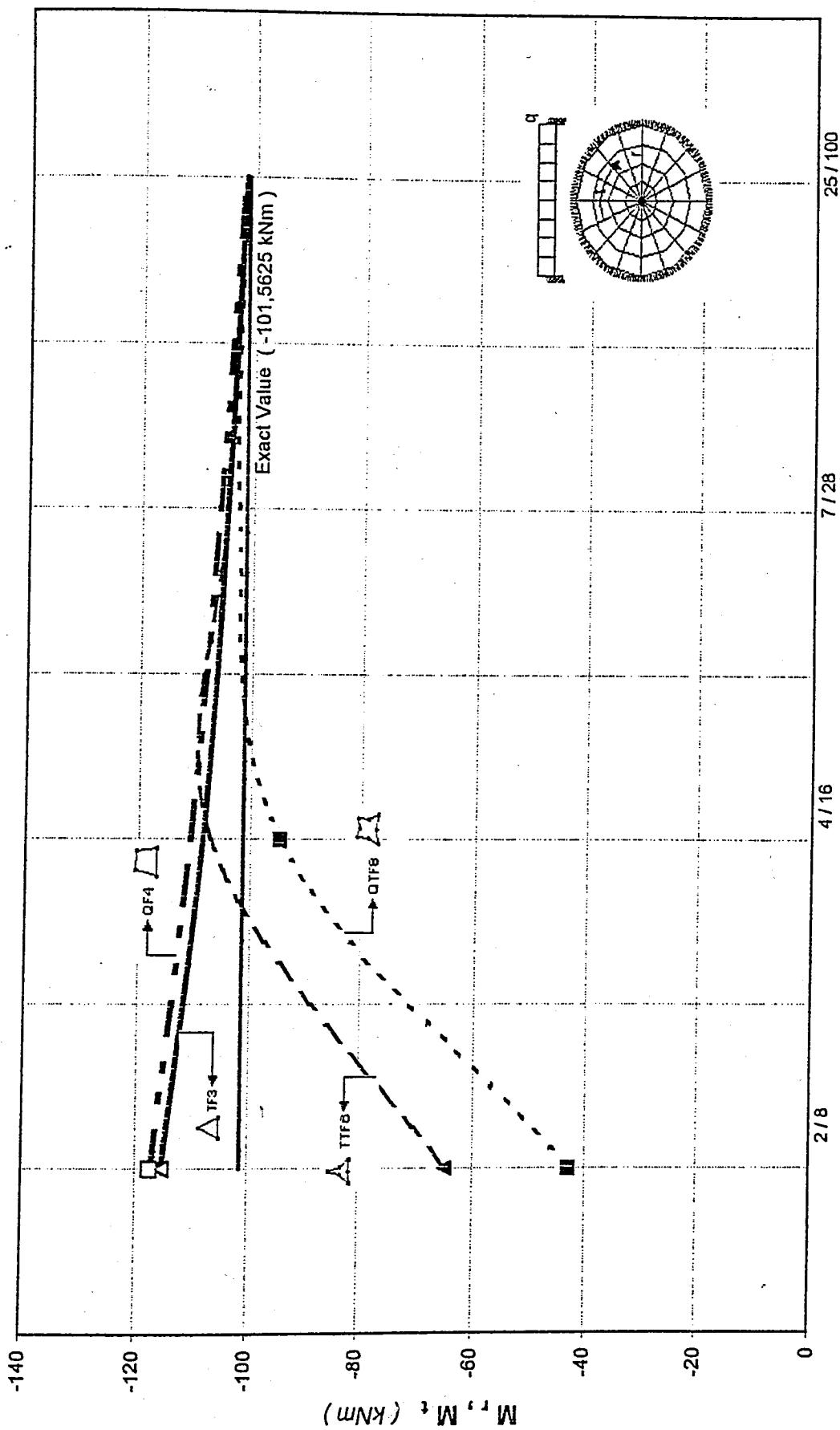


Figure 6.5. Moment (M_t, M_r) at the center of the plate

7. CASE STUDY NO. 3

(Rectangular Plate Bending Problem under Central Load)

7.1. Data for the Square Plate Bending Problem

The bending of a square plate with clamped edges under the action of a load P concentrated at the center is considered. The problem is illustrated in Figure 7.1. The width and the height of the plate, central load, thickness, Young's modulus, and Poisson's ratio of the problem are a , b , P , t , E and ν , respectively. This plate bending problem has the following data:

Dimensions of the Plate: $4 \times 4 \times 0.14$ m

Central Load: $P = 100$ kN

Young's modulus: $E = 30.10^6$ kN/m²

Poisson's ratio: $\nu = 0.3$

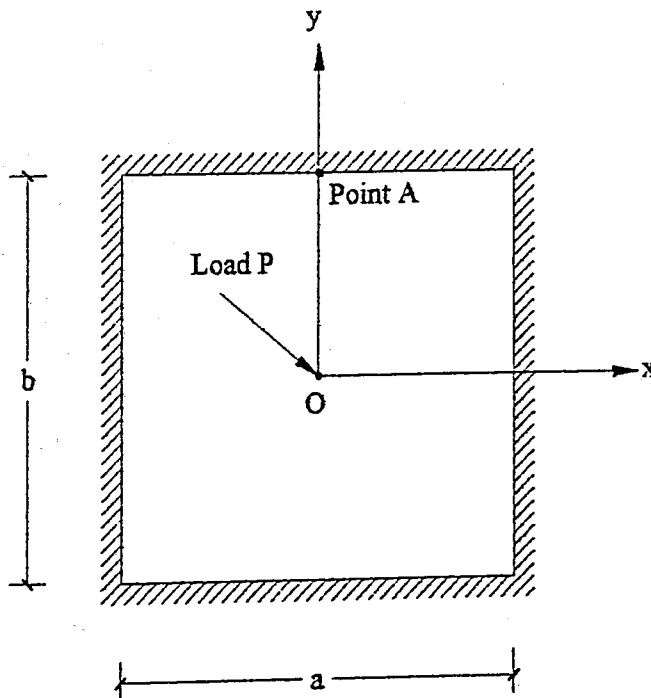


Figure 7.1. Square plate under central load

7.2. The Exact Solutions

The vertical deflection and the bending moments in x and y-directions at the center of the plate and also the bending moment in y-direction at point A is determined by means of the equations as follows [32].

The central vertical deflection is equal to

$$(w)_{x=y=0} = 0.0056 \frac{Pa^2}{D} = -0.001189 \text{ m} \quad (7.1)$$

where the flexural rigidity of the plate is expressed as

$$D = \frac{Et^3}{12(1-\nu^2)} = 7538.462 \text{ kNm} \quad (7.2)$$

The bending moment in the y-direction at point A is defined by

$$(M_y)_{x=0, y=b/2} = -0.1257P = 12.57 \text{ kNm} \quad (7.3)$$

Having the moments along the clamped edges, the corresponding deflections can be calculated. Superposing the deflections produced by the moments on the deflections of a simply supported plate, the deflections of the plate with built-in edges are obtained. By the same method of superposition the other information regarding, the deflections of plates with built-in edges under a central concentrated load can be obtained [32]. As a consequence, if the load P is distributed uniformly over the area of a small circle and rectangle, the bending moments at the center of the load area $x = y = 0$ can be obtained by combining the results valid for simply supported plates with some additional moments, which are [32]

$$(m_x = m_y)_{x=y=0} = -0.0536P \quad (7.4)$$

The shearing forces and bending moments become infinitely large at the point of application of the load. The stress distribution within a circle of small radius, c , with its center at the point of application of the load is substantially the same as that near the center of a centrally loaded circular plate [32]. The moments in both directions are equal because of the symmetry. The calculations of the bending moments are

$$M_x = \frac{P}{4\pi} \left[(1+\nu) \log \frac{2a \sin \frac{\pi\xi}{a}}{\pi c} + 1 \right] + \frac{\gamma_1 P}{4\pi} \quad (7.5)$$

$$M_y = \frac{P}{4\pi} \left[(1+\nu) \log \frac{2a \sin \frac{\pi\xi}{a}}{\pi c} + 1 \right] - (1-\nu-\gamma_2) \frac{P}{4\pi}$$

where $\xi = \frac{a}{2}$ and $\gamma_1 = -0.565$, $\gamma_2 = 0.135$ [25].

The different values of c that are close to zero are substituted into the equation (7.5). After a value of c smaller than $10^{-6} < c < 0.1$, the results become closer as shown in Figure 7.2.

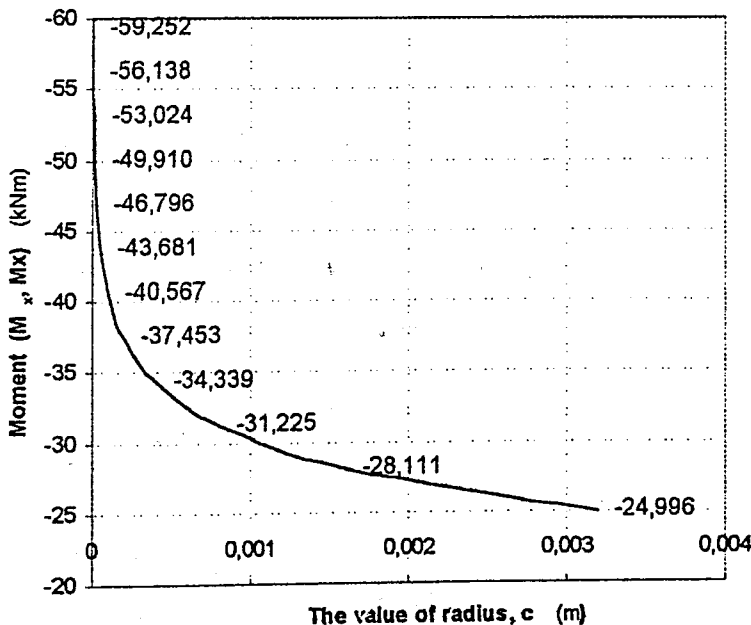


Figure 7.2. Bending moment at the center of the plate

The exact values of the vertical deflection and bending moments are summarized in Table 7.1.

Table 7.1. Vertical deflection and bending moments

AT THE CENTER		AT POINT A
w (m)	M _x , M _y (kNm)	M _y (kNm)
-0,001189	-49,9097	12,57

7.3. Finite Element Types used in the Analyses

The computer program used is the package program LUSAS 13.4. The quarter of the plate is designed due to two-way symmetry. The elements used are the same elements that are chosen in section 7.3. These plate elements are TF3, QF4, TTF6 and QTF8 as shown in Table 6.1 [31]. The higher order elements, TTF6 and QTF8, are a family of Serendipity family, which has no node at the center of the element.

In this section, four different mesh sizes are used again. The divisions of both sides are the same because of the equality of dimensions in both directions. The chosen mesh sizes are shown in Table 7.2. The analysis starts on a uniform coarse mesh which is the division of both sides in 2 parts. At last, this mesh size increases from 2 to 40 parts for obtaining higher accuracies.

Table 7.2. Mesh sizes of plate bending elements

Division of Both Sides	Total Number of Elements	
	TF3, TTF6	QF4, QTF8
2	8	4
6	72	36
10	200	100
40	3200	1600

7.4. Discussion of Results

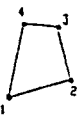
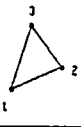
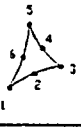
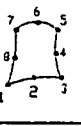
In this section, the accuracy of the higher order elements for a square plate bending problem is tested. The solutions for deflection and moments are mentioned in Table 7.3 to 7.5.

For all cases, the quadratic higher order triangular plate element, TTF6, gives the most inaccurate values in coarse mesh. Also, the vertical deflection for the linear triangular element, TF3, is far from the exact value. By refining the mesh size, the accuracy of the solution becomes perfect. But the higher order elements, TTF6 and QTF8 do not acquire a fast approximation like the lower order elements do. The reason for this phenomenon is explained earlier in connection with circular plate to be the influence of the transverse shear effects. For the isoflex thin plate elements, TF3 and QF4, the transverse shear effects do not influence the solution in the same order as Timoshenko's plate equations [32]. On the contrary, in the isoparametric thick Midlin plate elements, TTF6 and QTF8, the transverse shears affect the solution. The comparison of the results is shown in Figure 7.3 to Figure 7.5.

For the bending moment at the center of the plate, compared in Figure 7.5, all elements have a lower bound approach and yield accurate solutions for fine mesh sizes except the regular order element, TF3.

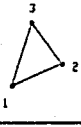
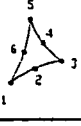
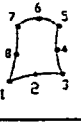
In addition, the mesh sizes, input and selected output data of the problem are included in Appendix F.

Table 7.3. Comparison of vertical deflection w , at the center of the plate (m)

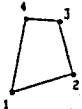
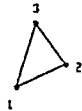
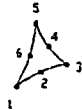
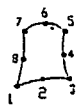
Name	Geometry	Lusas 13.4				Exact Solution
		2 / 2 *	6 / 6	10 / 10	40 / 40	
QF4		-0,001327	-0,001176	-0,001186	-0,001191	-0,0011886
TF3		-0,000559	-0,001133	-0,001169	-0,001189	-0,0011886
TTF6		-0,002980	-0,001183	-0,001241	-0,001265	-0,0011886
QTF8		-0,001359	-0,001231	-0,001245	-0,001264	-0,0011886

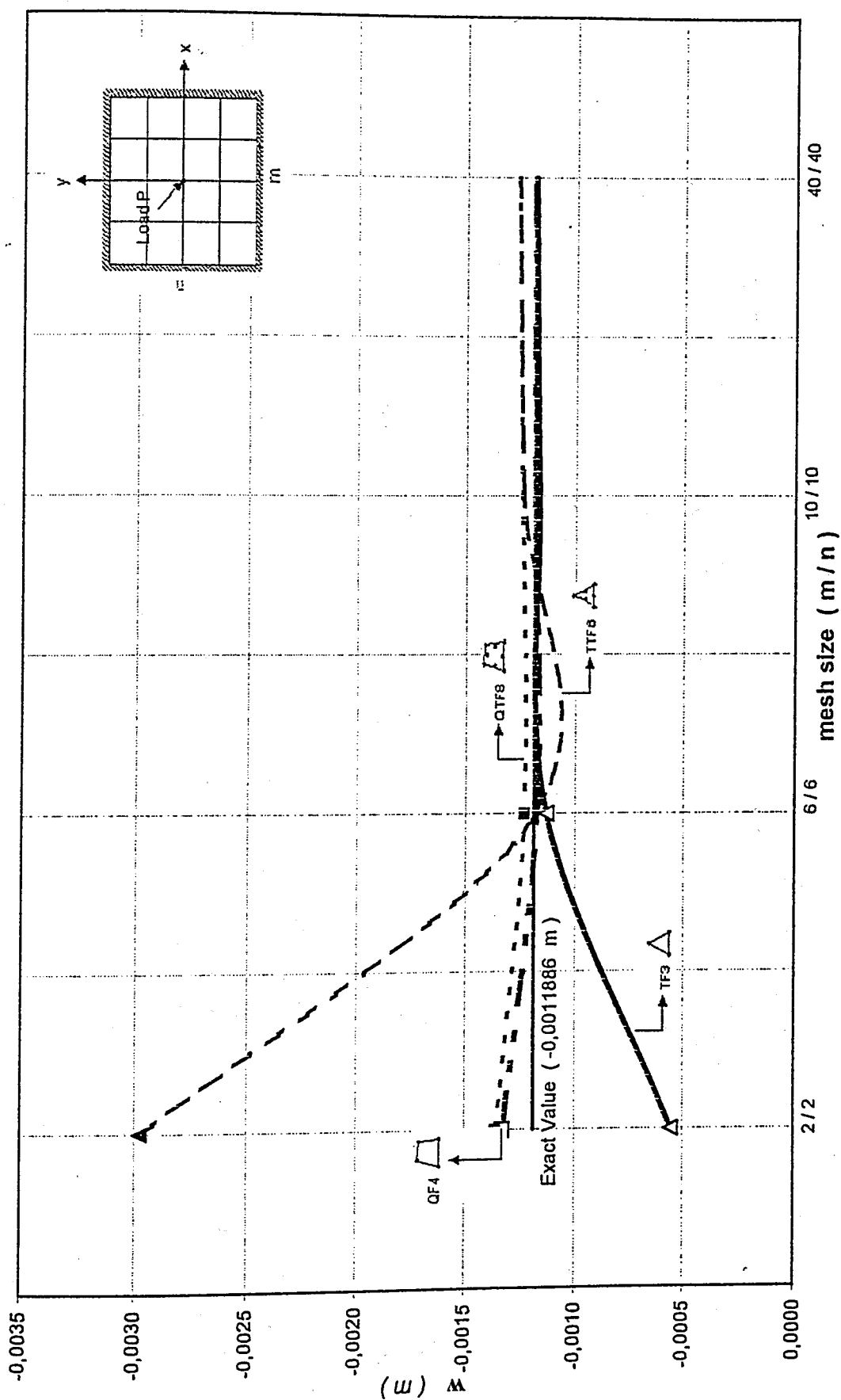
* Division of both sides into 2 parts

Table 7.4. Comparison of moment M_y , at the boundary of the plate (kNm)

Name	Geometry	LUSAS 13.4				Exact Solution
		2 / 2	6 / 6	10 / 10	40 / 40	
QF4		15,00	12,67	12,58	12,57	12,57
TF3		6,82	11,33	11,78	12,34	12,57
TTF6		2,21	10,56	11,33	12,39	12,57
QTF8		15,00	12,46	12,43	12,44	12,57

Tanle 7.5. Comparison of moment (M_x, M_y) at the center of the plate (kNm)

LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 2	6 / 6	10 / 10	40 / 40	
QF4		-19,50	-28,68	-33,94	-48,29	-49,91
TF3		-9,23	-22,81	-28,07	-42,45	-49,91
TTF6		-2,16	-23,29	-31,11	-49,09	-49,91
QTF8		-19,50	-29,63	-34,32	-48,66	-49,91

Figure 7.3. Vertical deflection w , at the center of the plate

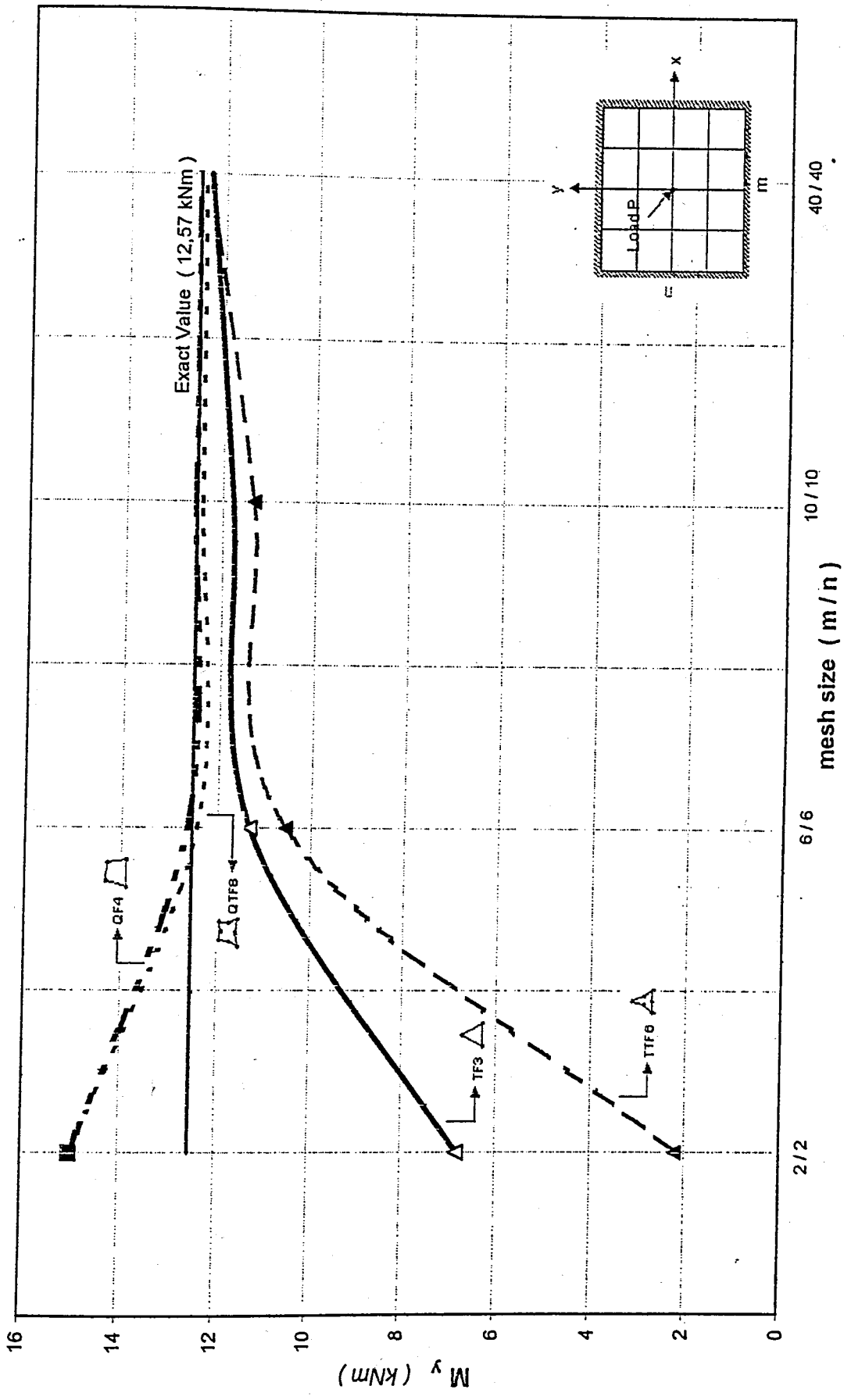


Figure 7.4. Moment (M_y) at the boundary of the plate

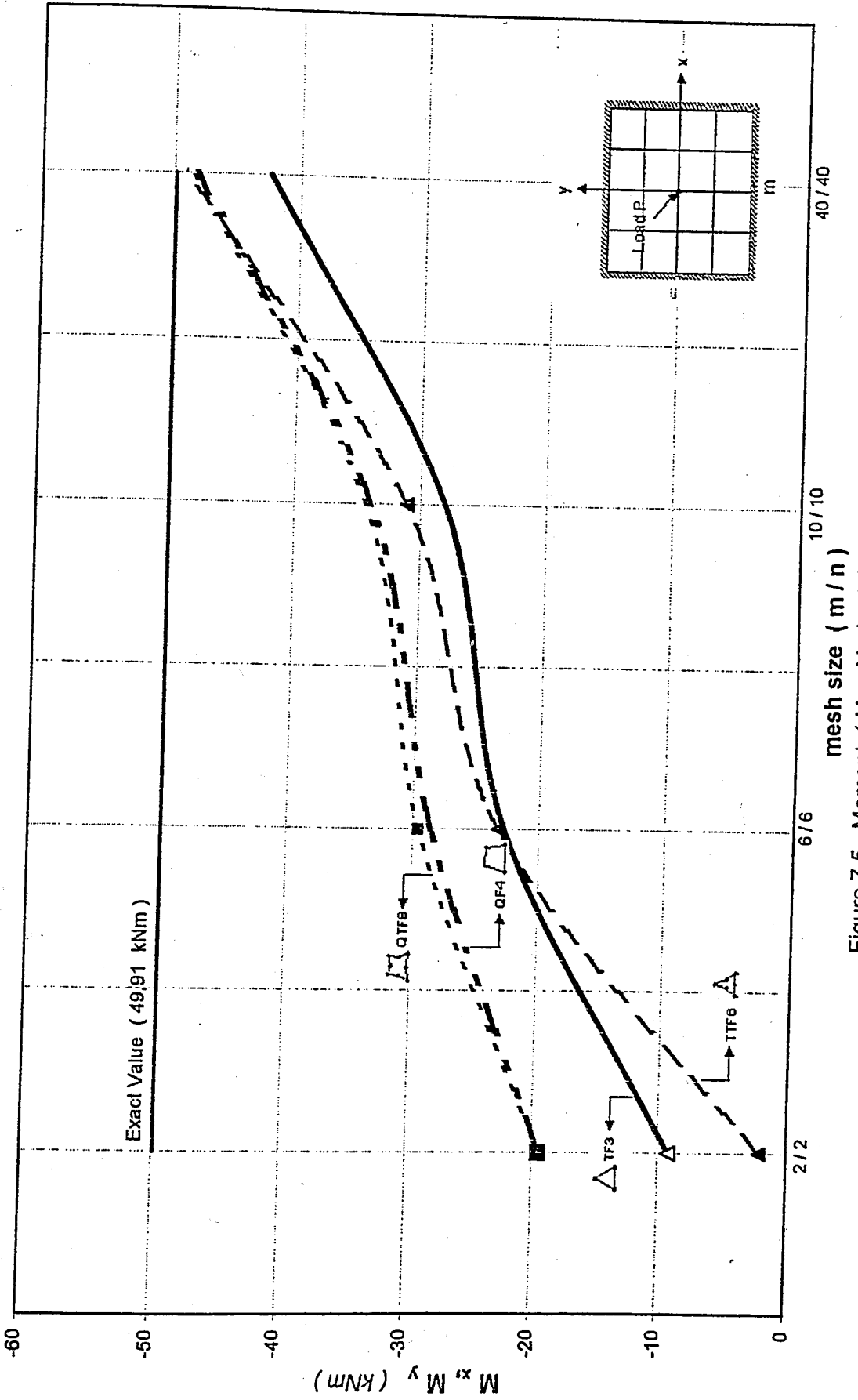


Figure 7.5. Moment (M_x, M_y) at the center of the plate

8. CASE STUDY NO. 4

(Rectangular Plate Bending Problem under Uniform Load)

8.1. Data for the Square Plate Bending Problem

The square plate bending problem with clamped edges under uniform load, q , has the same properties with the problem described earlier in Section 7. The shape of the square plate is presented in Figure 8.1. Once more, the quarter of the plate is taken into consideration due to the symmetry of the properties. The width and the height of the plate, uniform load, thickness, Young's modulus, and Poisson's ratio of the problem are a , b , q , t , E and ν , respectively. The properties of the plate are

Dimensions of the Plate: $4 \times 4 \times 0.14$ m

Uniform Load: $q = 20 \text{ kN/m}^2$

Young's modulus: $E = 30.10^6 \text{ kN/m}^2$

Poisson's ratio: 0.3

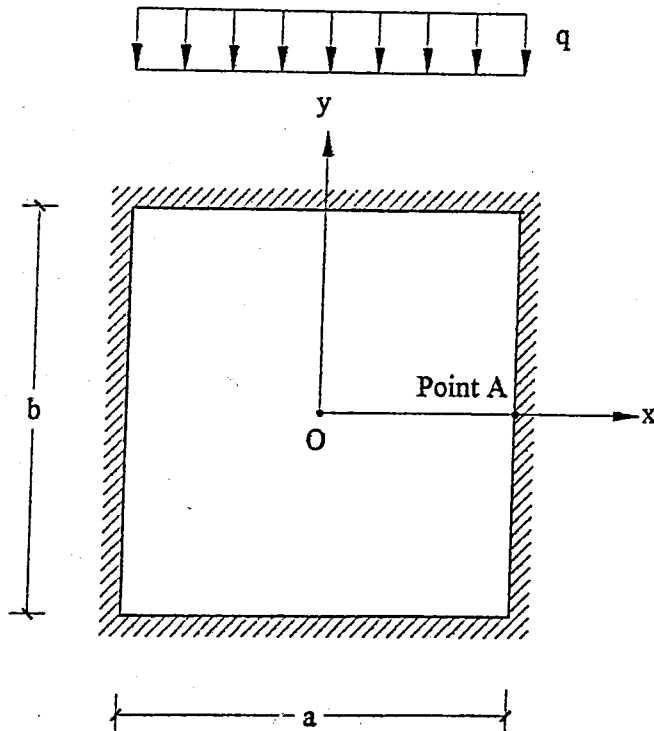


Figure 8.1. Square plate under uniform load

8.2. The Exact Solutions

The deflections and the moments are symmetrical with respect to the coordinate axes as shown in Figure 8.1. The vertical deflection and moments for each direction at the center of the plate and the moment for x direction at point A are calculated in this section. The central vertical deflection is equal to [32]

$$(w)_{x=y=0} = 0.00126 \frac{qa^4}{D} \quad (8.1)$$

where the flexural rigidity of the plate is expressed in equation (7.2), and a is the side of the plate in x-direction. The bending moment in x direction at point A is defined by

$$(M_x)_{x=a/2, y=0} = -0.0513 qa^2 \quad (8.2)$$

and the central bending moment is defined by

$$(M_x = M_y)_{x=0, y=0} = 0.0231 qa^2 \quad (8.3)$$

The exact values of the vertical deflection and bending moments are summarized in Table 8.1.

Table 8.1. Vertical deflection and bending moments

AT THE CENTER		AT POINT A
w (m)	M_x, M_y (kNm)	M_y (kNm)
-0,0008558	-7,392	16,416

8.3. Finite Element Types used in the Analyses

The problem as illustrated in Figure 8.1 is solved by the LUSAS package program considering only one quarter of a square plate with clamped edges under uniform load. The elements chosen for the meshing are the same as used for the same plate bending problems in Section 7. These are the isoflex thin plate elements, 4-noded QF4 and 3-noded TF3; the isoparametric thick Midlin plate elements, 6-noded TTF6 and 8-noded QTF8. The general information about these plate elements is presented in Section 7.3 and Appendix C and D.

Mesh sizes are in four categories which are obtained by dividing both sides of the plate by 2, 8, 30 and 100 parts. The total number of elements for quadrilateral and triangular elements increase from 4, 8 to 10000, and 20000 at the end, respectively. The total number of elements used are shown in Table 8.2.

Table 8.2. Mesh sizes of plate bending elements

Division of Both Sides	Total Number of Elements	
	TF3, TTF6	QF4, QTF8
2	8	4
8	128	64
30	1 800	900
100	20 000	10 000

8.4. Discussion of Results

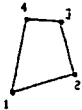
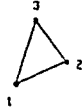
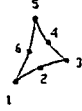
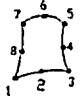
It has been shown for a square plate bending problem under uniform load that all chosen isoparametric elements gives appropriate exact solution by refining mesh sizes. The central vertical deflection and the moments at the center of the plate and at point A are solved and shown in Table 8.3 to Table 8.5. The results are compared in Figure 8.2 to Figure 8.4. Also, the mesh types are illustrated in Appendix G.

As it is seen from the tables and figures, the higher order elements, TTF6 and QTF8, do not approach the exact value in the same speed as the regular order elements, TF3 and QF4. It is mentioned earlier that, the influence of the transverse shear is neglected while solving the exact equations [32] by TF3 and QF4. In contrast, the transverse shears affect the solutions when using higher order elements, TTF6 and QTF8.

In Table 8.3 and Figure 8.2, the comparison of the central vertical deflection for different mesh sizes and elements is shown. The general tendency is that all elements give acceptable results for fine mesh size. The elements TTF6 and TF3 have lower bound approach up to eight elements. After the mesh sizes, they approach upper bound to the exact solution.

For the bending moment at the center of the plate, compared in Figure 8.4, a higher accuracy and better convergence are obtained similar to those for the central concentrated load. The numerical results are very accurate compared to the analytical solutions from the Timoshenko and Woinowsky-Krieger [32] plates for the fine mesh. The relative performance of each element as the mesh sizes become finer and finer, are illustrated in Figures 8.2 to Figure 8.4 and Table 8.3 to Table 8.5.

Table 8.3. Comparison of vertical deflection w , at the center (m)

LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 2 *	8 / 8	30 / 30	100 / 100	
QF4		-0,001061	-0,000857	-0,000859	-0,000859	-0,0008558
TF3		-0,000714	-0,000879	-0,000861	-0,000860	-0,0008558
TTF6		-0,000159	-0,000857	-0,000880	-0,000880	-0,0008558
QTF8		-0,001077	-0,000880	-0,000880	-0,000880	-0,0008558

* Division of sides into 2 parts

Table 8.4. Comparison of moment (M_x , M_y) at the boundary (kNm)

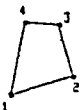
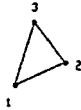
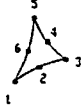
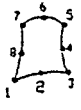
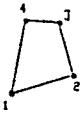
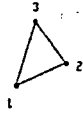
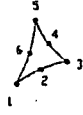
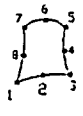
LUSAS 13. 4						Exact Solution
Name	Geometry	2 / 2	8 / 8	30 / 30	100 / 100	
QF4		12,000	16,024	16,391	16,423	16,416
TF3		12,012	16,298	16,423	16,433	16,416
TTF6		1,180	13,154	16,122	16,330	16,416
QTF8		12	15,930	16,311	16,337	16,416

Table 8.5. Comparison of moment (M_x , M_y) at the center (kNm)

Name	Geometry	LUSAS 13. 4				Exact Solution
		2 / 2	8 / 8	30 / 30	100 / 100	
QF4		-15,60	-7,58	-7,35	-7,33	-7,39
TF3		-9,84	-7,97	-7,38	-7,33	-7,39
TTF6		-1,15	-7,28	-7,36	-7,35	-7,39
QTF8		-15,60	-7,63	-7,36	-7,35	-7,39

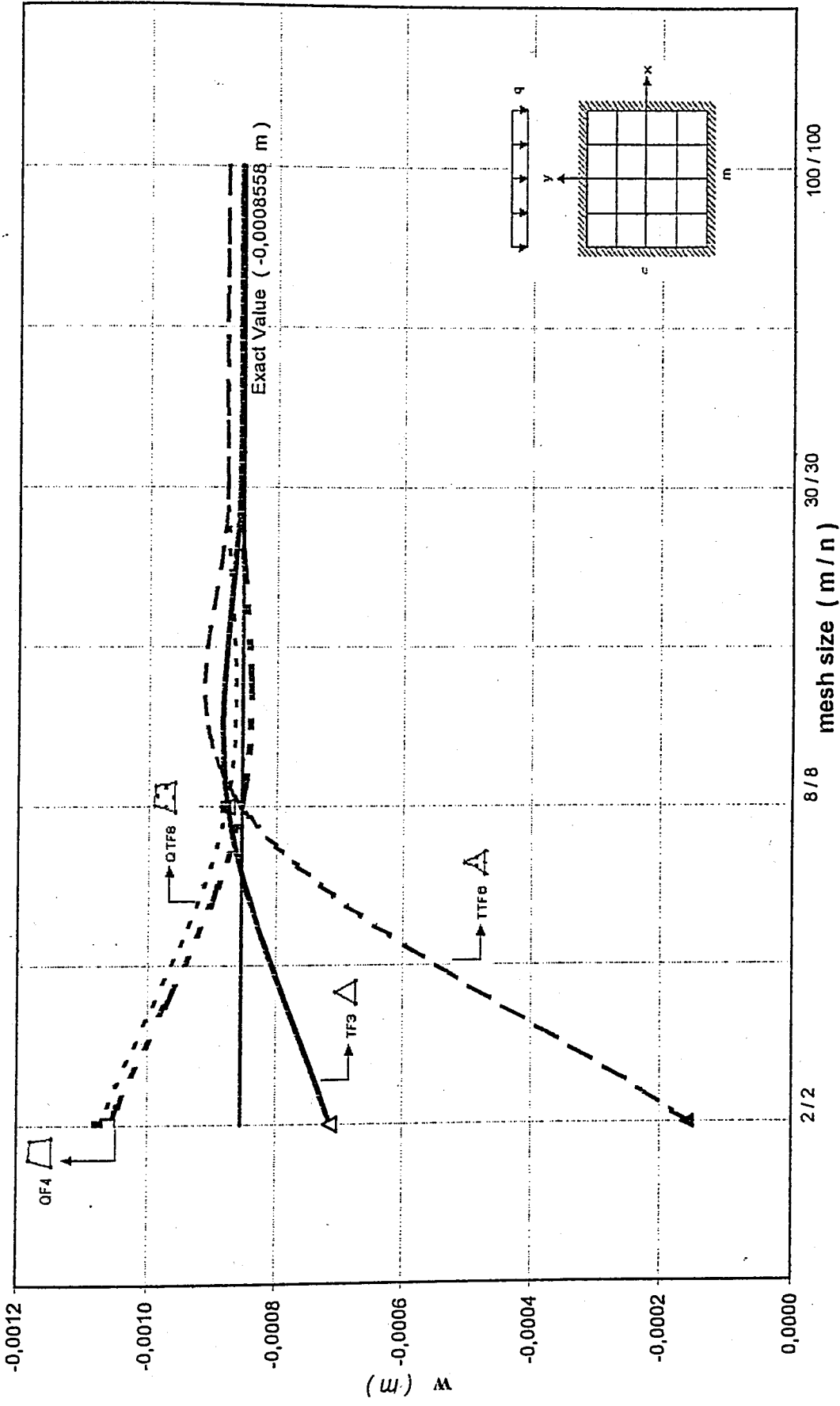


Figure 8.2. Vertical deflection w , at the center of the plate

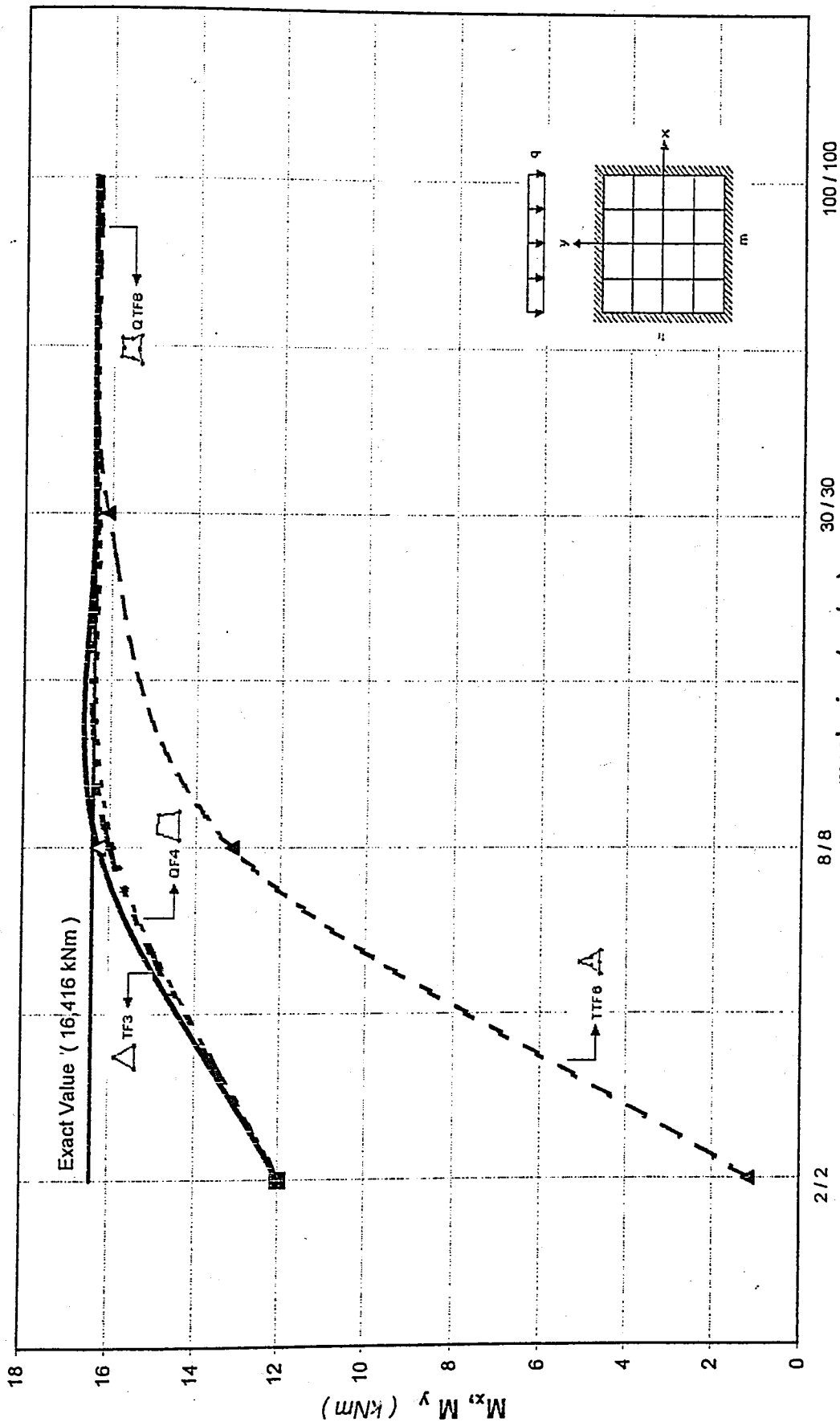


Figure 8.3. Moment (M_x, M_y) at the boundary of the plate

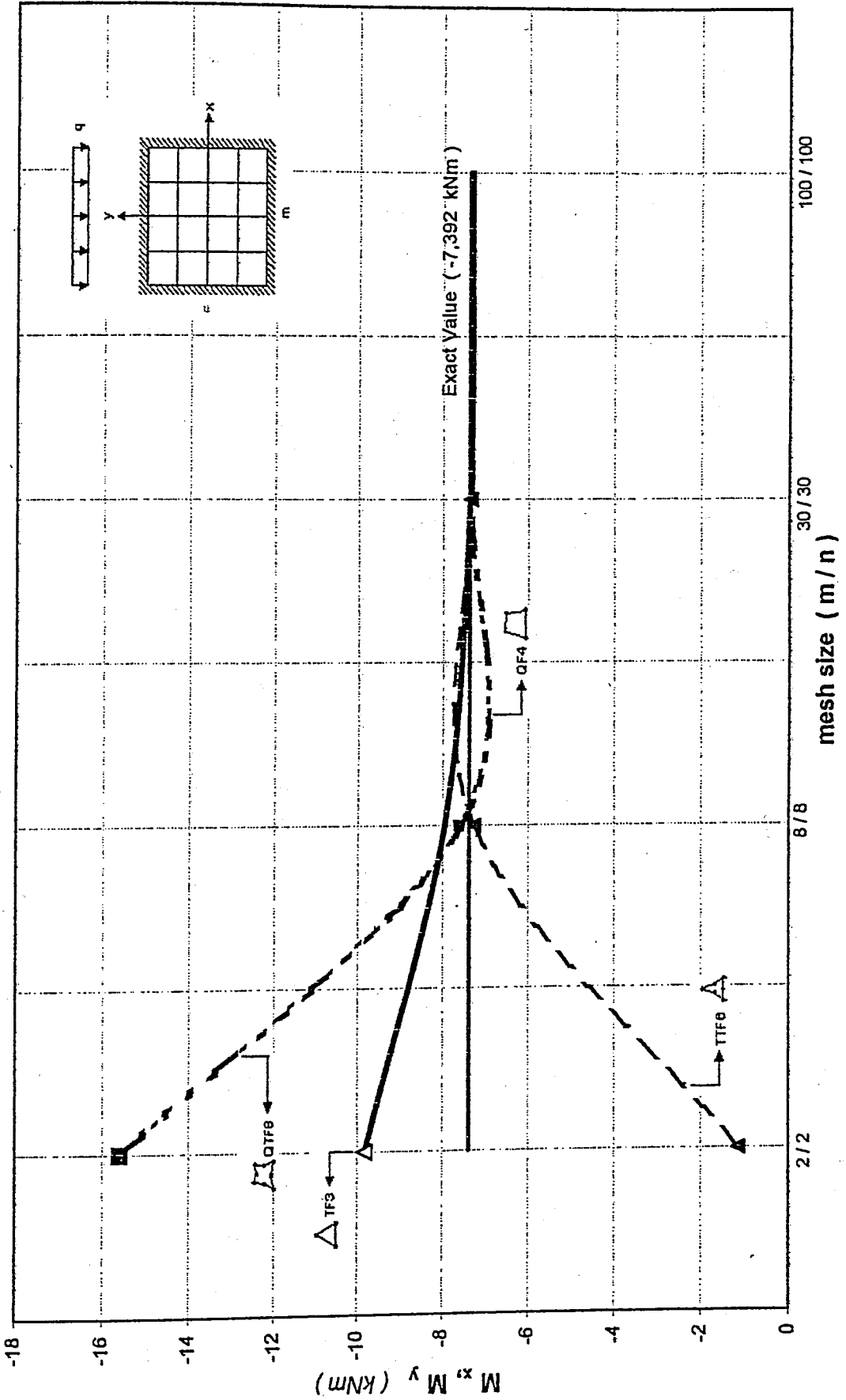


Figure 8.4. Moment (M_x , M_y) at the center of the plate

9. CONCLUSIONS

The discussions presented briefly after each case study indicate that the relatively higher accuracies are obtained when higher order finite elements are used even with a coarse mesh size.

The results and the comparative curves indicate also that the basic triangular elements are the most incapable and inaccurate elements for the solution of the plane stress problem unless very fine mesh sizes are used. The results of the regular order quadrilateral elements with enhanced strains however approach the exact value faster than regular order quadrilateral elements. These four-node elements may be used satisfactorily by increasing the number of elements. Higher order triangular elements may be preferred for a better performance instead of regular quadrilateral elements with a least element number. However, if the mesh size is selected to be extremely coarse, then the higher order quadrilateral elements produce the most accurate results.

For plate bending problems, the higher order triangular and quadrilateral elements produce slightly different results than those of the regular order elements and exact solutions based on thin plate theory. The reason for this discrepancy is that the higher order plate bending elements are used to analyze the plate bending problems by including the transverse shear effects into the solution. On the other hand, these transverse shear effects are negligible for the thin plate bending problems. As a result, the efficiency and accuracy of higher order finite elements are confirmed even for the plate bending problems.

The extension of the types of the problem to be discussed is quite possible. For instance, a cylindrical concrete shell roof, a three-dimensional solid problem or an axisymmetric membrane problem may be analyzed for the demonstration of the performance of higher order elements. The results will be the same: The higher order finite elements provide relatively more accurate solutions than the regular order elements, for the same mesh size.

APPENDIX A : ENHANCED STRAIN ELEMENTS

The lower order enhanced strain elements exhibit improved accuracy in coarse meshes when compared with their parent elements QPM4, QPN4 and QAX4, particularly if bending predominates. In addition, these elements do not suffer from 'locking' in the nearly incompressible limit. The elements are based on a three-field mixed formulation in which stresses, strains and displacements are represented by three independent functions in three separate vector spaces. The formulation is based on the inclusion of an assumed 'enhanced' strain field which is related to internal degrees of freedom. These internal degrees of freedom are eliminated at the element level before assembly of the stiffness matrix for the structure. The formulation provides for the following three conditions to be satisfied [23]

- (i) Independence of the enhanced and standard strain interpolation functions.
- (ii) L_2 orthogonality of the stress and enhanced strains.
- (iii) Capability of the element to model a constant state of stress after enforcing the orthogonality condition, i.e. requirement for passing the patch test.

In addition to ensuring that the element passes the patch test, these conditions also allow the stress field to be eliminated from the formulation.

A.1. Formulation

The formulation requires that the total strain is expressed as the sum of a 'compatible' strain and an 'enhanced' strain

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}_c + \boldsymbol{\varepsilon}_e \quad (\text{A.1})$$

The compatible strain is directly related to the displacements of the element nodes in the standard manner. The enhanced strain is related to internal degrees of freedom which are eliminated using static condensation at the element level. The enhanced strains are

therefore discontinuous between elements. The weak form of the three field variational equations for equilibrium, compatibility and constitutive relationship may be expressed as

$$\begin{aligned} \int_{\Omega} \delta \varepsilon_c^T \sigma d\Omega - R^T \delta a &= 0 \\ \int_{\Omega} \delta \sigma^T \varepsilon_e d\Omega &= 0 \\ \int_{\Omega} \delta \varepsilon^T \left[-\sigma + \frac{\partial W}{\partial \varepsilon} \right] d\Omega &= 0 \end{aligned} \quad (A.2)$$

where R is the applied loading, W is the strain energy density, a are nodal displacements and Q is the stress vector.

By enforcing the so-called L2 orthogonality condition between stress and enhanced strain, terms involving $\sigma^T \varepsilon_e$ will disappear. This allows the stress field to be eliminated from the formulation.

The compatible and enhanced strains are computed from

$$\begin{aligned} \varepsilon_c &= B a \\ \varepsilon_e &= G \alpha_e \end{aligned} \quad (A.3)$$

where G operates on the assumed strain parameters α_e to provide the enhanced strains.

Substitution of these expressions into the two remaining field variational equations yields

$$\int_{\Omega} \delta \varepsilon_c^T D \left\{ B da + G d\alpha_e \right\} - R^T \delta a d\Omega + \int_{\Omega} \delta \varepsilon_e^T D \left\{ B da + G d\alpha_e \right\} d\Omega = 0 \quad (A.4)$$

The following matrices are defined for use in discretising this equation

$$\begin{aligned}
 \underset{\approx}{K}^{(k)} &= \int_{\underset{\approx}{\Omega}} \underset{\approx}{B}^T \underset{\approx}{D} \underset{\approx}{B} d\Omega && (n_{el} * n_{el} \text{ matrix}) \\
 \underset{\approx}{H}^{(k)} &= \int_{\underset{\approx}{\Omega}} \underset{\approx}{G}^T \underset{\approx}{D} \underset{\approx}{G} d\Omega && (m_{el} * m_{el} \text{ matrix}) \\
 \underset{\approx}{\Gamma}^{(k)} &= \int_{\underset{\approx}{\Omega}} \underset{\approx}{G}^T \underset{\approx}{D} \underset{\approx}{B} d\Omega && (m_{el} * n_{el} \text{ matrix})
 \end{aligned} \tag{A.5}$$

n_{el} is the dimension of the element displacement field, m_{el} is the number of element enhanced strain modes. D is the modulus matrix at loadstep k .

The internal force vectors are given by

$$\begin{aligned}
 \underset{\approx}{P}^{(k)} &= \int_{\underset{\approx}{\Omega}} \underset{\approx}{B}^{(k)T} \underset{\approx}{\sigma}^{(k)} d\Omega \\
 \underset{\approx}{h}^{(k)} &= \int_{\underset{\approx}{\Omega}} \underset{\approx}{G}^{(k)T} \underset{\approx}{\sigma}^{(k)} d\Omega
 \end{aligned} \tag{A.5}$$

where $\underset{\approx}{h}^{(k)}$ is the internal force vector relating to the incompatible modes which is subsequently eliminated at the element level.

Using standard finite element techniques for assembling the system of equations gives

$$\begin{bmatrix} \underset{\approx}{K}^{(k)} & \underset{\approx}{\Gamma}^{(k)T} \\ \underset{\approx}{\Gamma}^{(k)} & \underset{\approx}{H}^{(k)} \end{bmatrix} \begin{Bmatrix} \Delta \underset{\approx}{a}^{(k+1)} \\ \Delta \underset{\approx}{\alpha}^{(k+1)} \end{Bmatrix} = \begin{Bmatrix} \underset{\approx}{R} - \underset{\approx}{P}^{(k)} \\ \underset{\approx}{O} - \underset{\approx}{h}^{(k)} \end{Bmatrix} \tag{A.6}$$

This nonlinear system of the equations is solved using a Newton-Raphson iteration scheme. However, for the linear case, no iterations are necessary as h will be 0 and P will not be considered.

Static condensation of this system of equations eliminates the equations included to enforce the orthogonality condition. The element stiffness and internal forces used to assemble the equations for the structure then become

$$\tilde{\mathbf{K}}_{\approx}^{(k)} \Delta \mathbf{a}^{(k+1)} = \Psi^{(k)} \quad (\text{A.7})$$

where

$$\begin{aligned} \tilde{\mathbf{K}}_{\approx}^{(k)} &= \mathbf{K}_{\approx}^{(k)} - \Gamma_{\approx}^{(k)T} \left[\mathbf{H}_{\approx}^{(k)} \right]^{-1} \Gamma_{\approx}^{(k)} \\ \Psi^{(k)} &= \mathbf{R} - \tilde{\mathbf{P}}^{(k)} \end{aligned} \quad (\text{A.8})$$

$$\tilde{\mathbf{p}}^{(k)} = \mathbf{P}^{(k)} - \Gamma_{\approx}^{(k)T} \left[\mathbf{H}_{\approx}^{(k)} \right]^{-1} \mathbf{h}_{\approx}^{(k)}$$

In nonlinear analyses, the enhanced strain parameters are updated as

$$\alpha^{(k+1)} = \alpha^{(k)} - \left[\mathbf{H}_{\approx}^{(k)} \right]^{-1} \left[\Gamma_{\approx}^{(k)} \Delta \mathbf{a}^{(k)} + \mathbf{h}_{\approx}^{(k)} \right] \quad (\text{A.9})$$

The actual implementation of this formulation requires the orthogonality condition to be related to the isoparametric space. Transformations are therefore required to assemble matrices and vector that relate to covariant strains and contravariant stresses. Standard transformations are applied and full details of this procedure are given. It is postulated that the covariant enhanced strain field is given by

$$\varepsilon_e = \mathbf{E} \alpha_e \quad (\text{A.10})$$

where $\tilde{E}$ is the equivalent of $\tilde{G}$ in the isoparametric space.

A.2. Enhanced Strain Interpolation

The incompatible displacement field is given by

$$\tilde{u} = \tilde{N}_1(\xi)\lambda_1 + \tilde{N}_2(\eta)\lambda_2 \quad (A.10)$$

where

$$\tilde{N}_1(\xi) = \frac{1}{2}(1-\xi^2), \quad \tilde{N}_2(\eta) = \frac{1}{2}(1-\eta^2) \quad (A.11)$$

and the λ_i represent the incompatible modes

$$\lambda_1 = \{\tilde{u}_1, \tilde{v}_1\}^T, \quad \lambda_2 = \{\tilde{u}_2, \tilde{v}_2\}^T \quad (A.12)$$

The covariant base vectors associated with the isoparametric space are

$$g_\xi = \begin{Bmatrix} x^T N_{,\xi} \\ y^T N_{,\xi} \end{Bmatrix} = \begin{Bmatrix} x^T a_1 \\ y^T a_1 \end{Bmatrix} + \eta \begin{Bmatrix} x^T h \\ y^T h \end{Bmatrix} = g_1^o + \eta g \quad (A.13)$$

$$g_\eta = \begin{Bmatrix} x^T N_{,\eta} \\ y^T N_{,\eta} \end{Bmatrix} = \begin{Bmatrix} x^T a_2 \\ y^T a_2 \end{Bmatrix} + \xi \begin{Bmatrix} x^T h \\ y^T h \end{Bmatrix} = g_1^o + \xi g$$

where

$$N = a_0 + a_1\xi + a_2\eta + h\xi\eta \quad (A.14)$$

and

$$a_0 = \frac{1}{4} [1 \quad 1 \quad 1 \quad 1]^T$$

$$a_1 = \frac{1}{4} [-1 \quad 1 \quad 1 \quad -1]^T$$

$$a_2 = \frac{1}{4} [-1 \quad -1 \quad 1 \quad 1]^T$$

$$h = \frac{1}{4} [1 \quad -1 \quad 1 \quad -1]^T$$

$$x = \frac{1}{4} [x_1 \quad x_2 \quad x_3 \quad x_4]^T$$

$$y = \frac{1}{4} [y_1 \quad y_2 \quad y_3 \quad y_4]^T$$

The initial enhanced strain field in isoparametric space is then given by

$$\varepsilon_i = \begin{Bmatrix} (\tilde{u}_{,\xi})^T g_\xi \\ (\tilde{u}_{,\eta})^T g_\eta \\ (\tilde{u}_{,\xi})^T g_\eta + (\tilde{u}_{,\eta})^T g_\xi \end{Bmatrix} = \begin{bmatrix} \xi & 0 & 0 & 0 & \xi\eta & 0 \\ 0 & \eta & 0 & 0 & 0 & \xi\eta \\ 0 & 0 & \xi & \eta & \xi^2 & \eta^2 \end{bmatrix} \begin{bmatrix} -\lambda_1^T g_1^0 \\ -\lambda_2^T g_2^0 \\ -\lambda_1^T g_2^0 \\ -\lambda_2^T g_1^0 \\ -\lambda_1^T g \\ -\lambda_2^T g \end{bmatrix} \approx E^i \alpha_e^i \quad (A.15)$$

The stress field for the element is derived from the linear uncoupled stress field

$$\sigma = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & \xi & \eta & & & \\ & & & 1 & \xi & \eta \\ & & & & & 1 & \xi & \eta \end{bmatrix} \beta \quad (A.16)$$

The introduction of four internal degrees of freedom allows four of the nine stress parameters (β) to vanish. The remaining terms satisfy the equilibrium equations. By basing the formulation on natural coordinates the element is less sensitive when distorted

and possesses no zero-energy deformation modes. Full details of the elimination of the four stress parameters are described for a hybrid element. The final contravariant stress field using five, β parameters are defined as

$$\Sigma = \begin{Bmatrix} \Sigma^\xi \\ \Sigma^\eta \\ \Sigma^{\xi\eta} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & \eta & 0 \\ 0 & 1 & 0 & 0 & \xi \\ 0 & 0 & 1 & 0 & 0 \end{bmatrix} \beta \quad (\text{A.17})$$

To satisfy the L_2 orthogonality condition $\langle \Sigma, \epsilon \rangle_{L_2}$

$$\int_{-1}^1 \int_{-1}^1 \epsilon^T \Sigma \, d\xi \, d\eta \approx 0 \quad (\text{A.18})$$

This condition is violated if the six initial enhanced strain parameters (α) are used. However, the condition is satisfied if

$$\alpha_5 = -\alpha_6 \quad (\text{A.19})$$

Forcing this equality, and hence L_2 orthogonality, gives the final enhanced strain interpolation matrix as

$$\underset{\approx}{E} = \begin{bmatrix} \xi & 0 & 0 & 0 & \xi\eta \\ 0 & \eta & 0 & 0 & -\xi\eta \\ 0 & 0 & \xi & \eta & \xi^2 - \eta^2 \end{bmatrix} \quad (\text{A.20})$$

This matrix is used in linear analyses but for nonlinear applications four enhanced strain parameters are used with the final column of $\underset{\approx}{E}$ deleted. The interpolation functions $\underset{\approx}{E}$ also allow condition (iii) to be satisfied. This is a requirement for passing the patch test and is implied in the sense that:

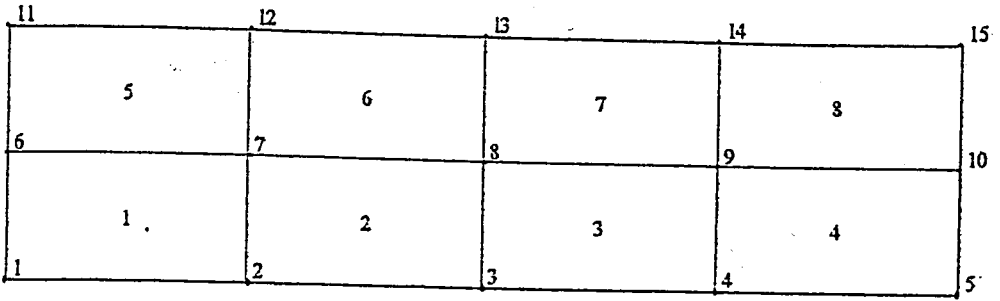
$$\int_{-1}^1 \int_{-1}^1 E \, d\xi \, d\eta = 0 \quad (\text{A.21})$$

A.3. Cases on Usage

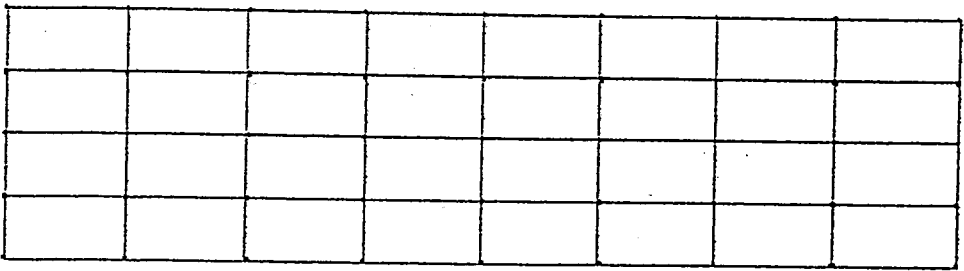
These elements exhibit an improved performance when compared with their parent elements QPM4, QPN4 and QAX4. The integration rules are applicable to the enhanced strain elements. The elements do not suffer from locking due to parasitic shear or when the material approaches the incompressible limit. The elements are also free of any spurious zero energy modes. The enhanced strain elements should be preferred to the standard elements in all cases.

APPENDIX B: MESH SIZES OF PLANE STRESS PROBLEM

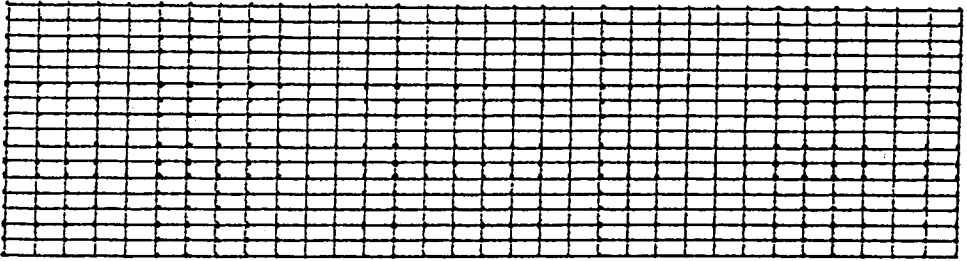
In this section, geometries of the mesh sizes of plane stress elements QPM4, QPM4M, QPM6, TPM3 and TPM6 are shown. Also, the comparison figures of the stress components σ_x , σ_y , τ_{xy} and shear components ε_x , ε_y , γ_{xy} for sections II-II' and III-III'; displacements for sections I-I' and II-II' are included.



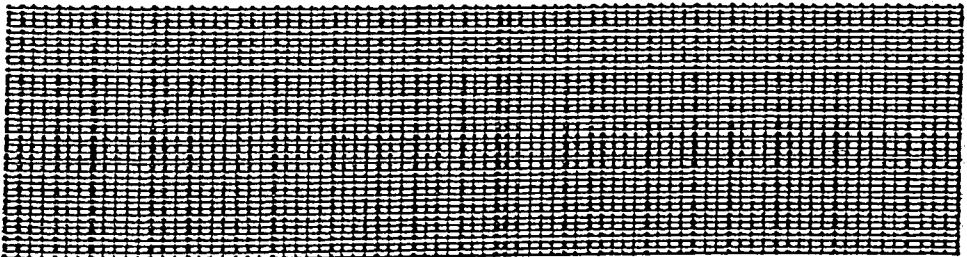
2 / 4 Mesh size



4 / 8 Mesh size

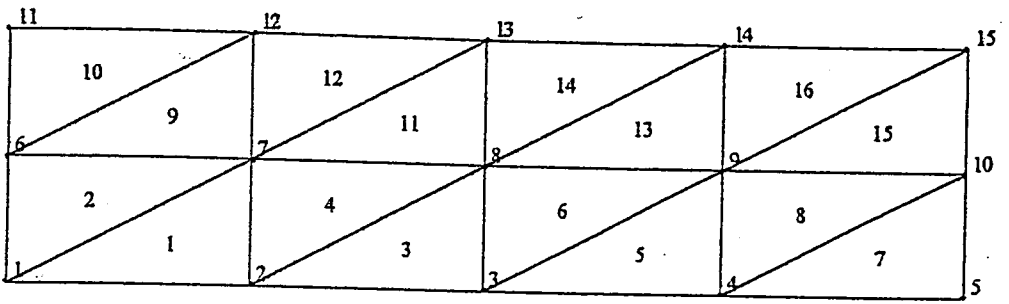


16 / 32 Mesh size

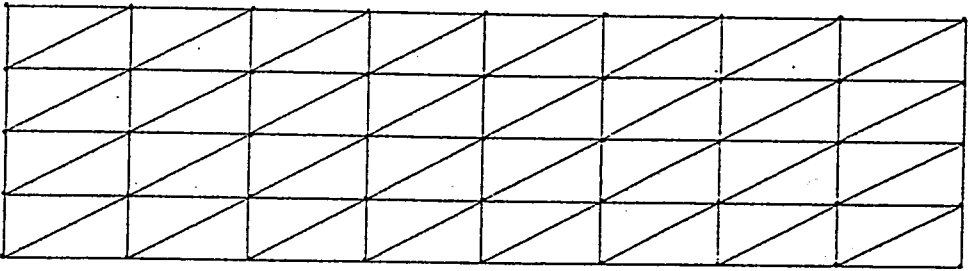


40 / 80 Mesh size

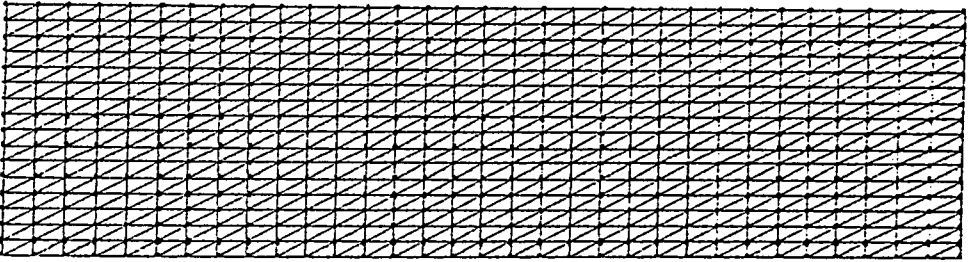
Figure B.1. Mesh sizes of plane stress elements (QPM4, QPM4M)



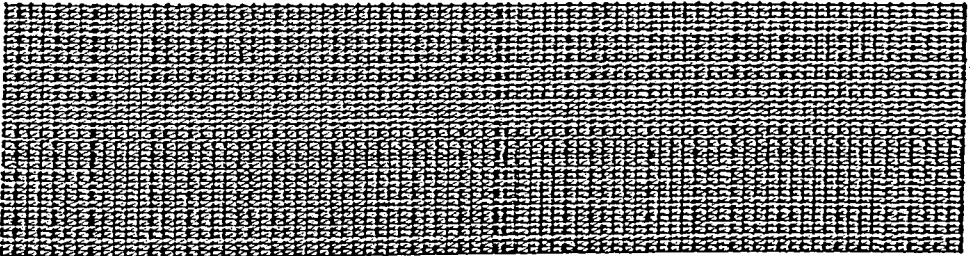
2 / 4 Mesh size



4 / 8 Mesh size

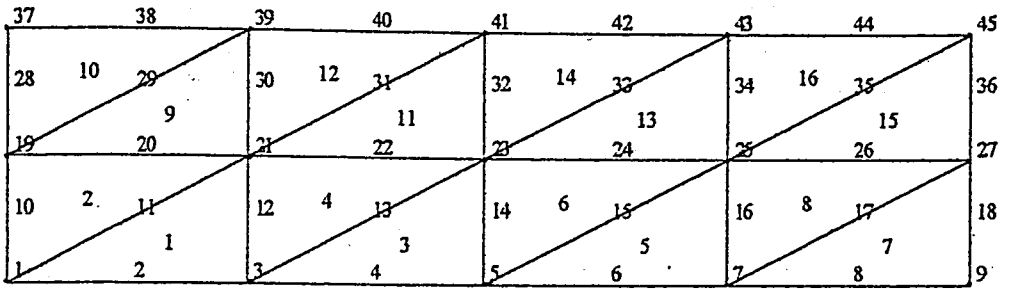


16 / 32 Mesh size

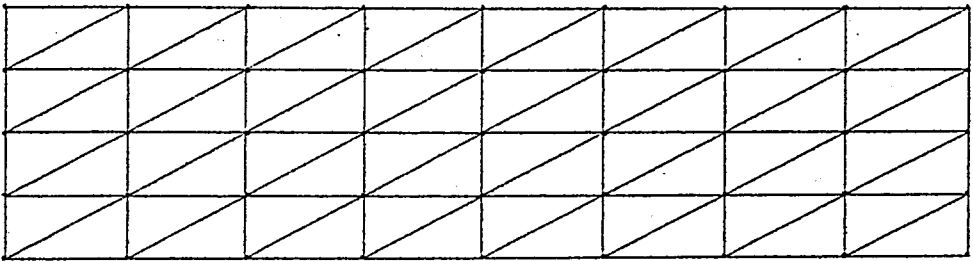


40 / 80 Mesh size

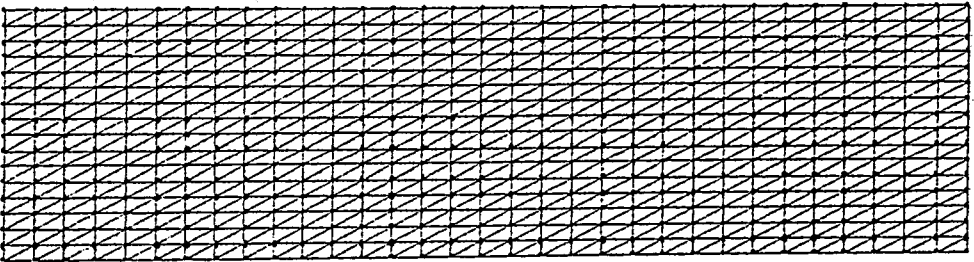
Figure B.3. Mesh sizes of plane stress elements (TPM3)



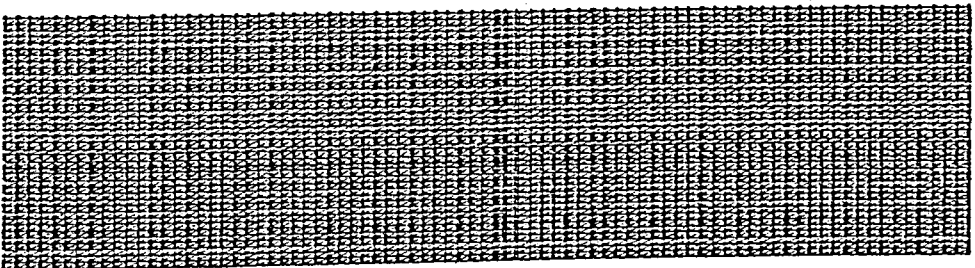
2 / 4 Mesh size



4 / 8 Mesh size



16 / 32 Mesh size



40 / 80 Mesh size

Figure B.4. Mesh sizes of plane stress elements (TPM6)

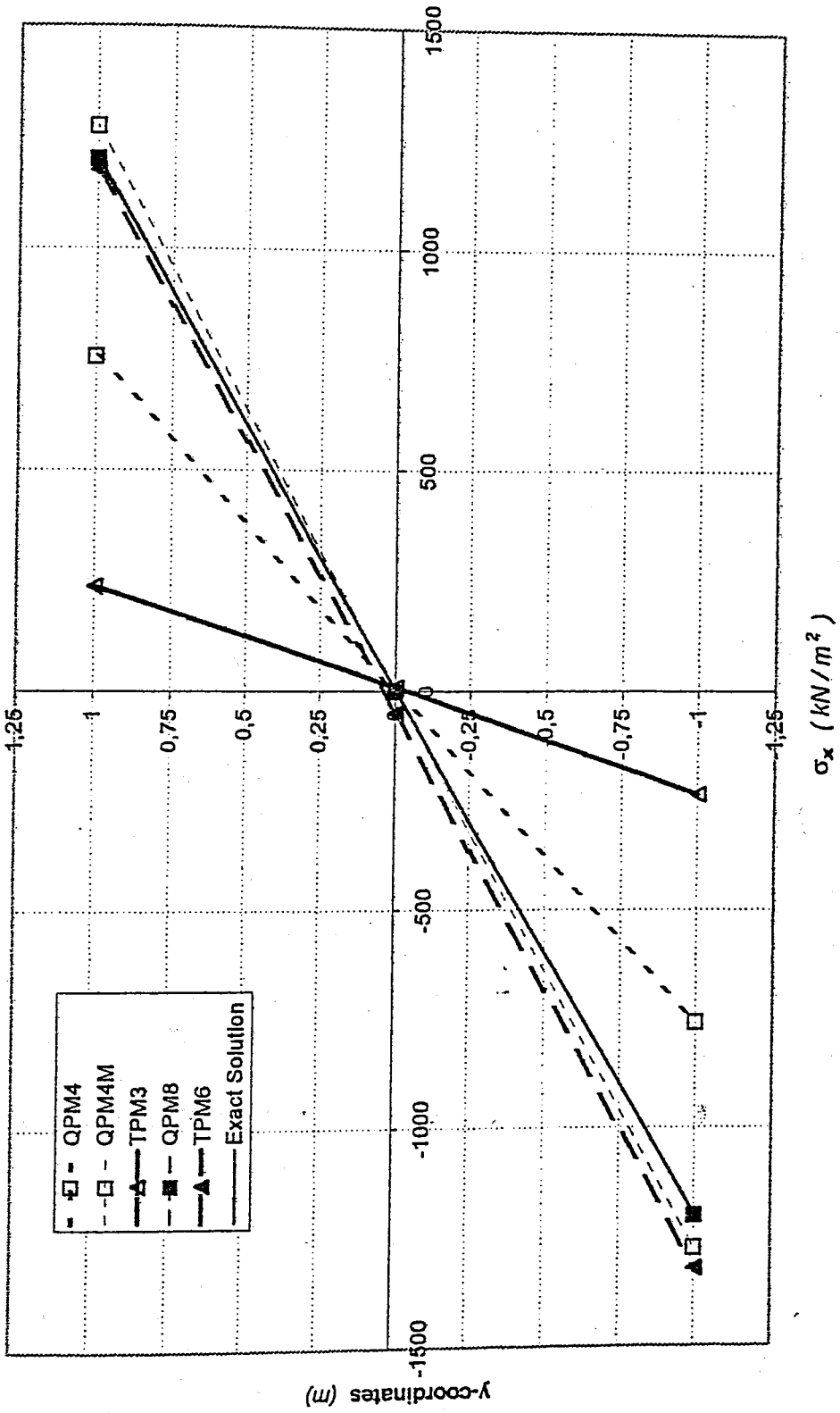


Figure B.5. Comparison of maximum normal stress (σ_x) at section II-II' (2/4 mesh size)

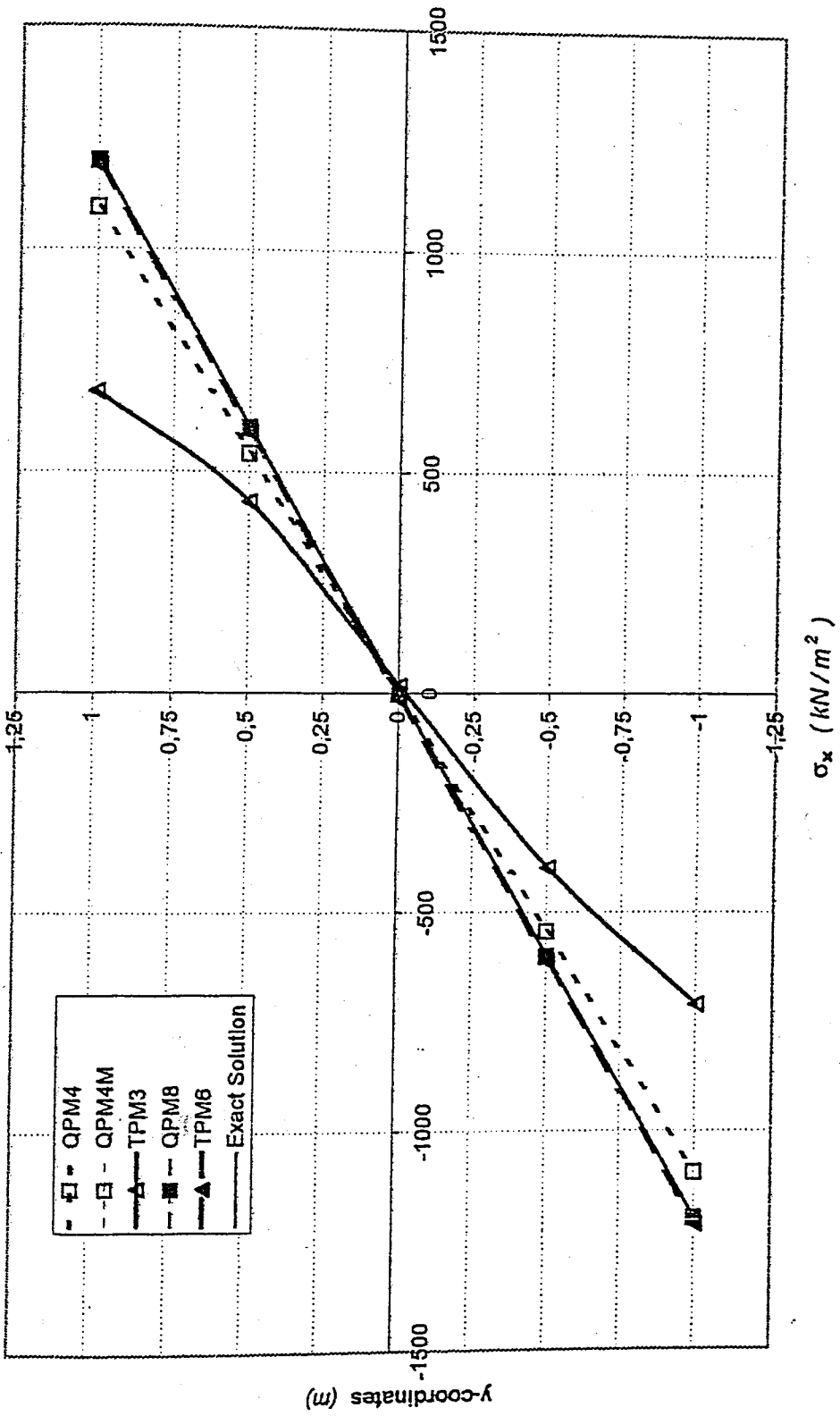


Figure B.6. Comparison of maximum normal stress (σ_x) at section II-II' (4/8 mesh size)

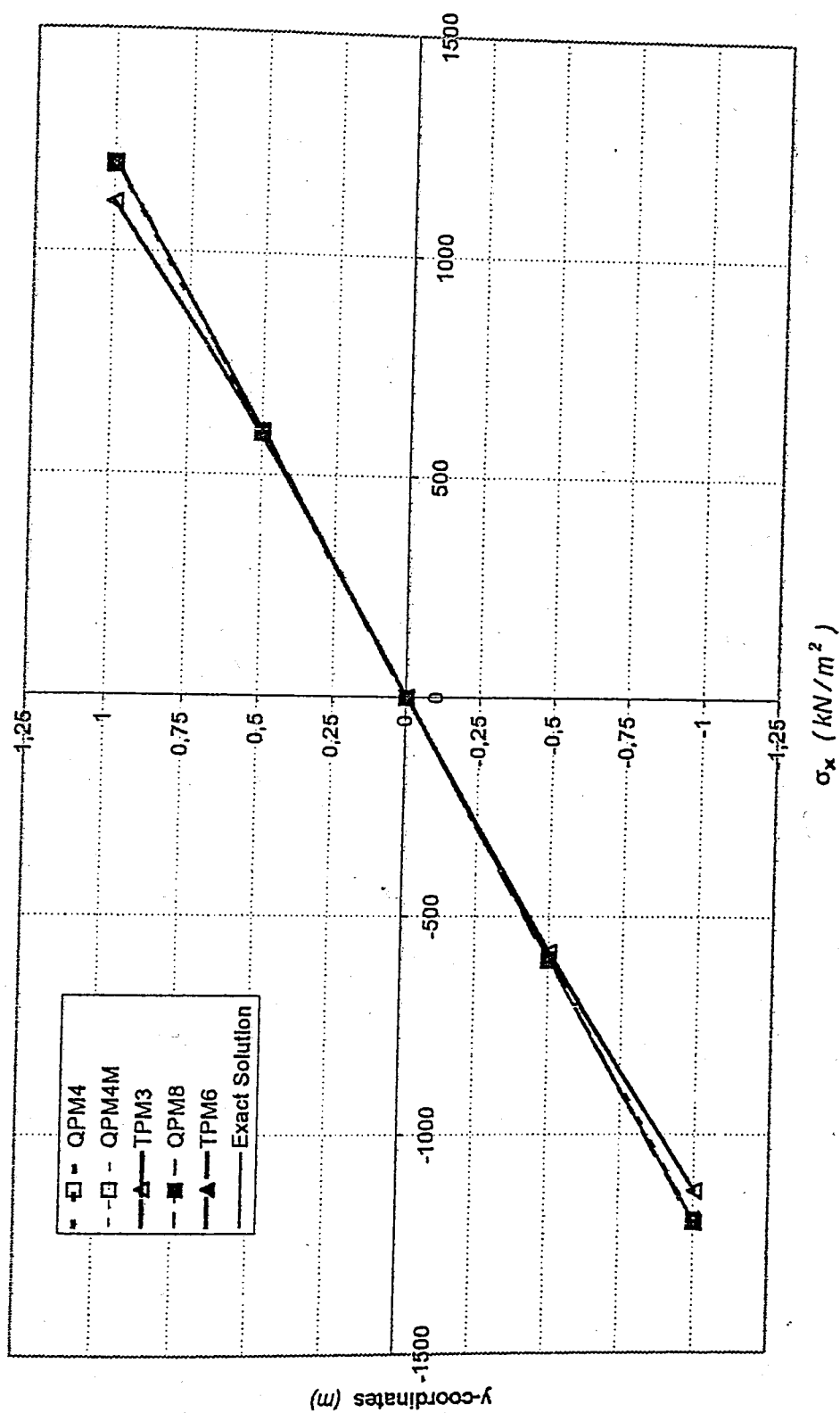


Figure B.7. Comparison of maximum normal stress (σ_x) at section II-II' (16 / 32 mesh size)

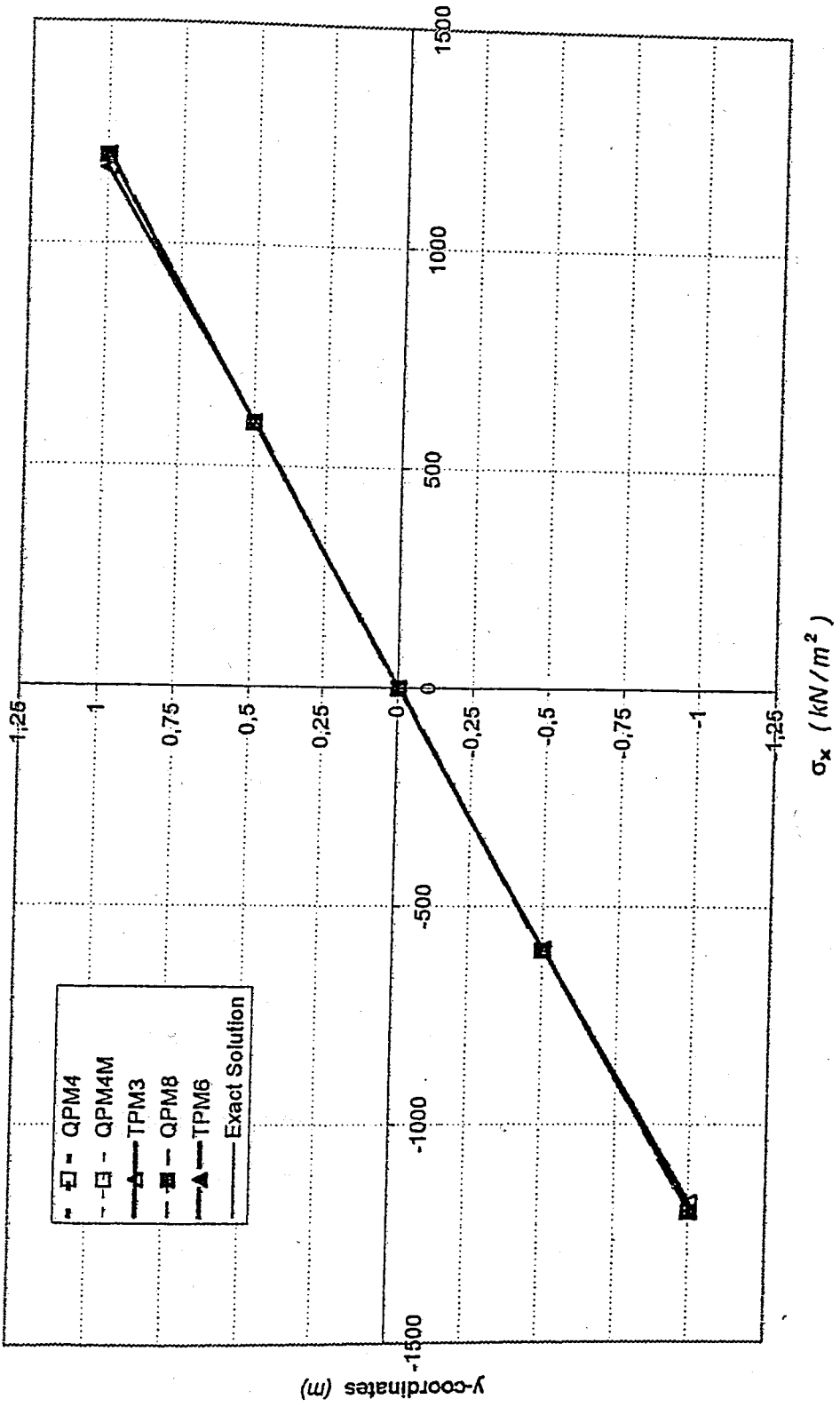
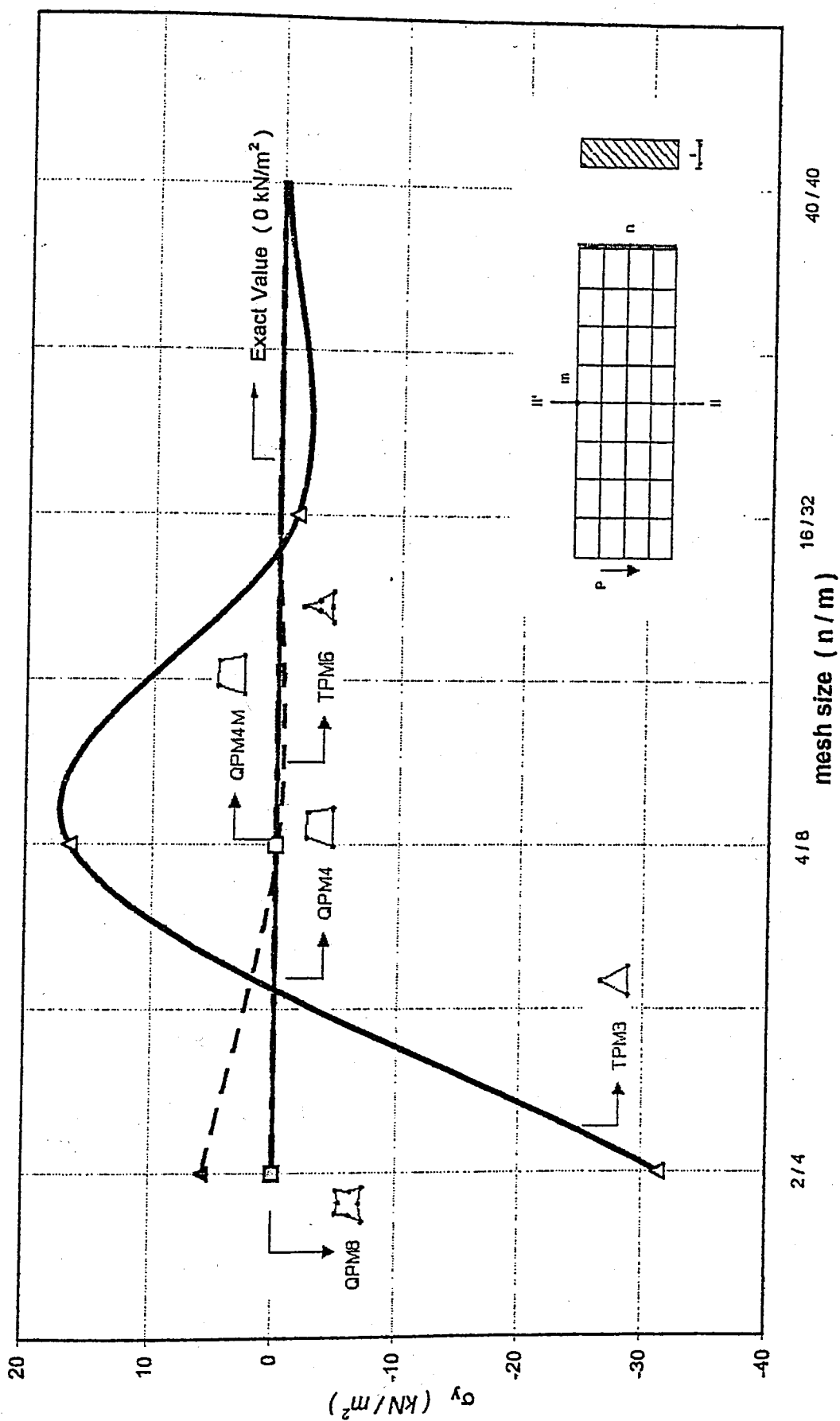


Figure B.8. Comparison of maximum normal stress (σ_x) at section II-II" (40 / 80 mesh size)

Figure B.9. Maximum normal stress (σ_y) at section II-II'

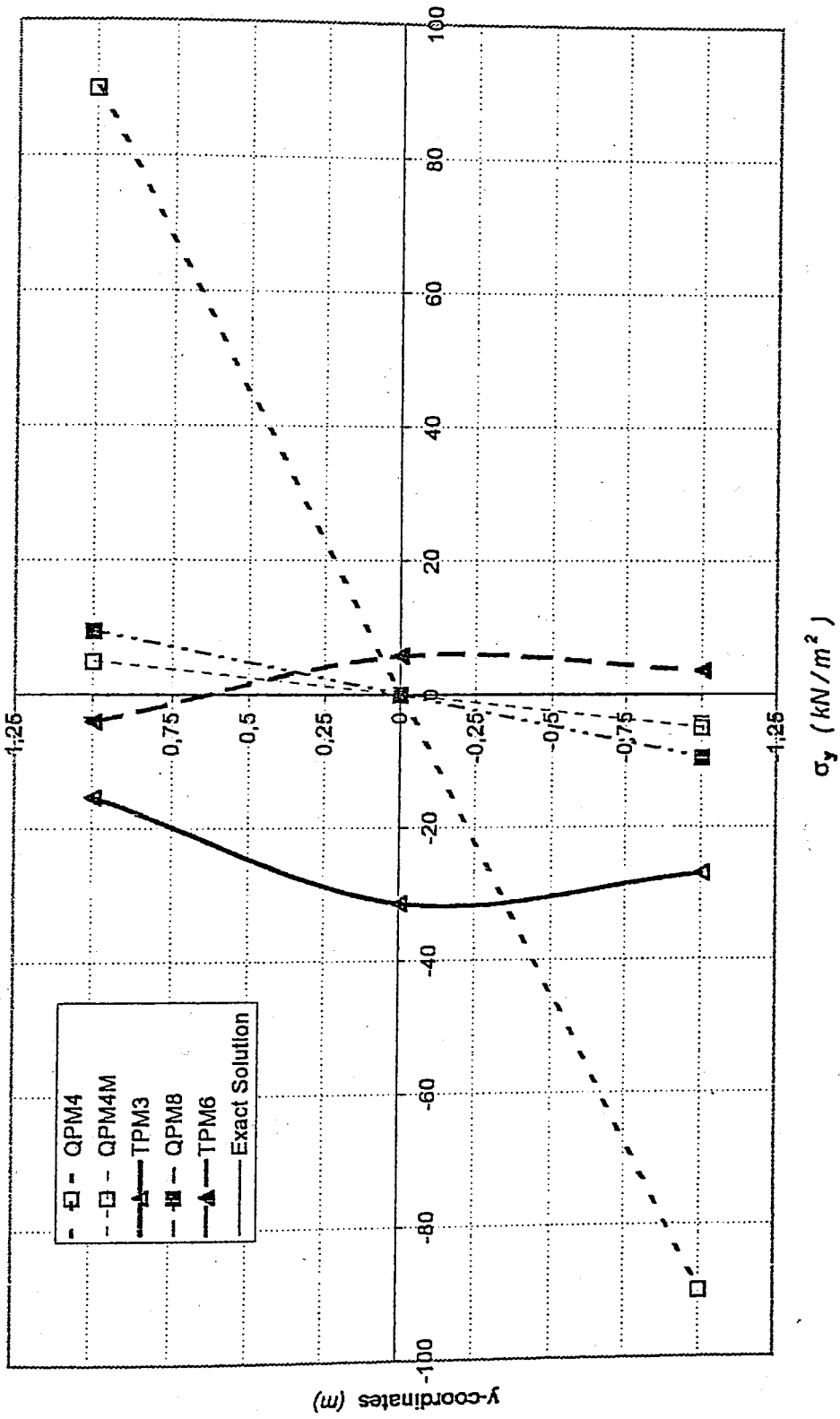


Figure B.10. Comparison of maximum normal stress (σ_y) at section II-II' (2 / 4 mesh size)

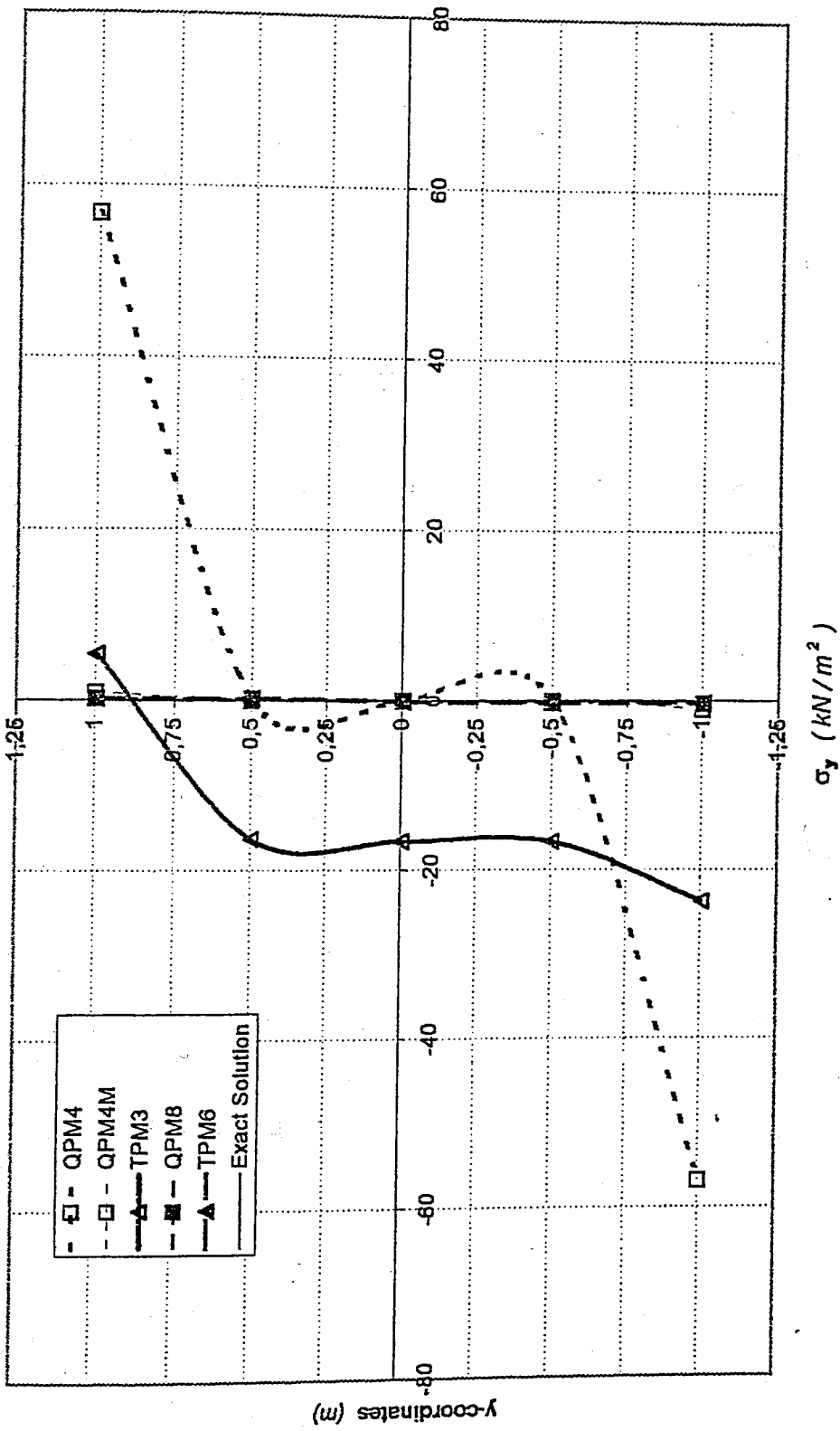


Figure B.11. Comparison of maximum normal stress (σ_y) at section II-II' (4/8 mesh size)

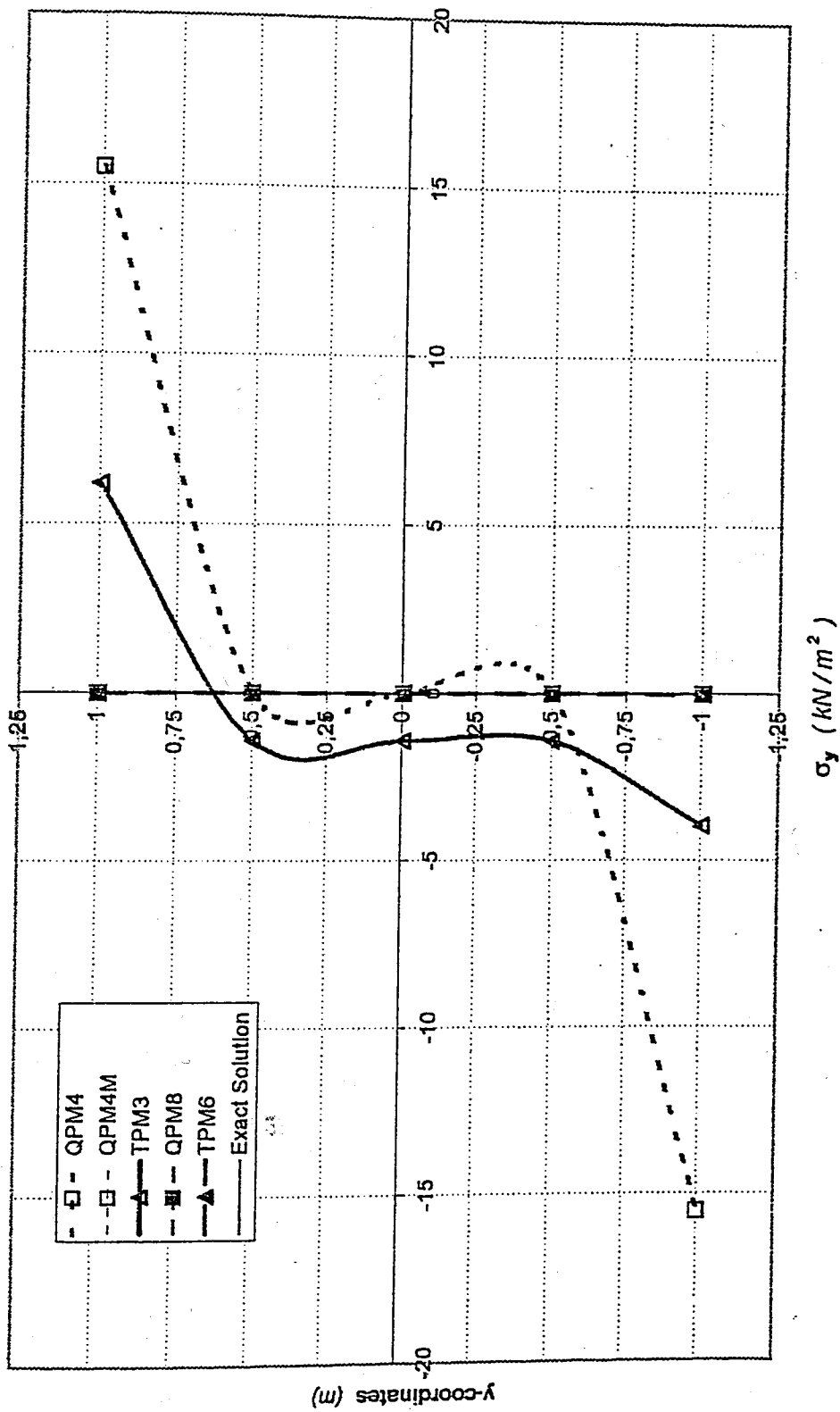


Figure B.12. Comparison of maximum normal stress (σ_y) at section II-II' (16 / 32 mesh size)

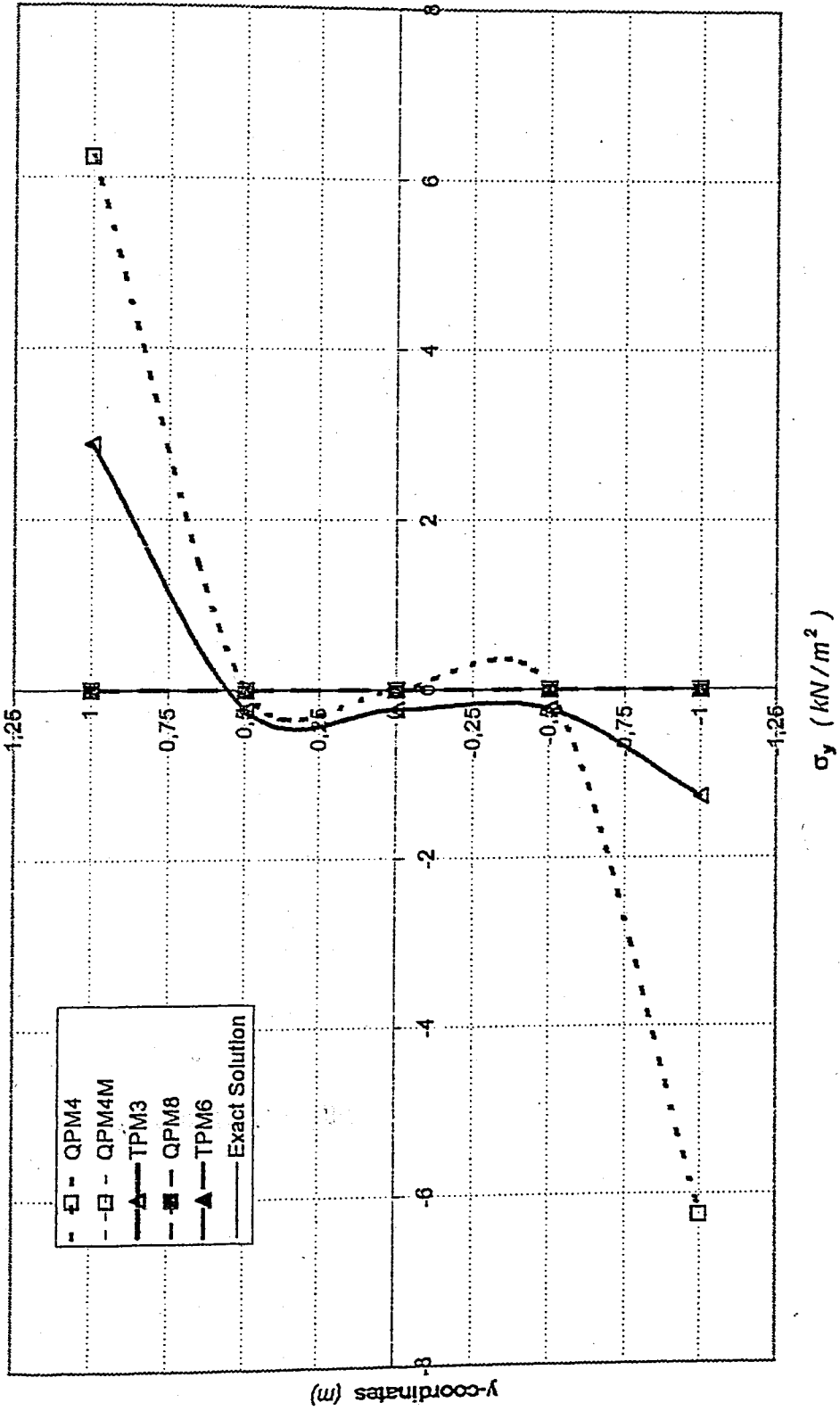


Figure B.13. Comparison of maximum normal stress (σ_y) at section II-II' (40 / 80 mesh size)

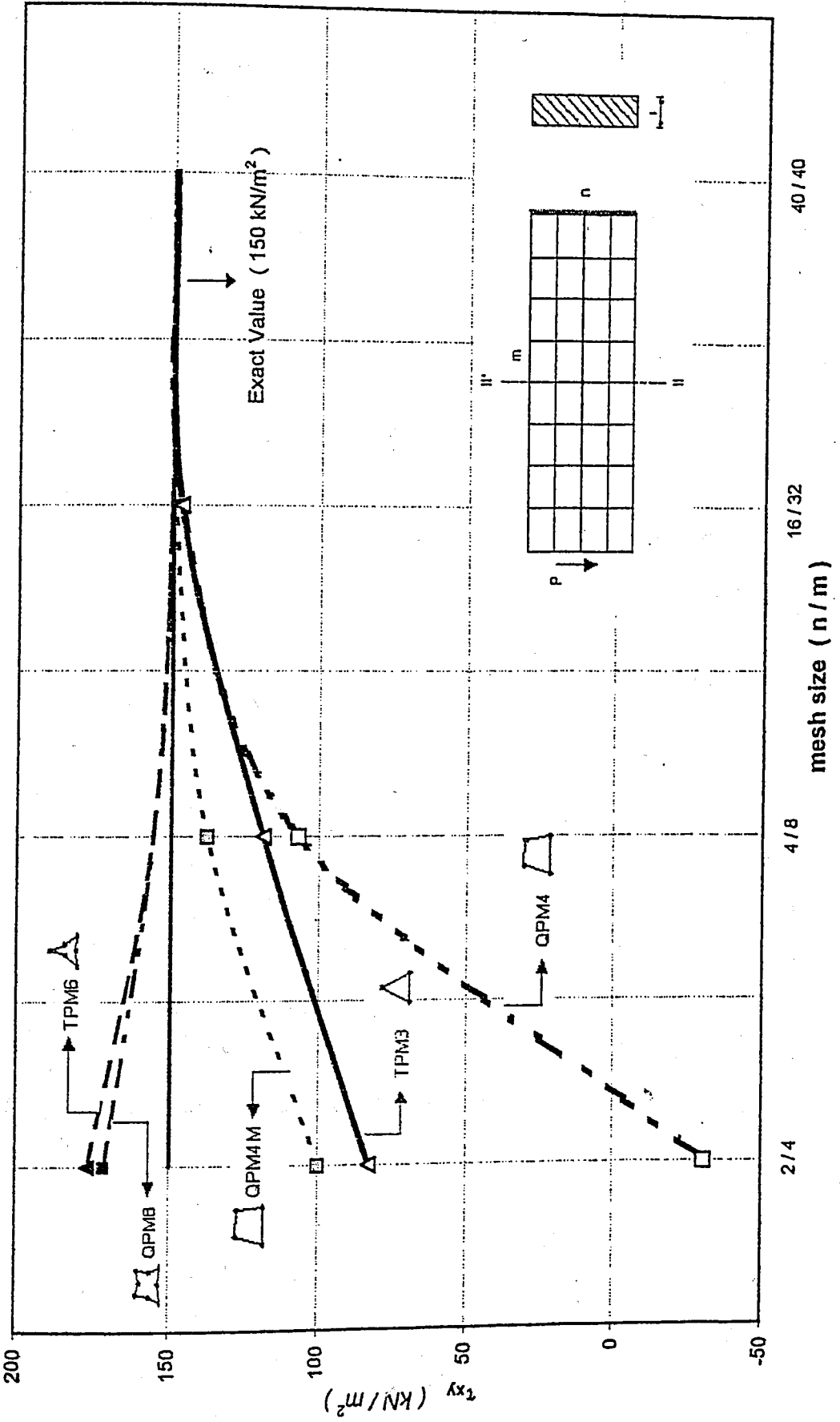


Figure B.14. Maximum shear stress (τ_{xy}) at section II-II'

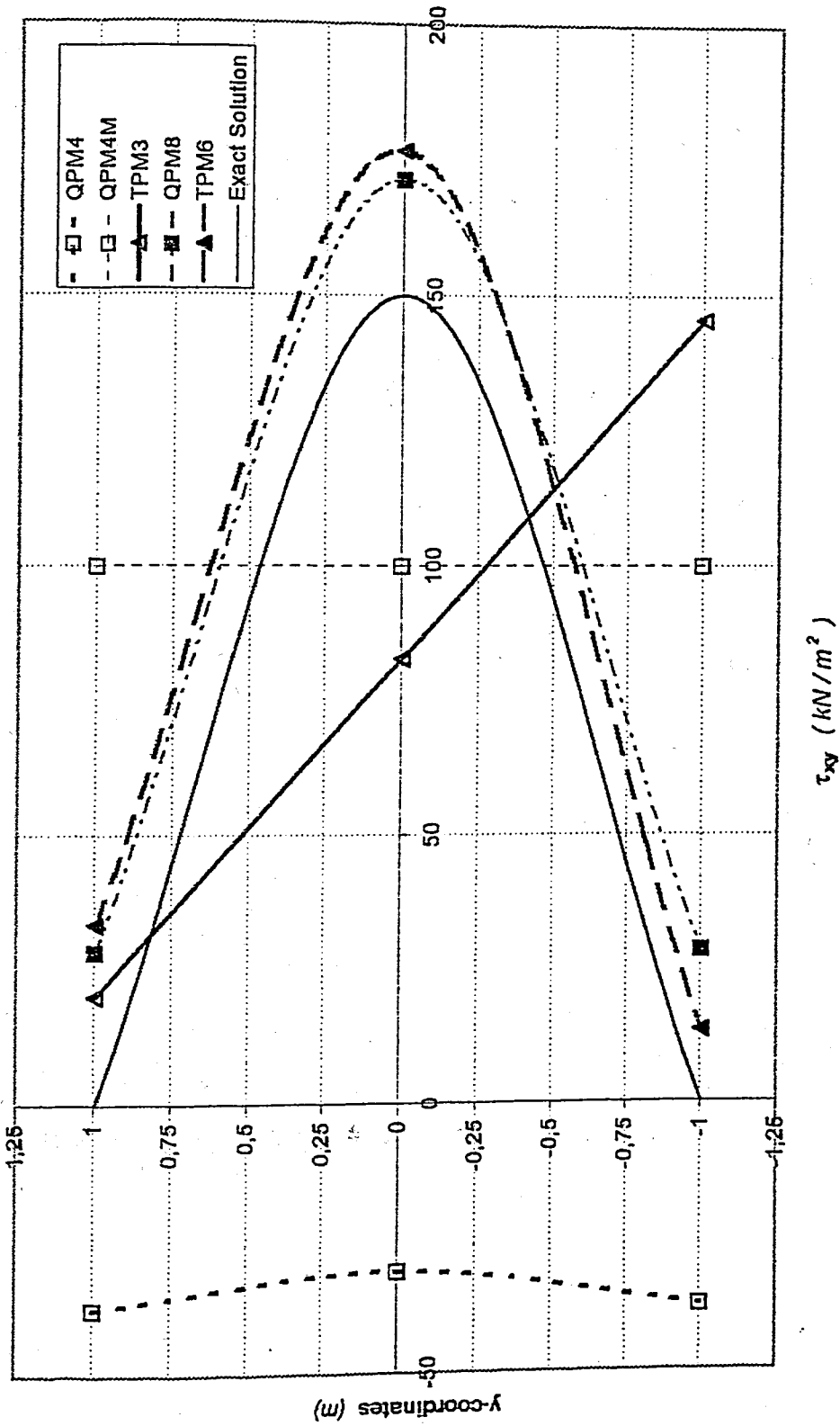


Figure B.15. Comparison of maximum shear stress (τ_{xy}) at section II-II' (2 / 4 mesh size)

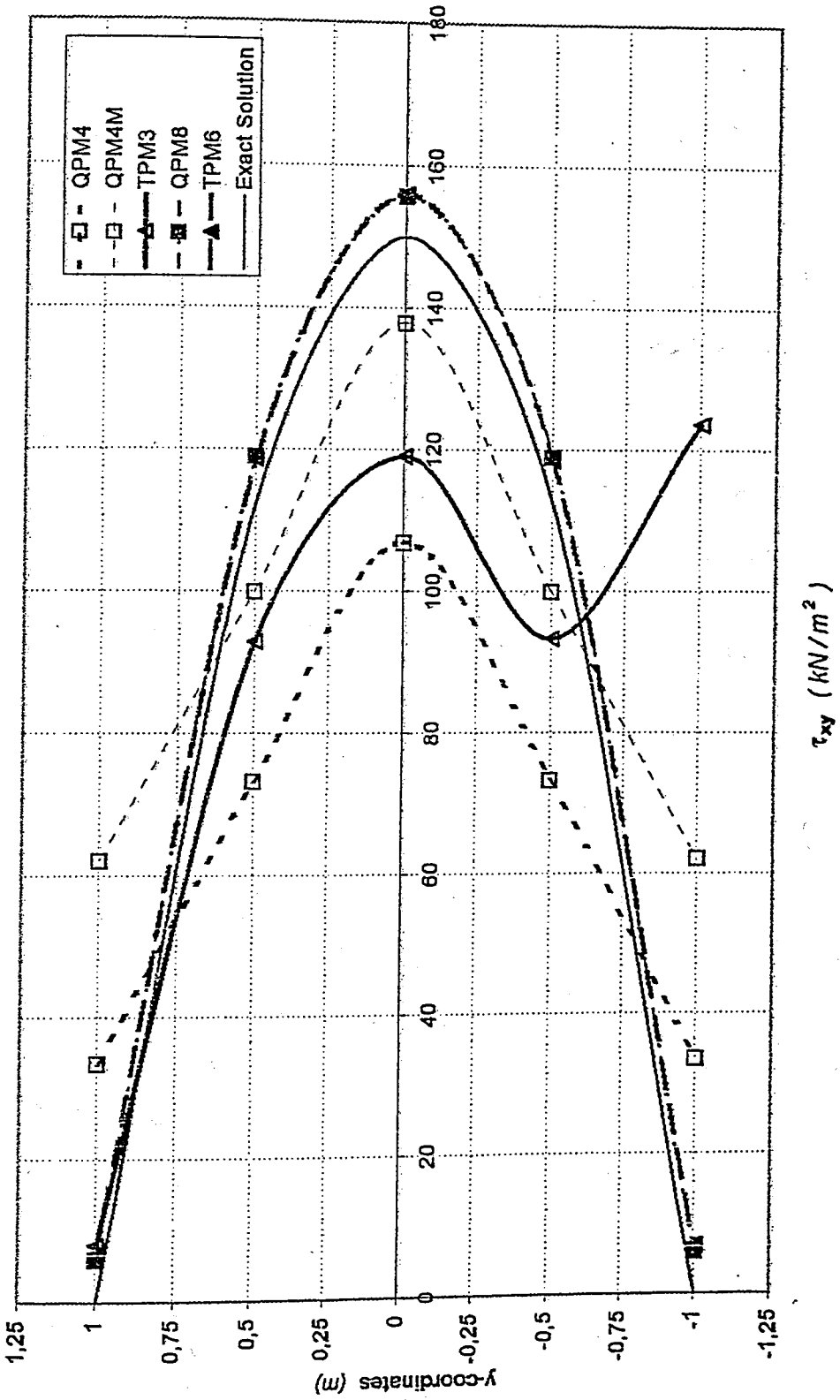


Figure B.16. Comparison of maximum shear stress (τ_{xy}) at section II-II' (4/8 mesh size)

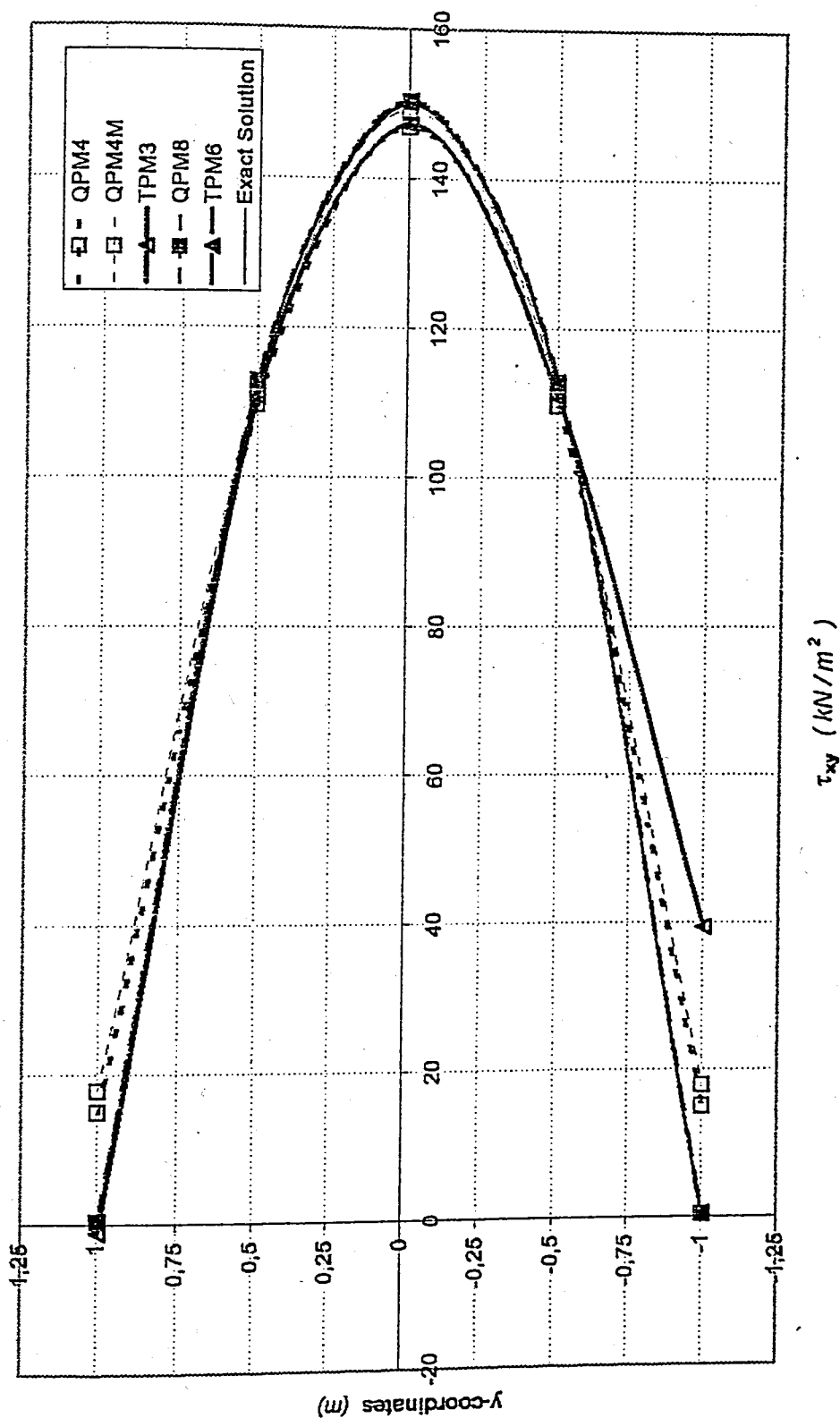


Figure B.17. Comparison of maximum shear stress (τ_{xy}) at section II-II' (16 / 32 mesh size)

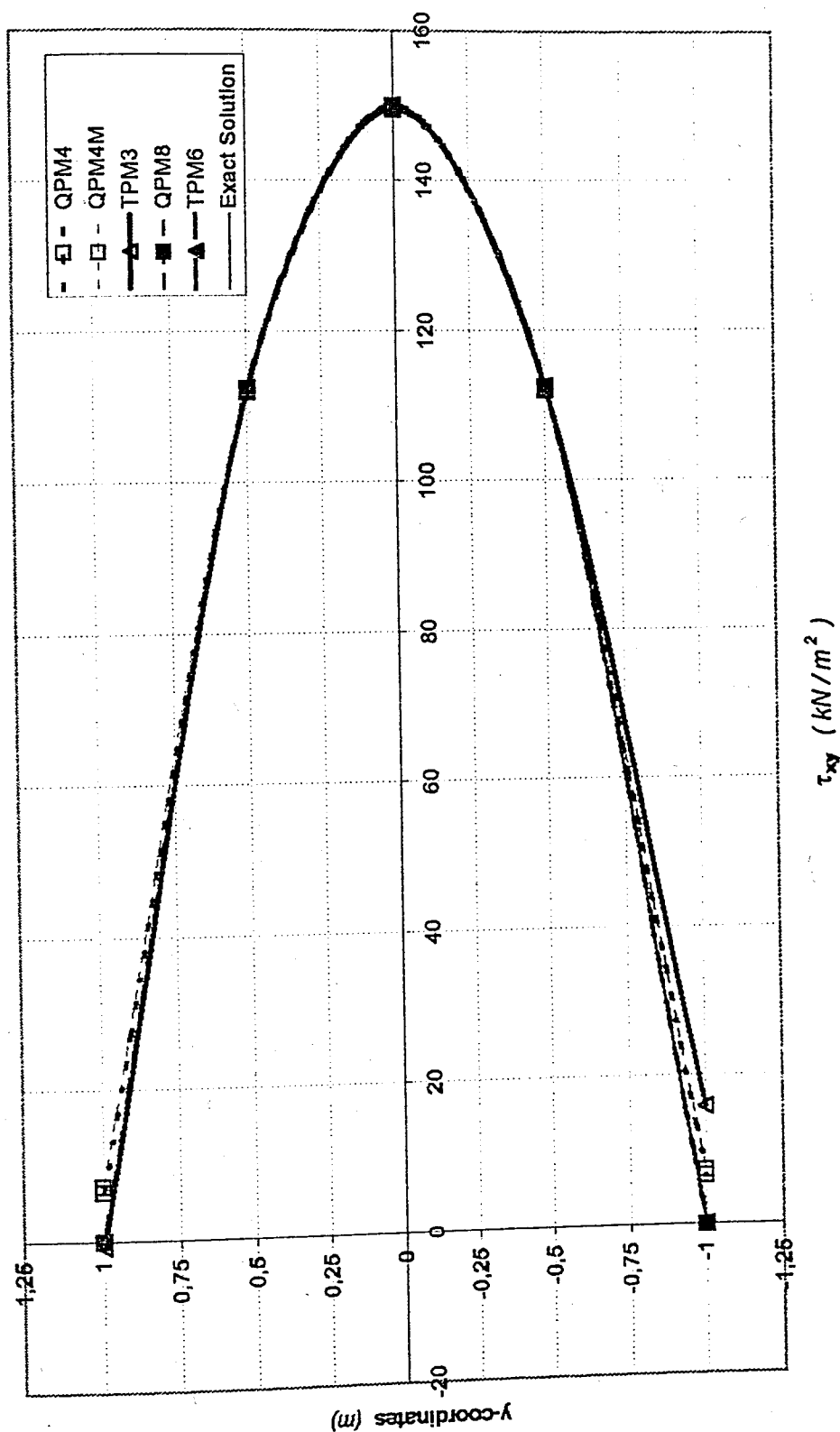


Figure B.18. Comparison of maximum shear stress (τ_{xy}) at section II-II' (40/80 mesh size)

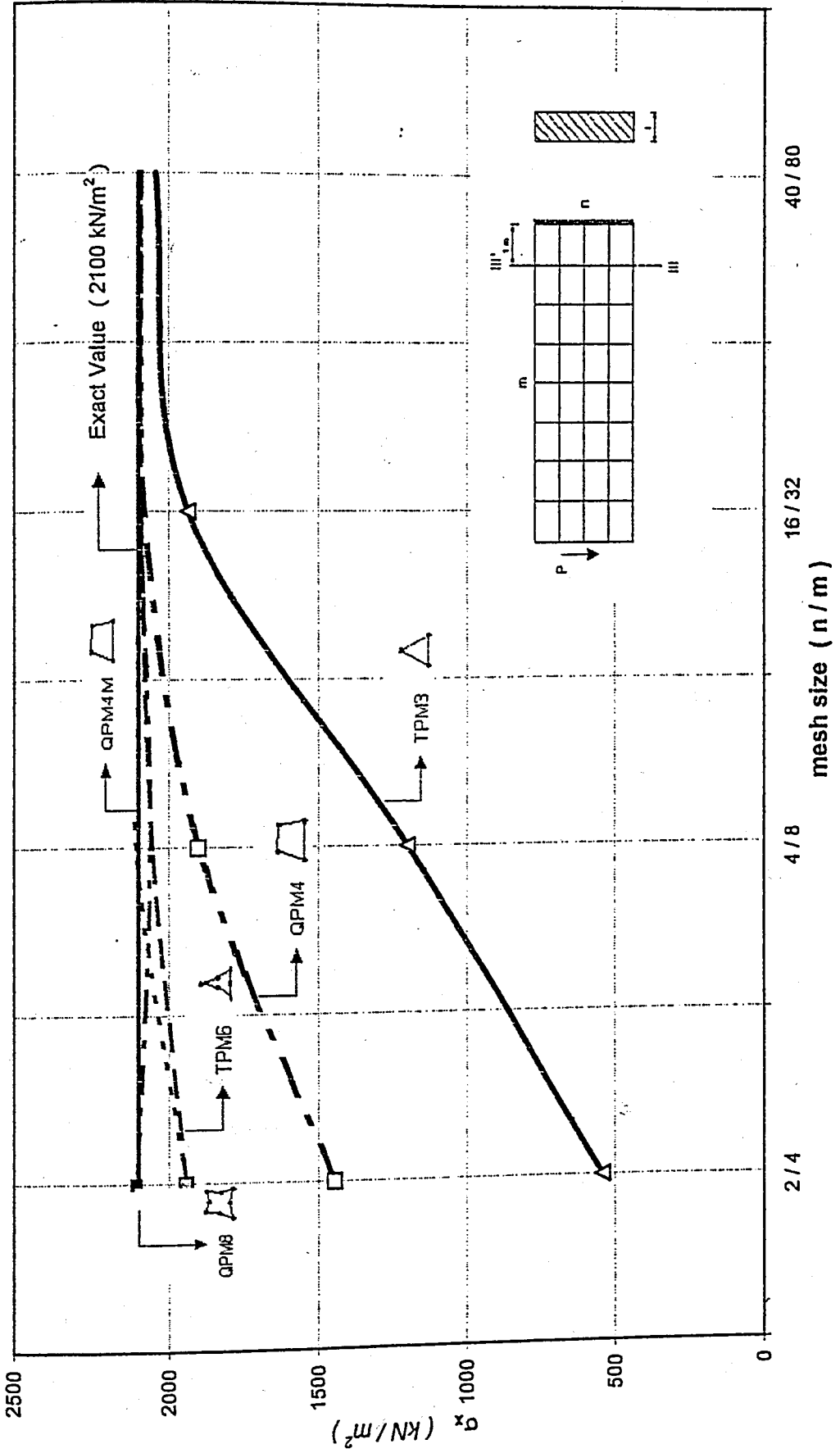


Figure B.19. Maximum normal stress (σ_x) at section III-III"

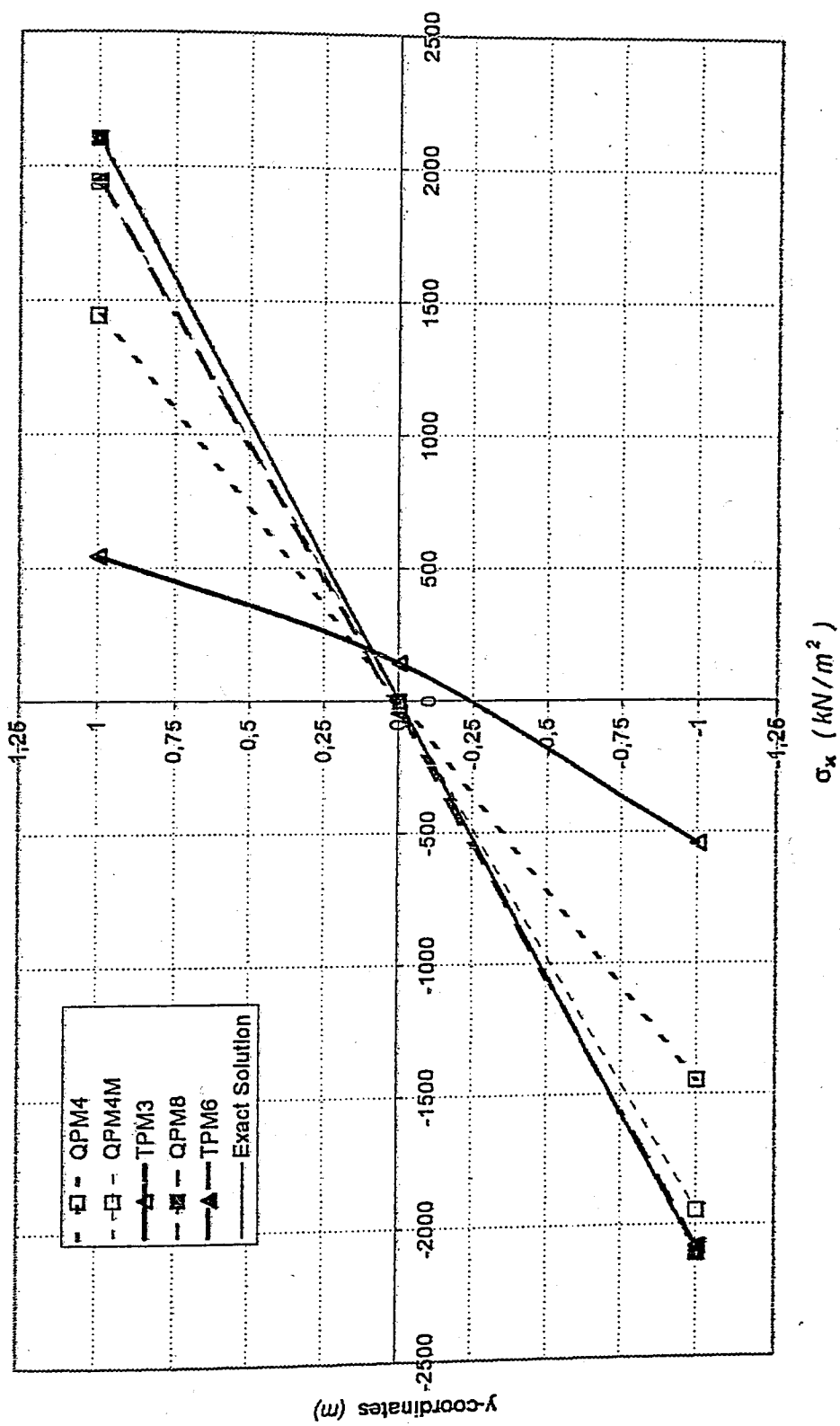
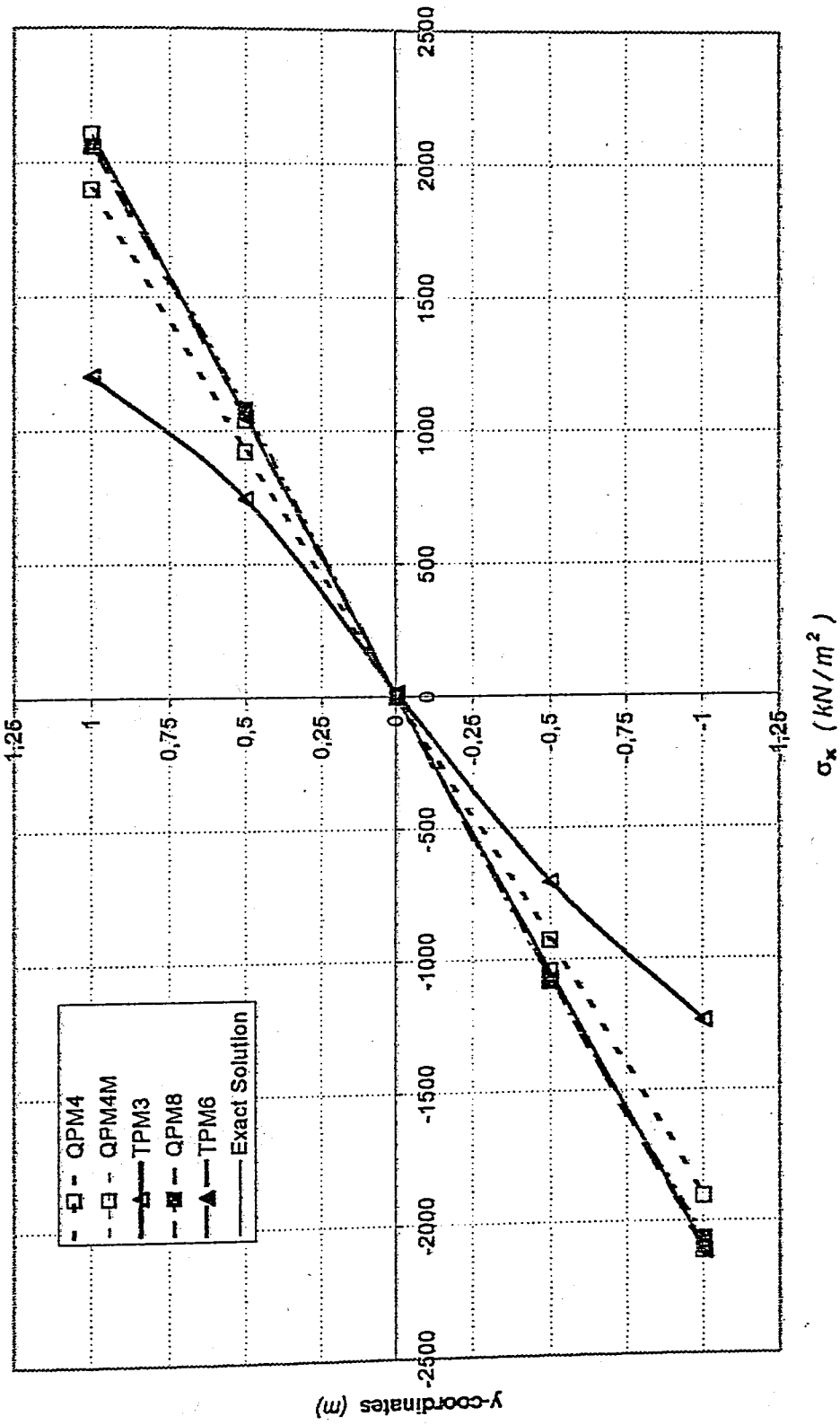


Figure B.20. Comparison of maximum normal stress (σ_x) at section III-III' (2 / 4 mesh size)

Figure B.21. Comparison of maximum normal stress (σ_x) at section III-III' (4/8 mesh size)

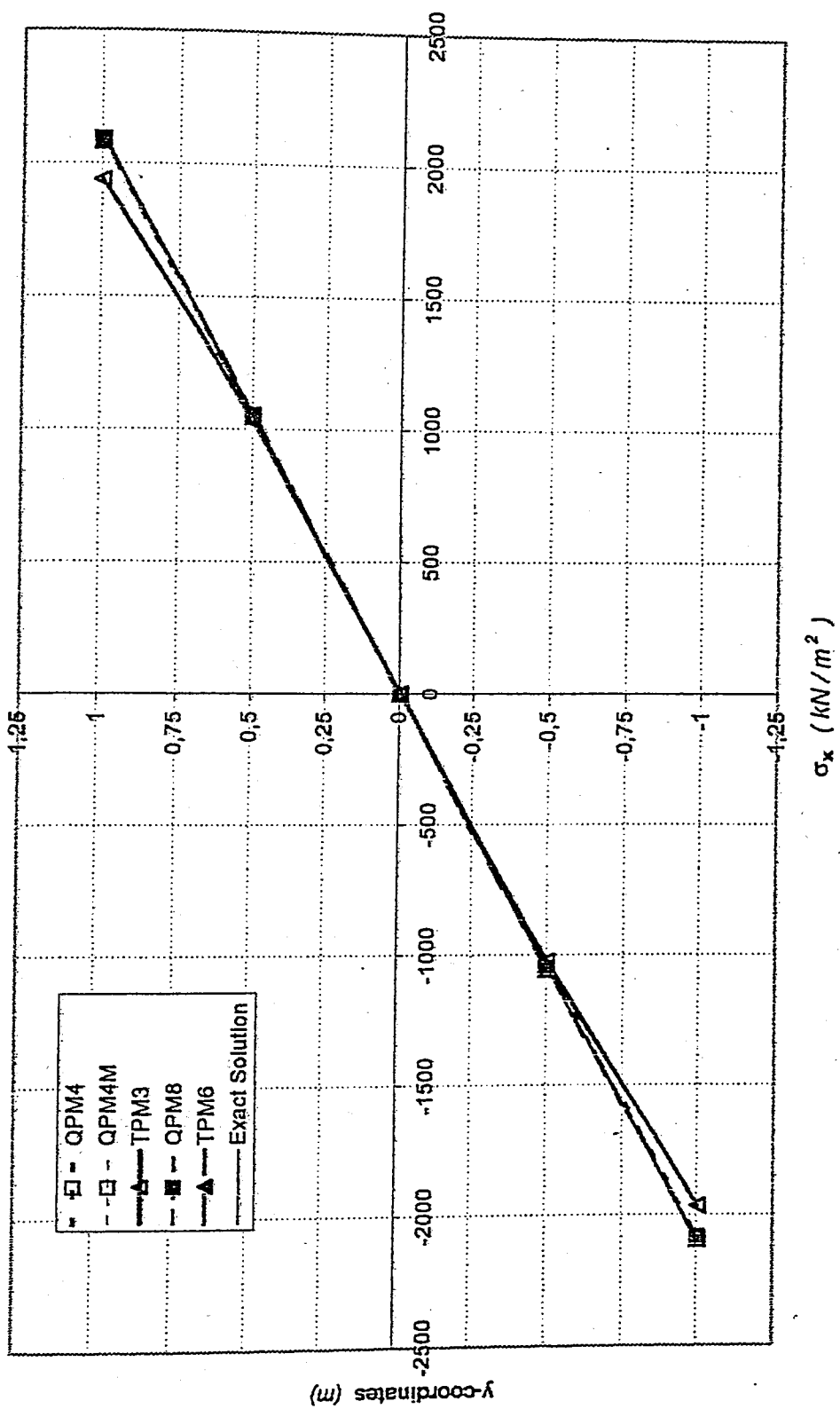


Figure B.22. Comparison of maximum normal stress (σ_x) at section III-III' (16 / 32 mesh size)

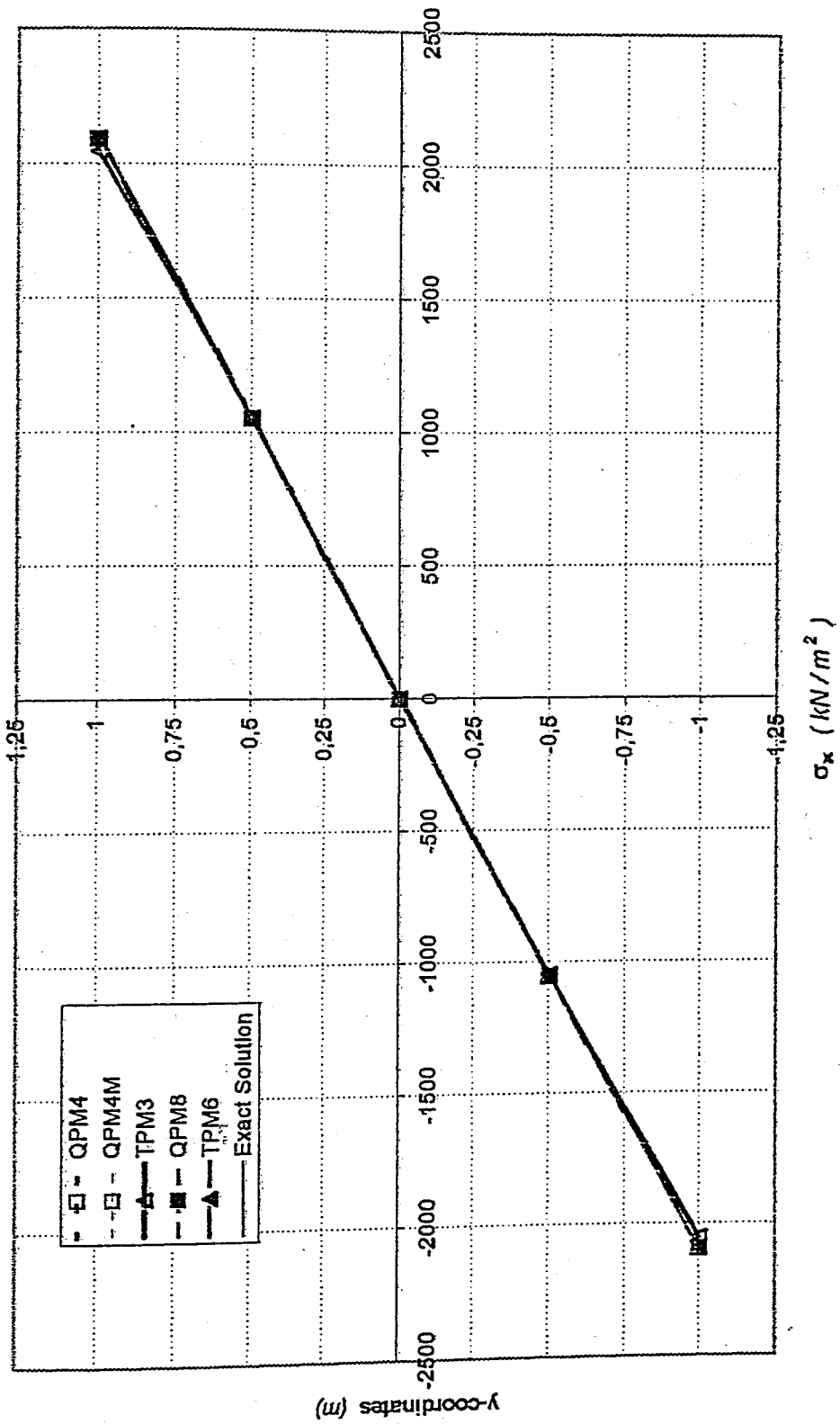


Figure B.23. Comparison of maximum normal stress (σ_x) at section III-III' (40 / 80 mesh size)

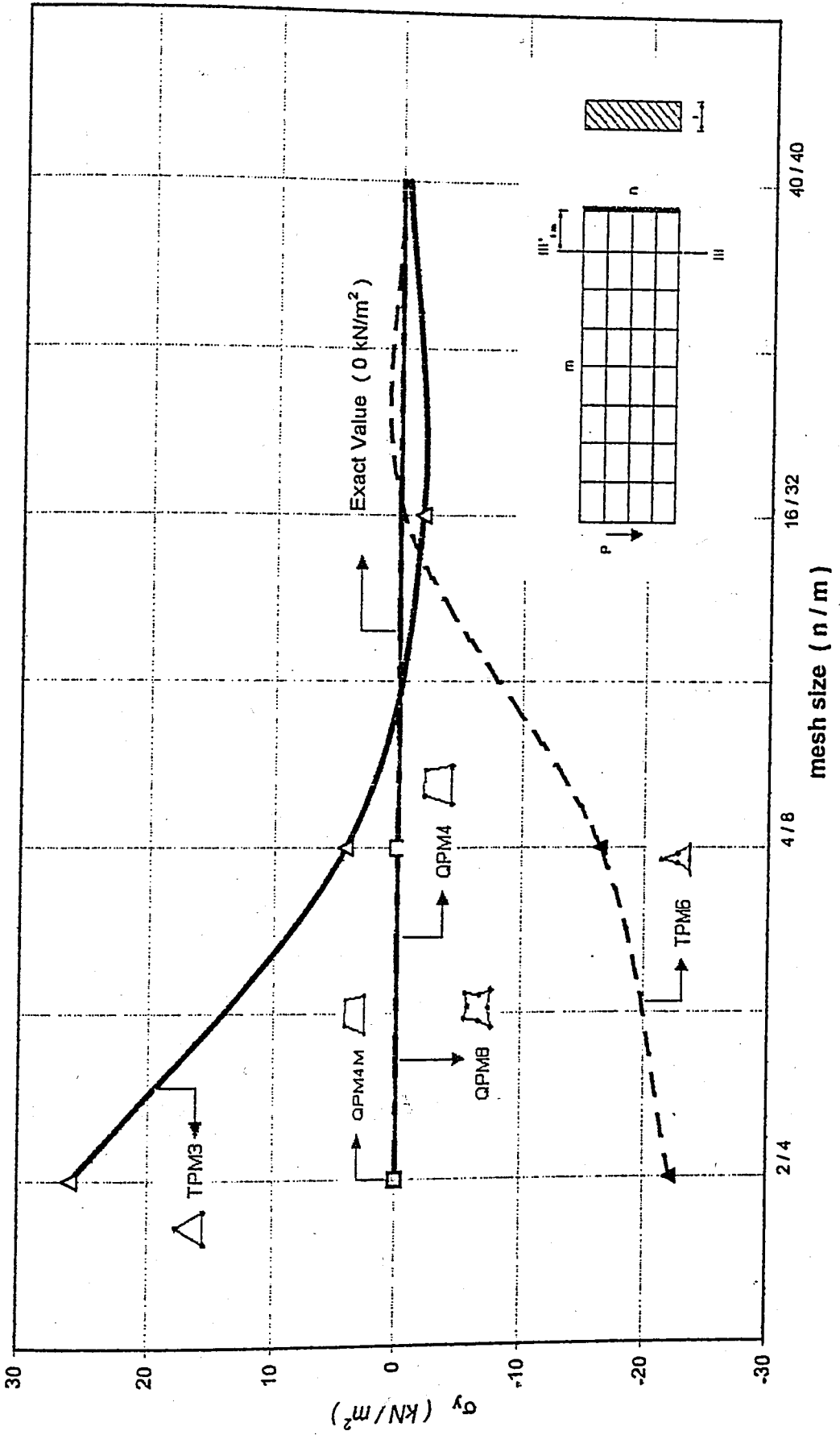


Figure B.24. Maximum normal stress (σ_y) at section III-III'

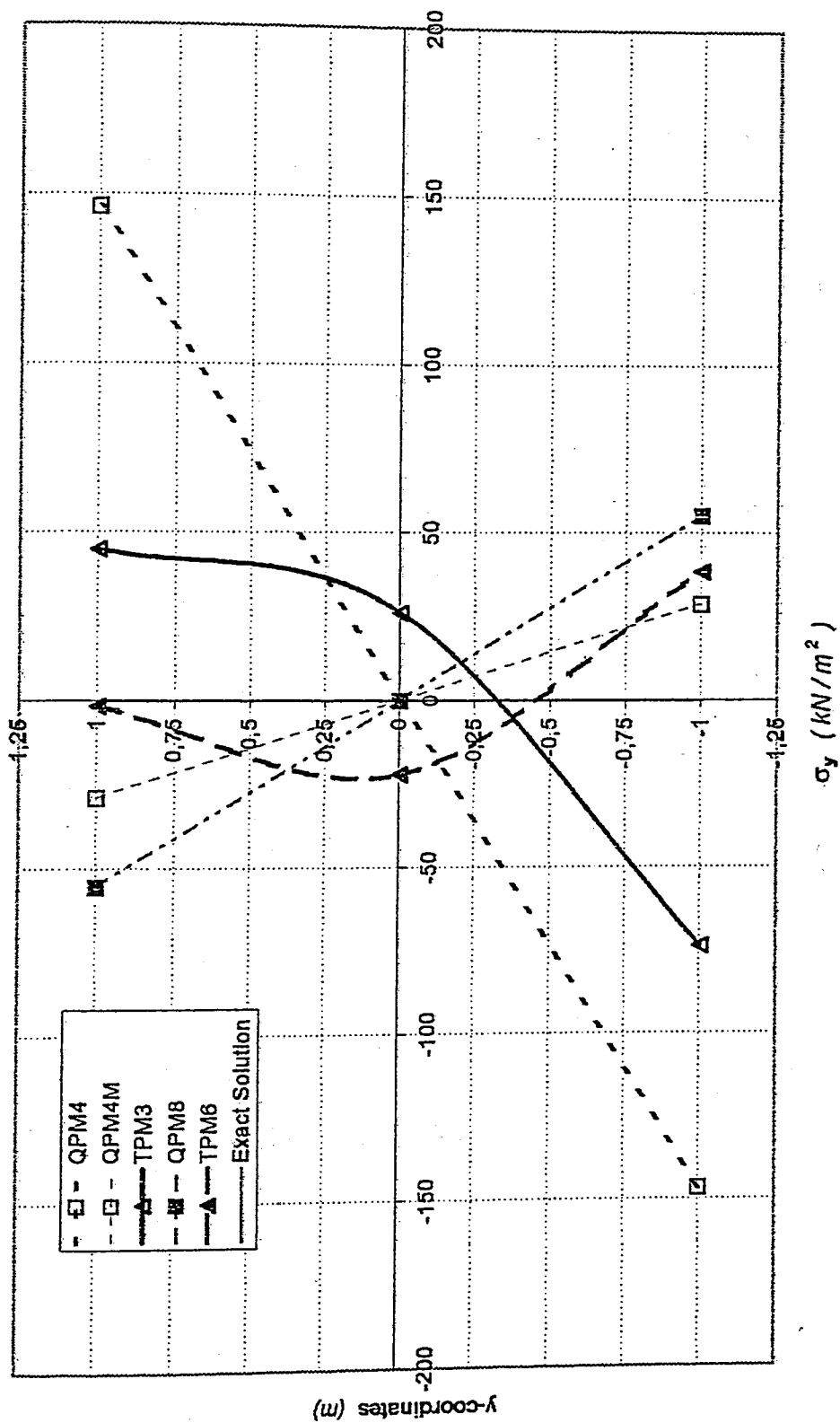


Figure B.25. Comparison of maximum normal stress (σ_y) at section III-III' (2/4 mesh size)

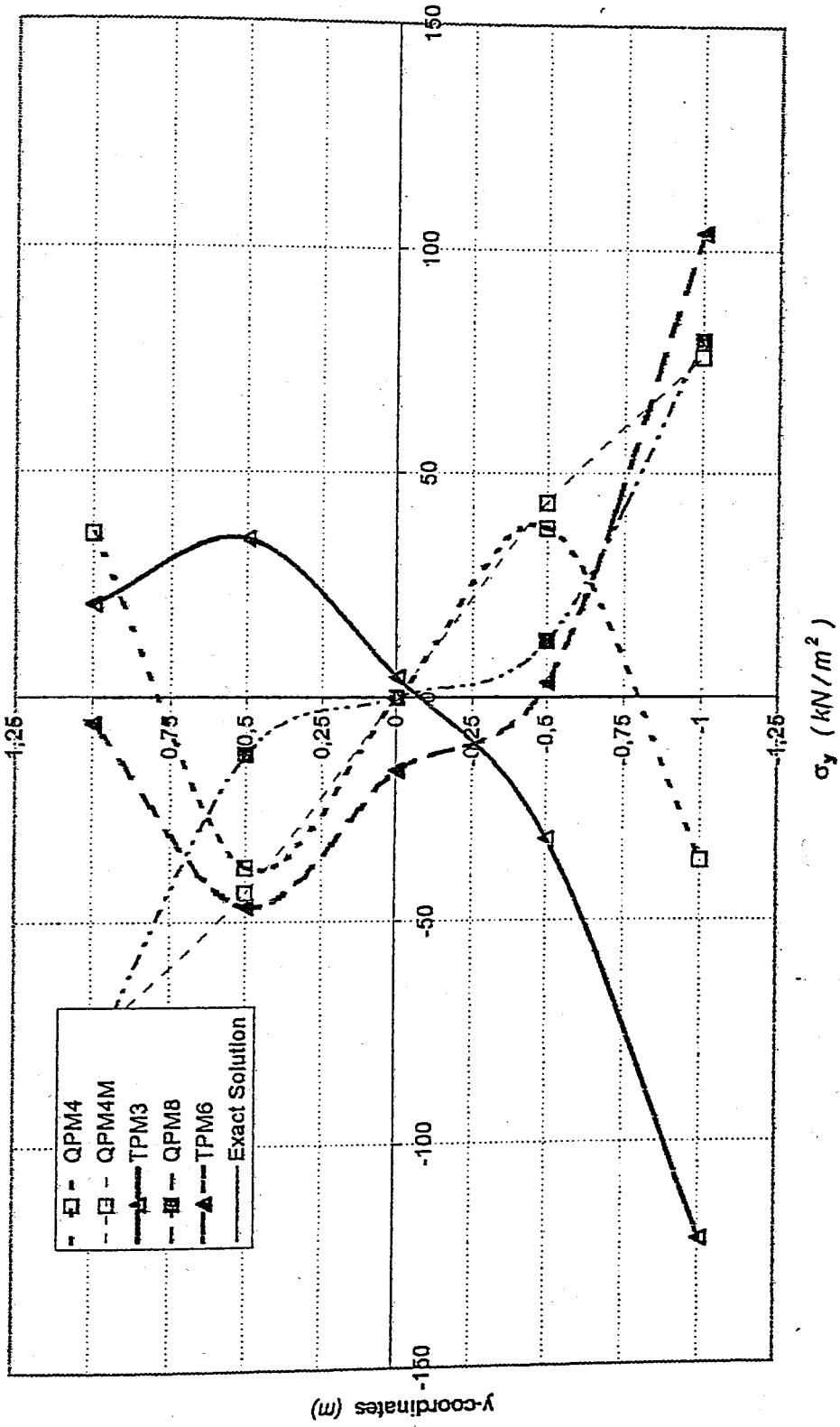


Figure B.26. Comparison of maximum normal stress (σ_y) at section III-III' (4/8 mesh size)

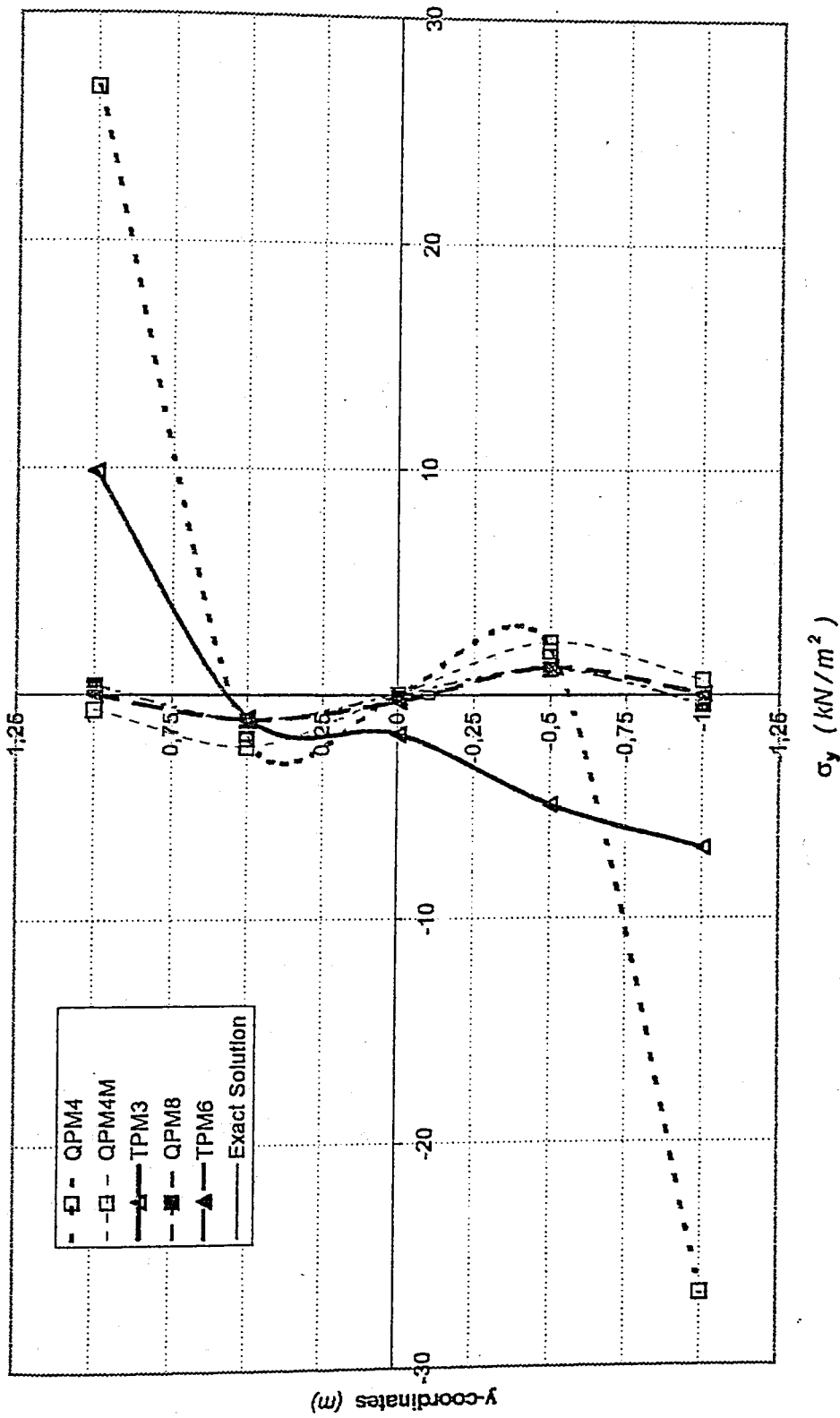


Figure B.27. Comparison of maximum normal stress (σ_y) at section III-III' (16 / 32 mesh size)

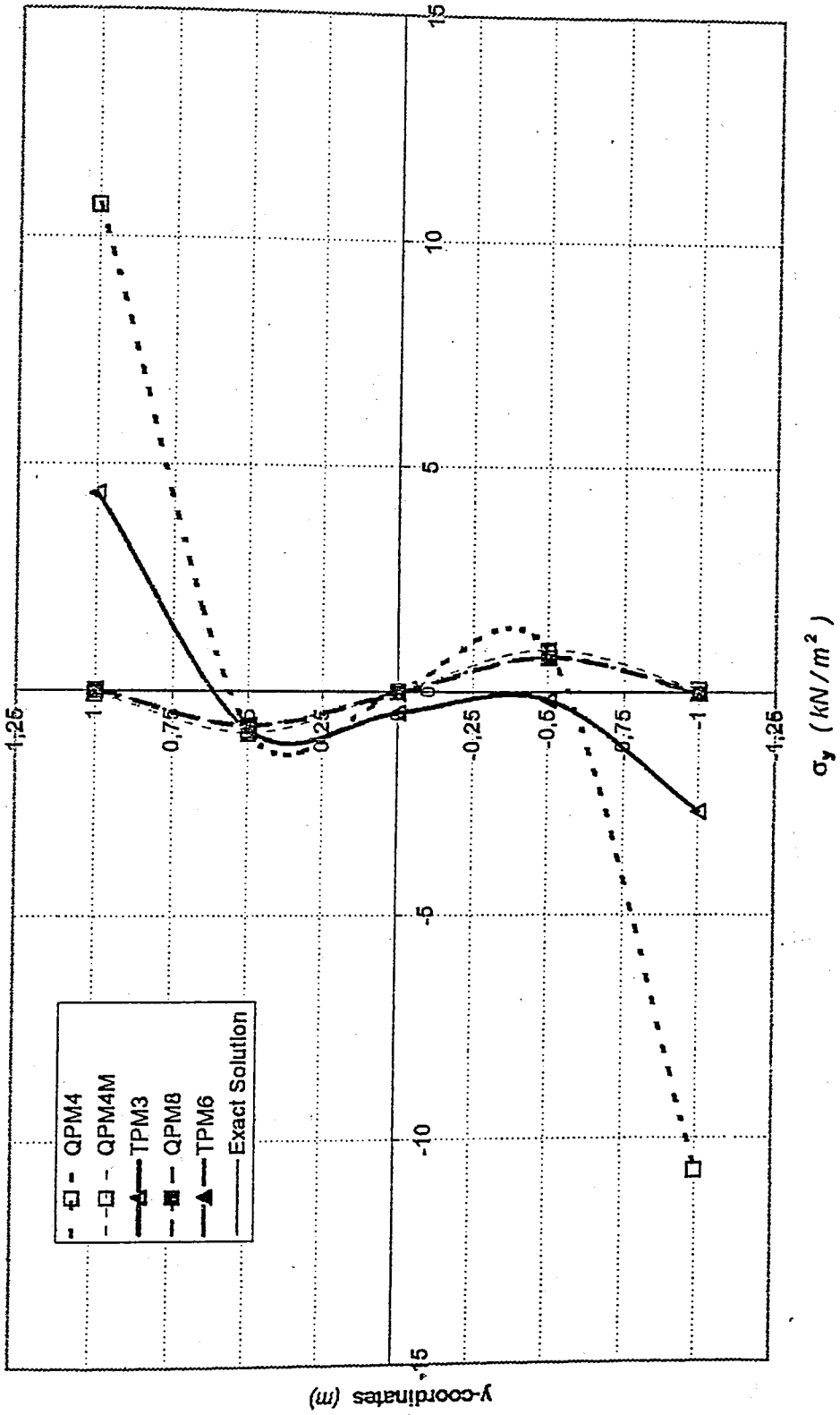


Figure B.28. Comparison of maximum normal stress (σ_y) at section III-III' (40 / 80 mesh size)

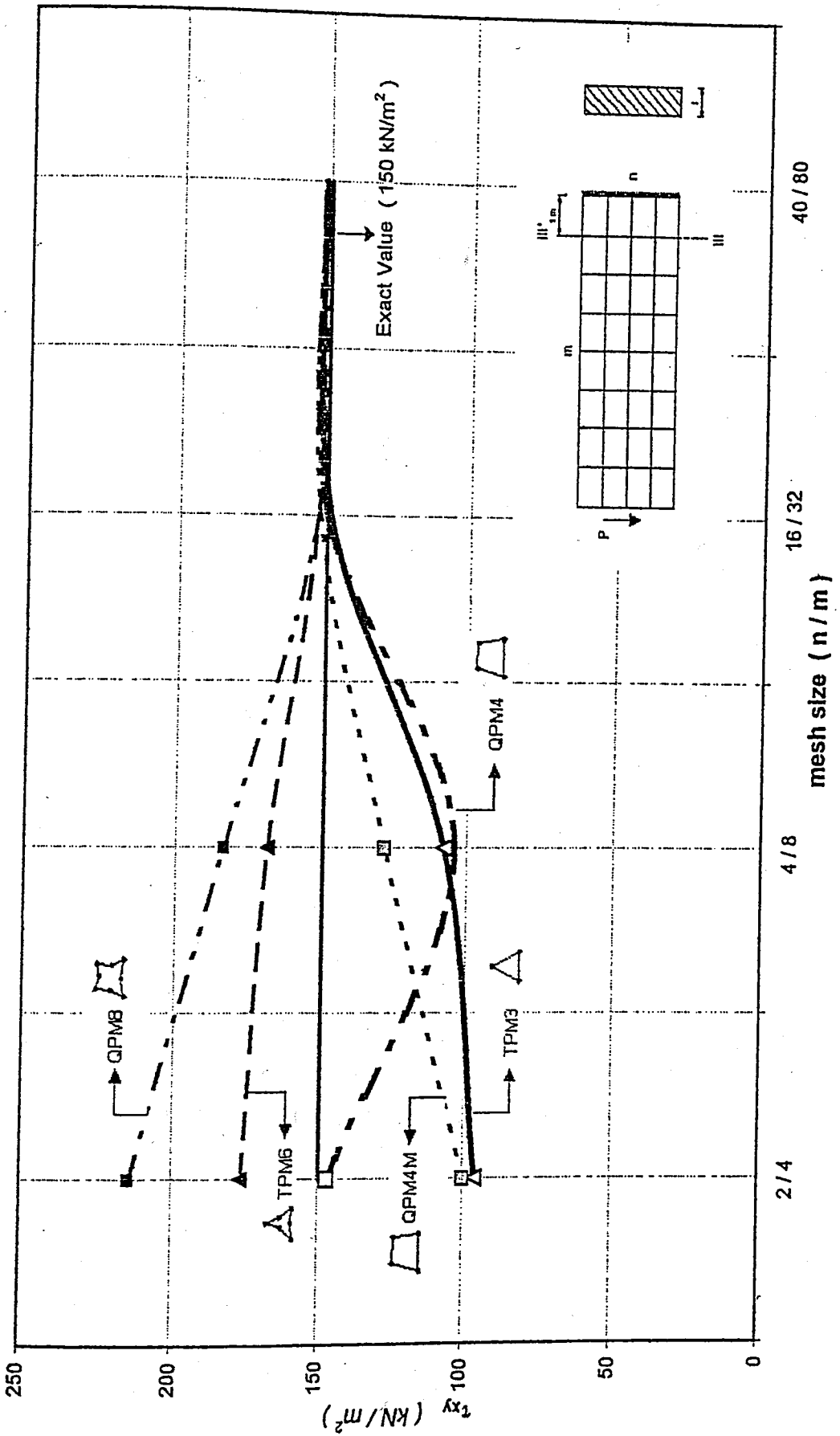


Figure B.29. Maximum shear stress (τ_{xy}) at section III-III'

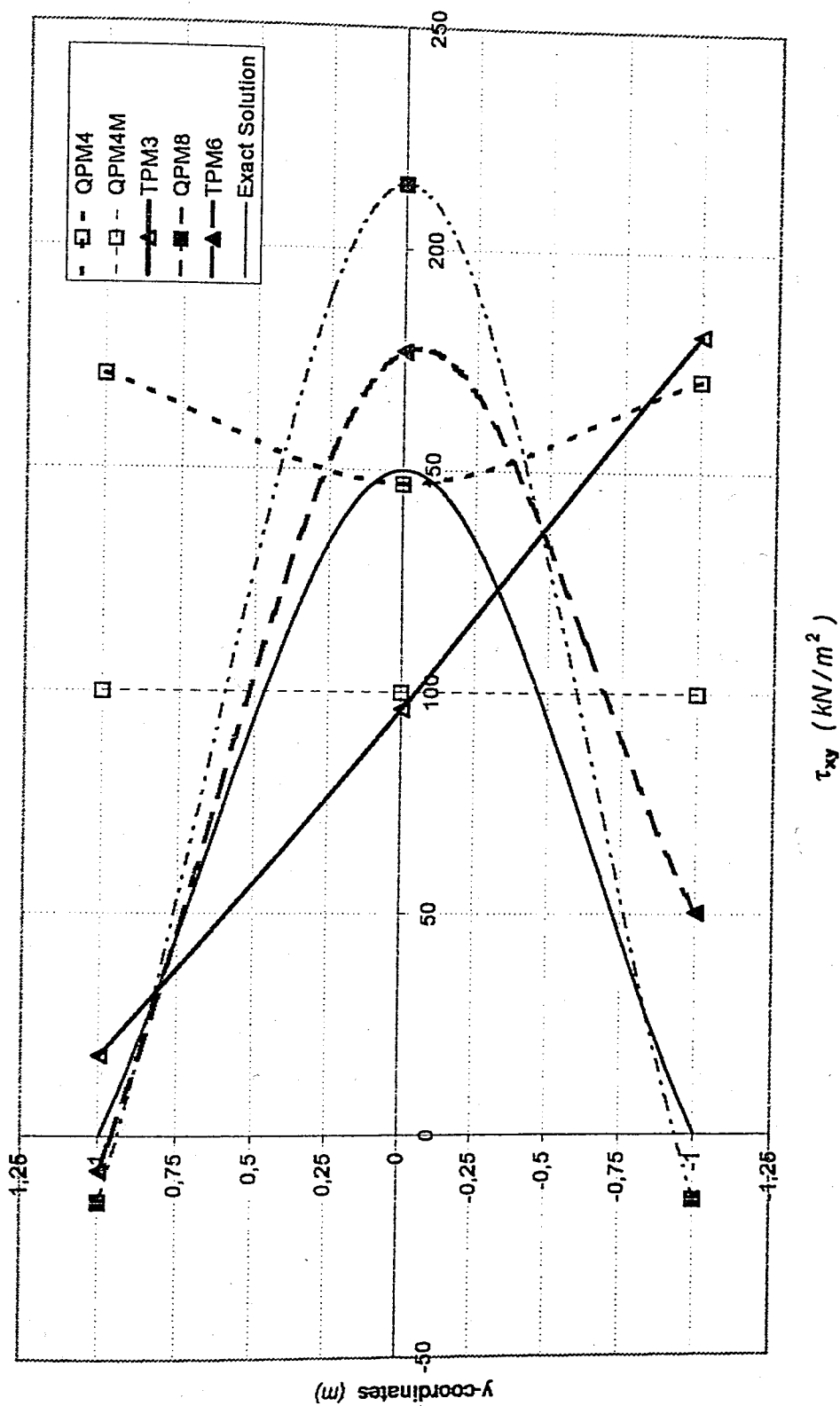


Figure B.30. Comparison of maximum shear stress (τ_{xy}) at section III-III' (2 / 4 mesh size)

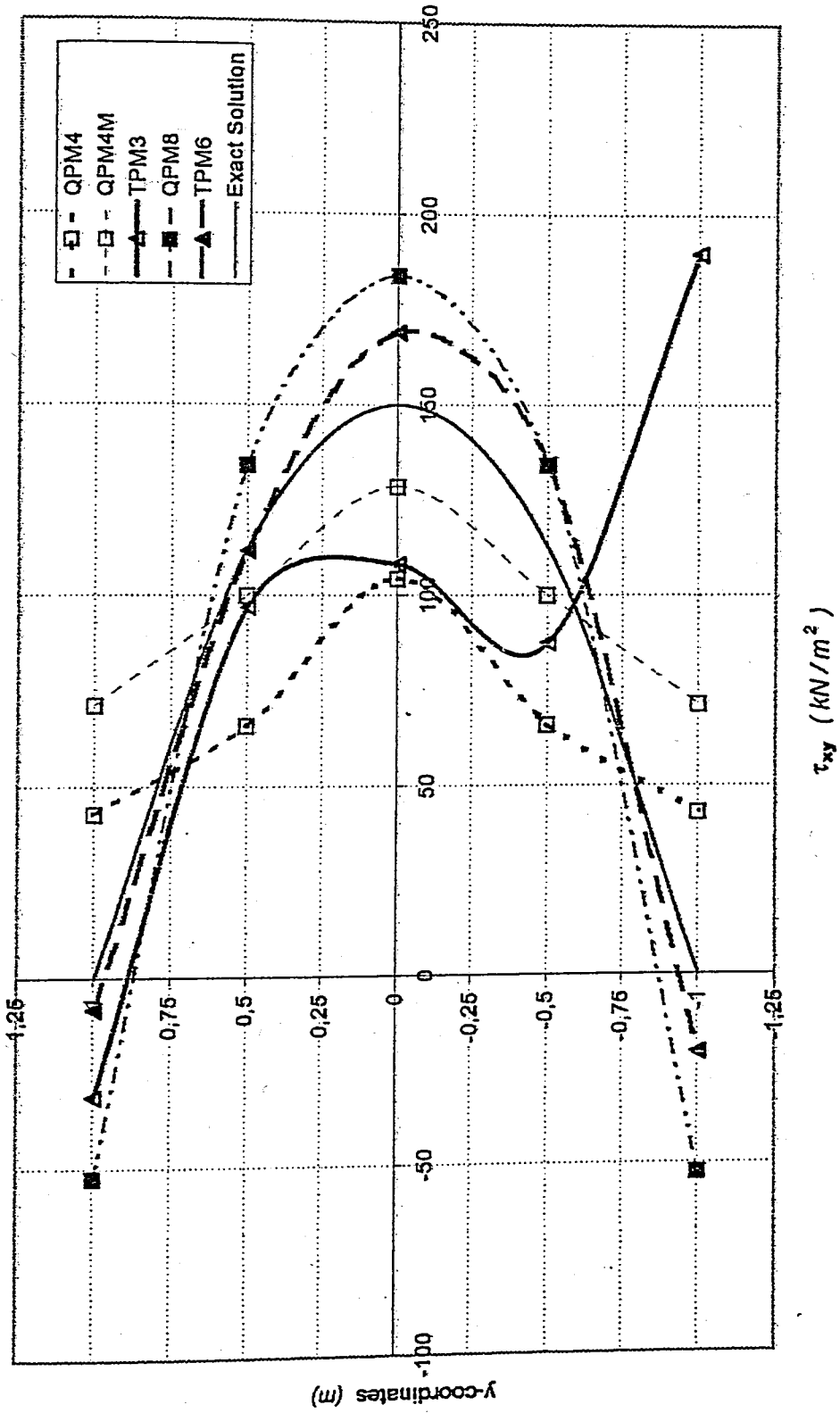


Figure B.31. Comparison of maximum shear stress (τ_{xy}) at section III-III' (4 / 8 mesh size)

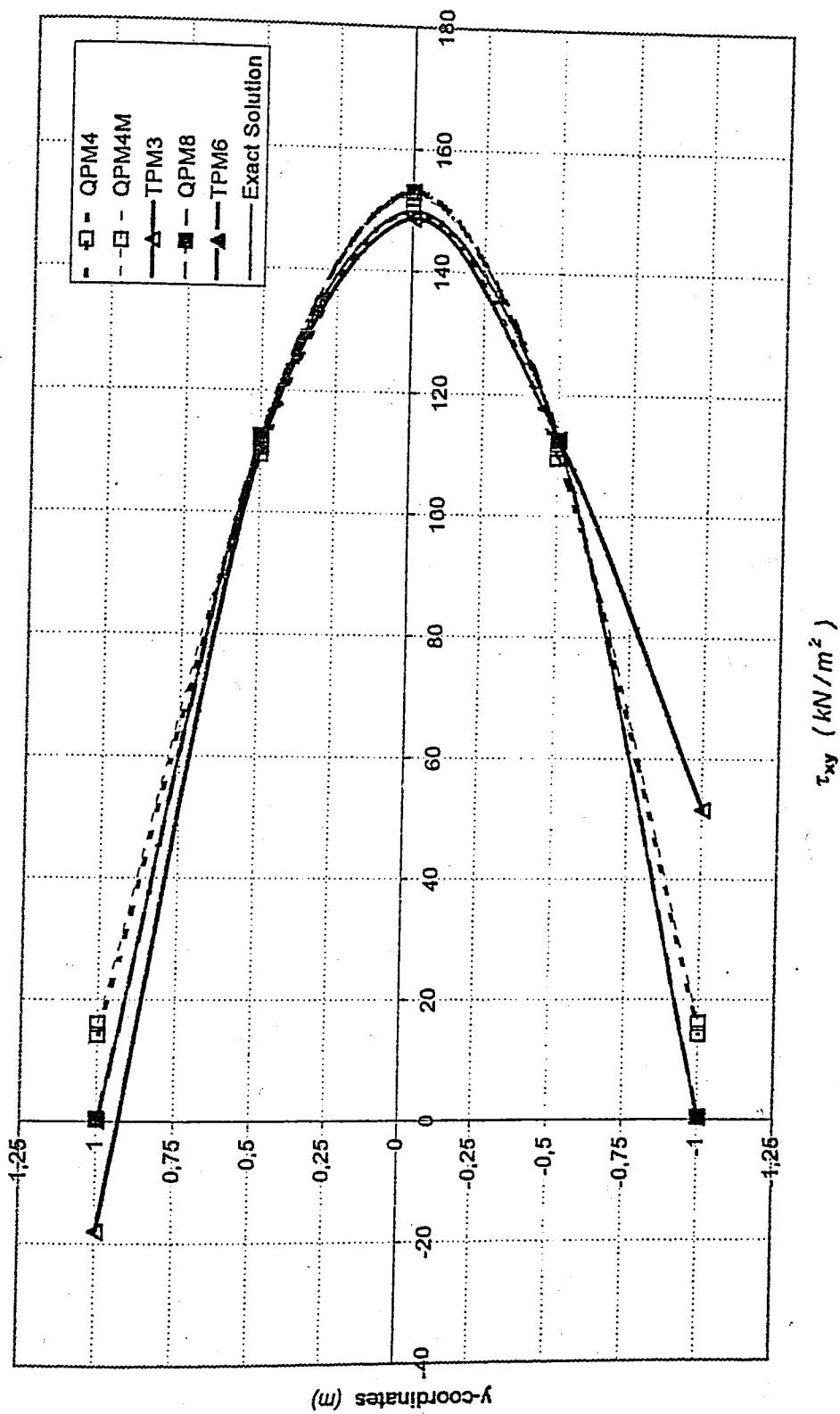


Figure B.32. Comparison of Maximum Shear Stress (τ_{xy}) at section III-III' (16 / 32 mesh size)

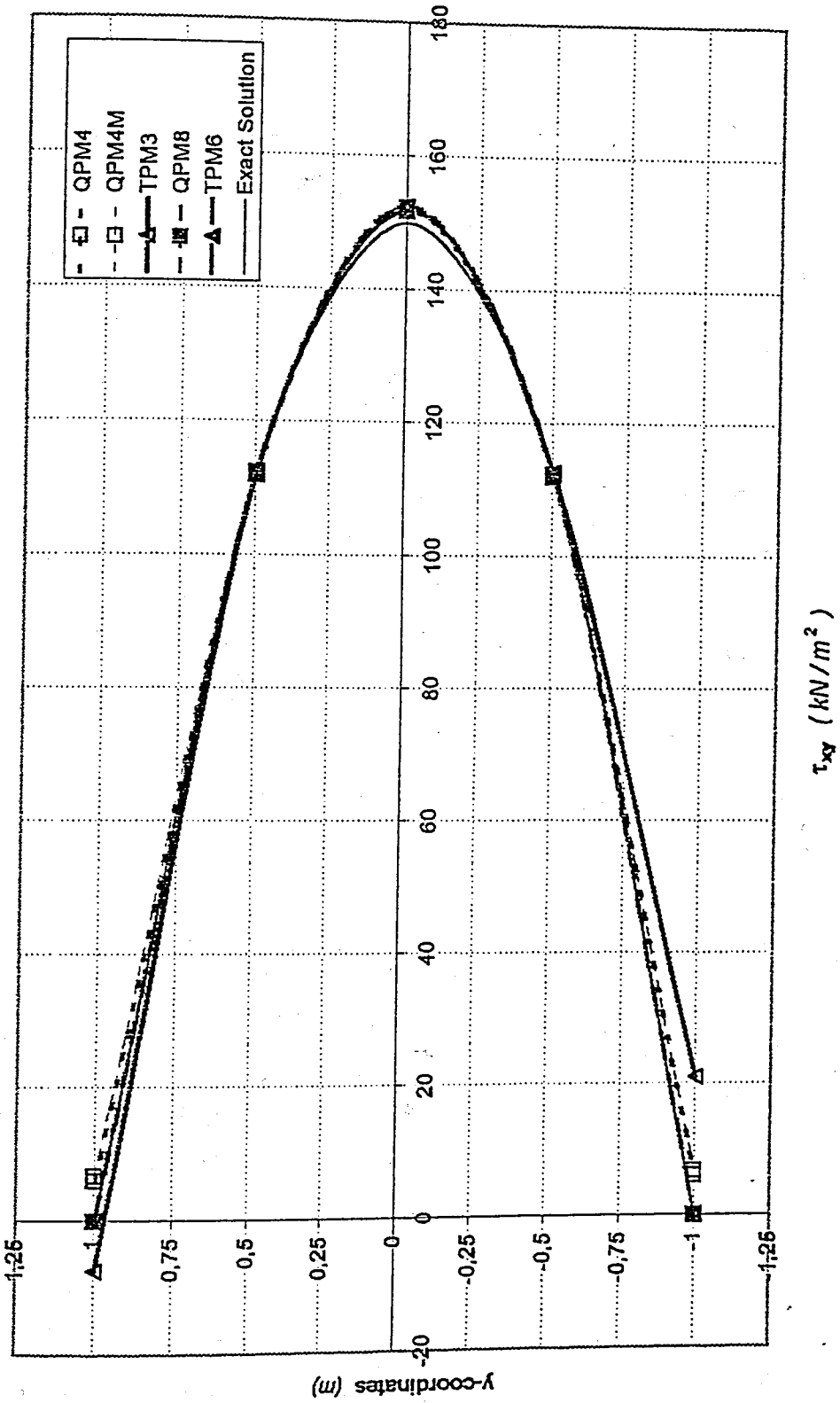
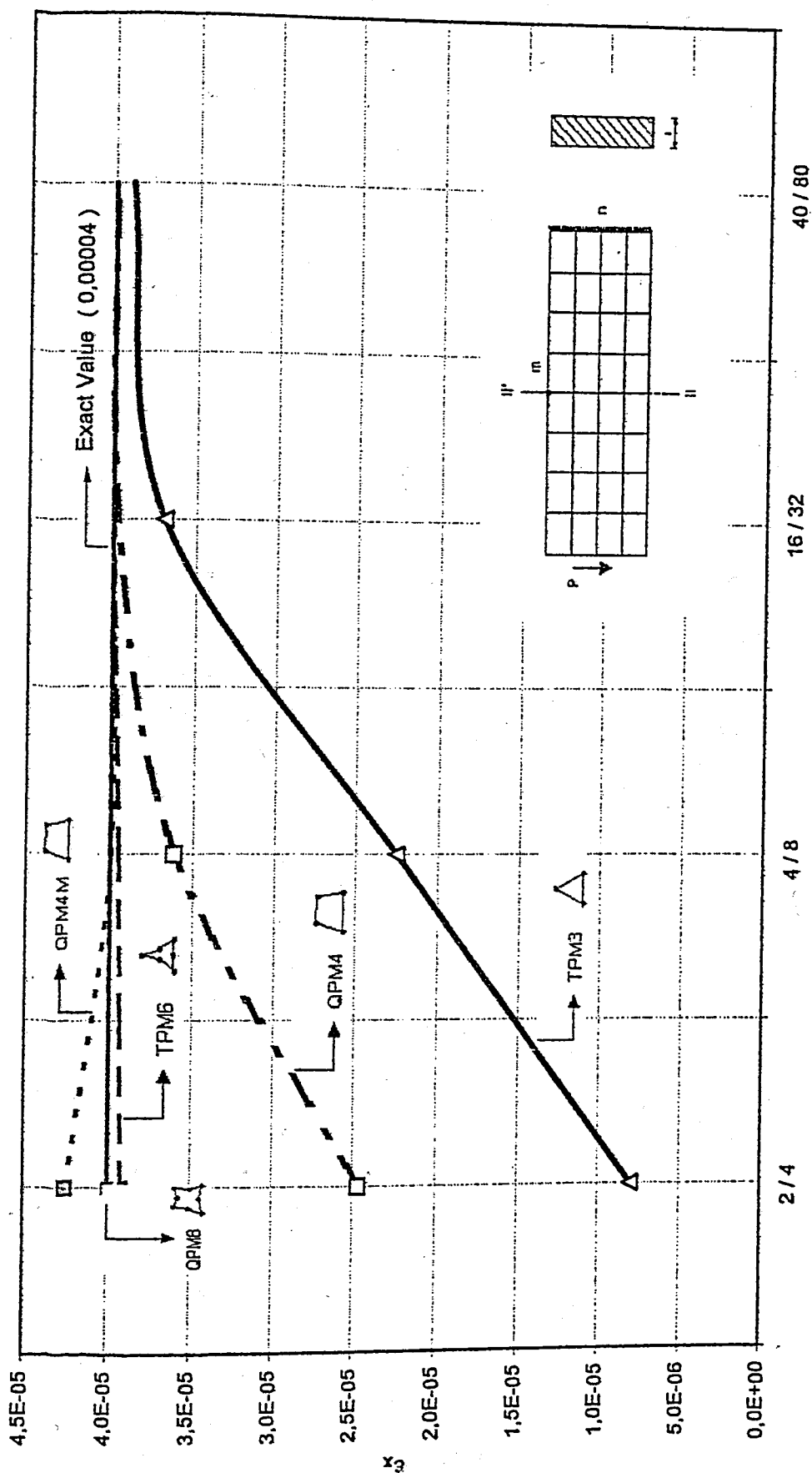


Figure B.33. Comparison of maximum shear stress (τ_{xy}) at section III-III' (40 / 80 mesh size)

Figure B.34. Maximum strain (ϵ_x) at section II-II'

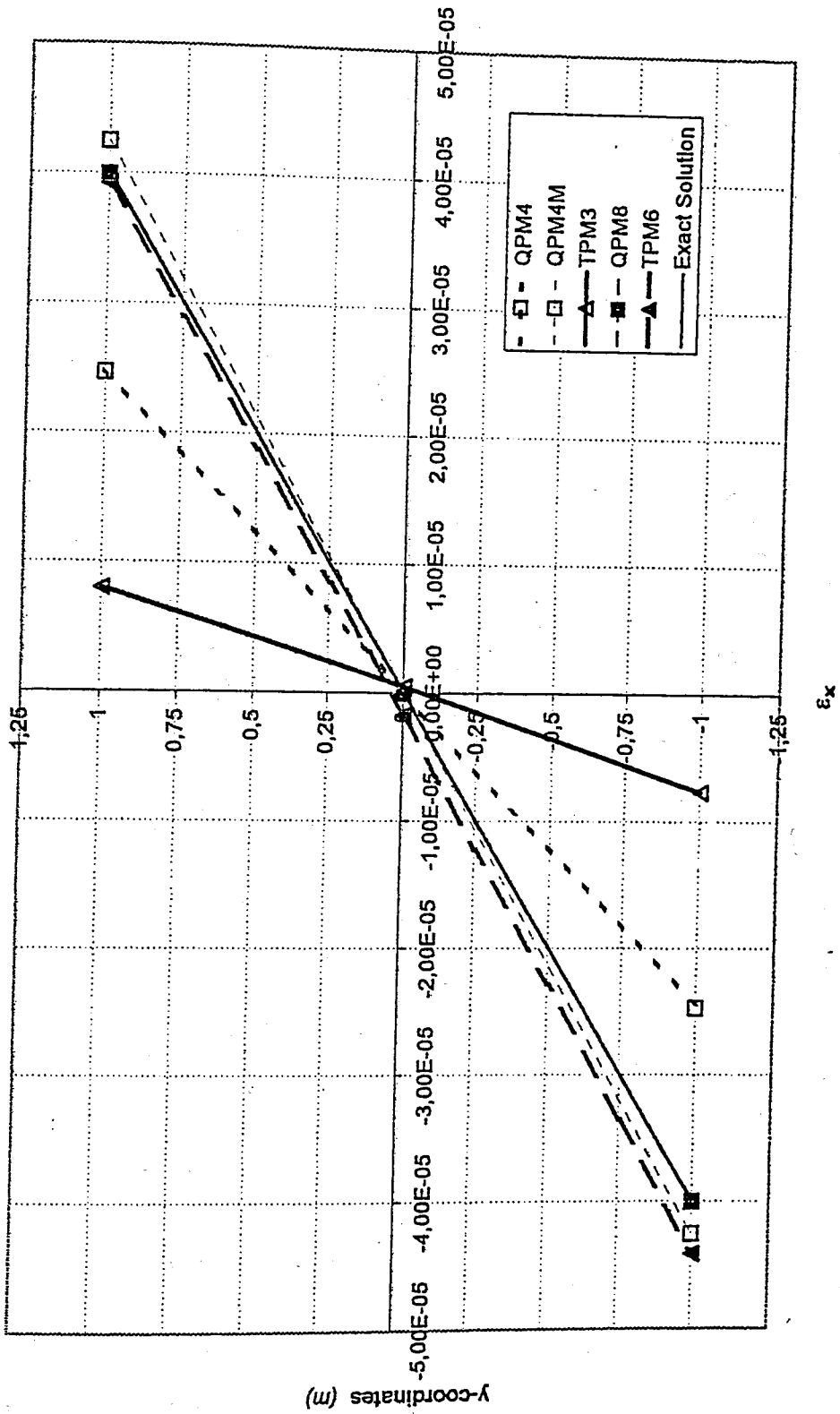
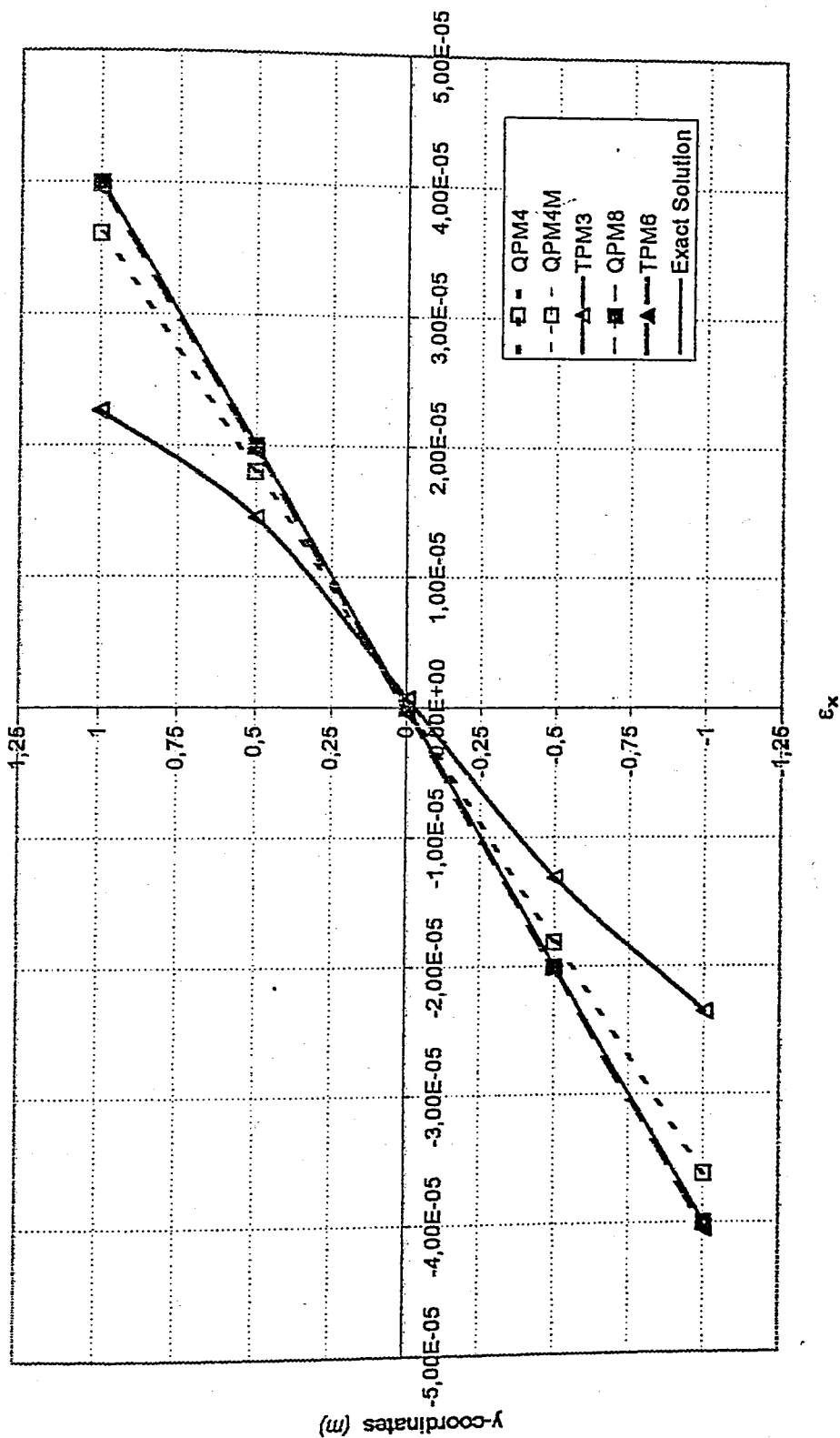


Figure B.35. Comparison of maximum strain (ϵ_x) at section II-II' (2 / 4 mesh size)

Figure B.36. Comparison of maximum strain (ϵ_x) at section II-II' (4 / 8 mesh size)

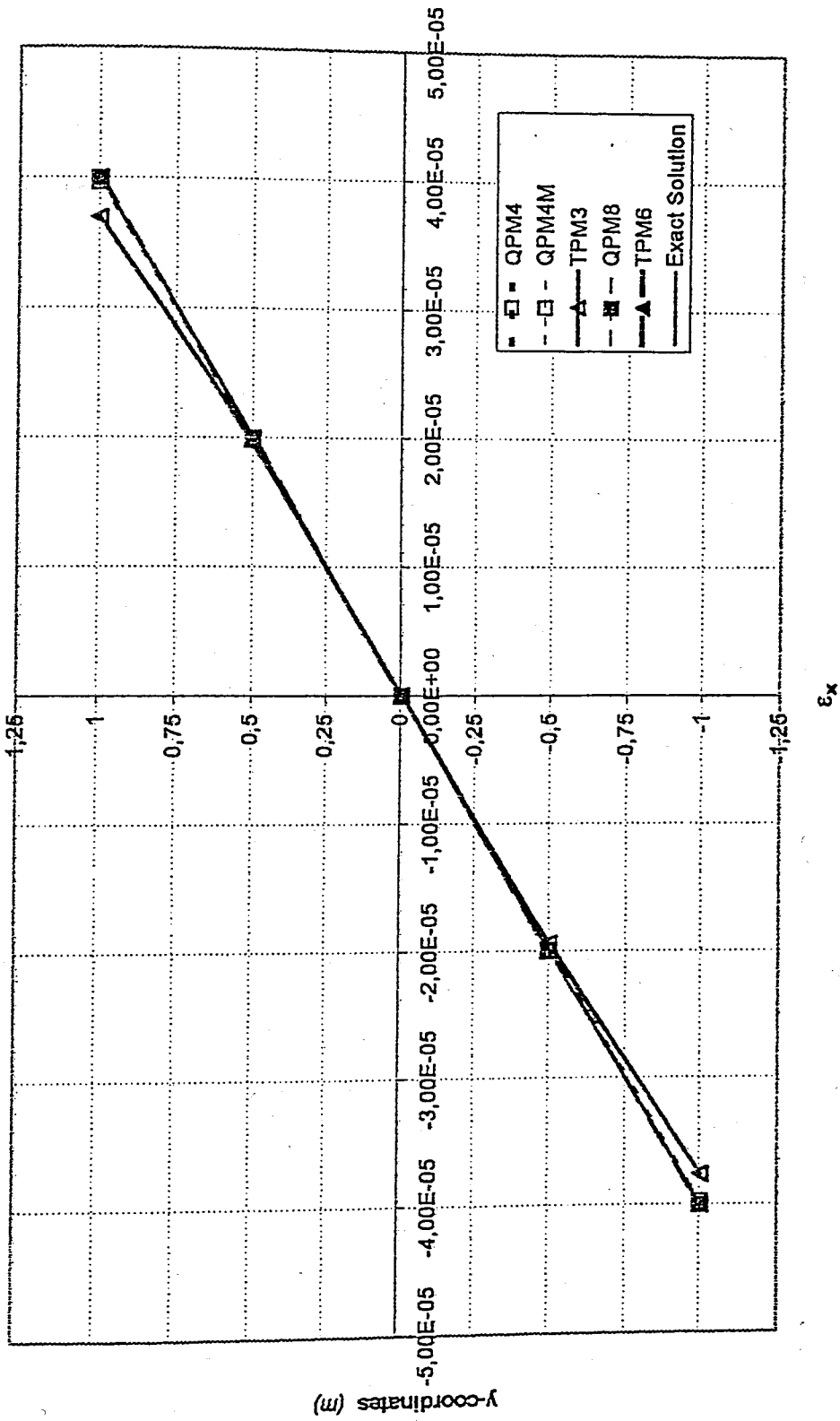


Figure B.37. Comparison of maximum strain (ϵ_x) at section II-II' (16/32 mesh size)

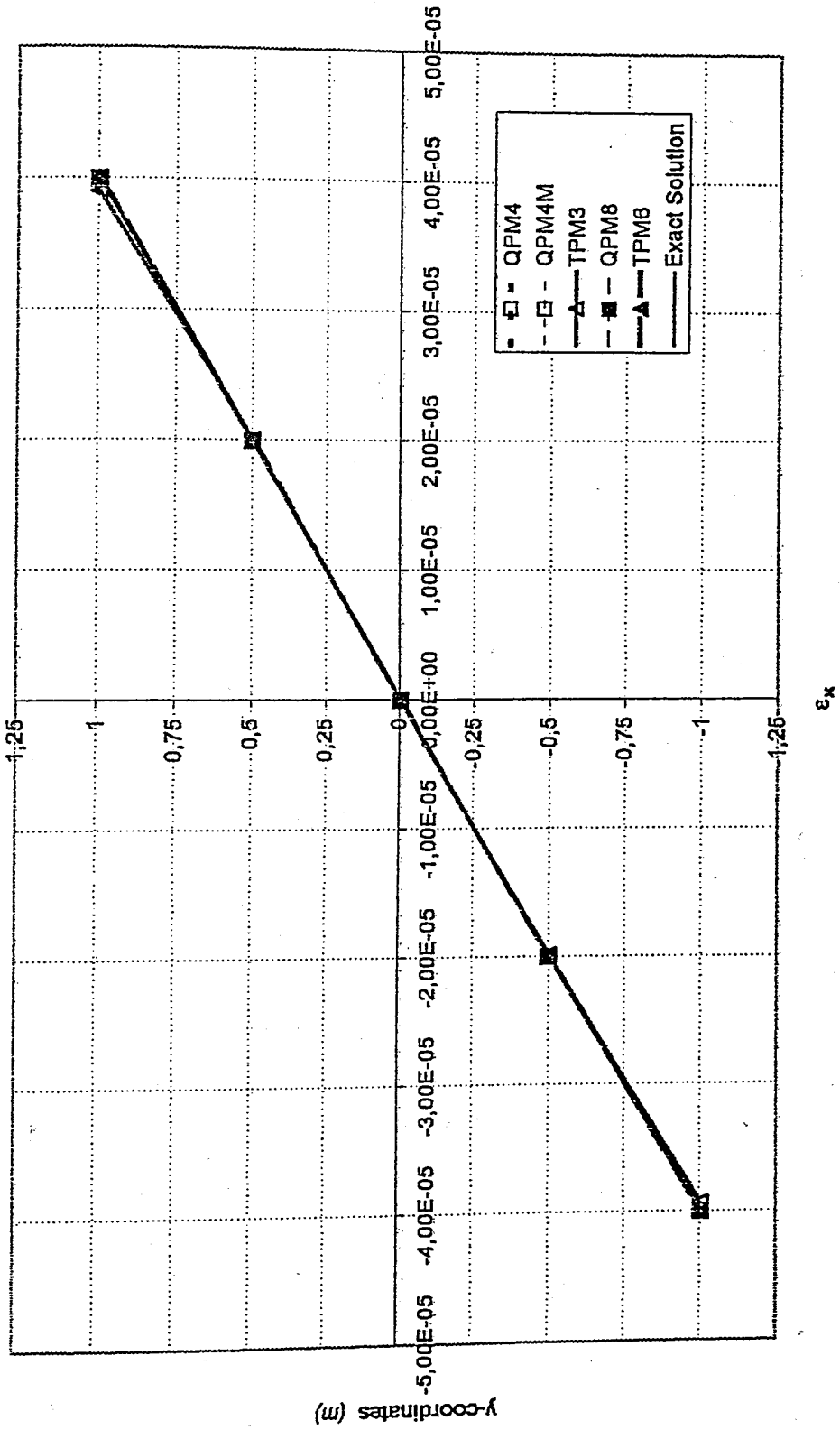


Figure B.38. Comparison of maximum strain (ϵ_x) at section II-II' (40 / 80 mesh size)

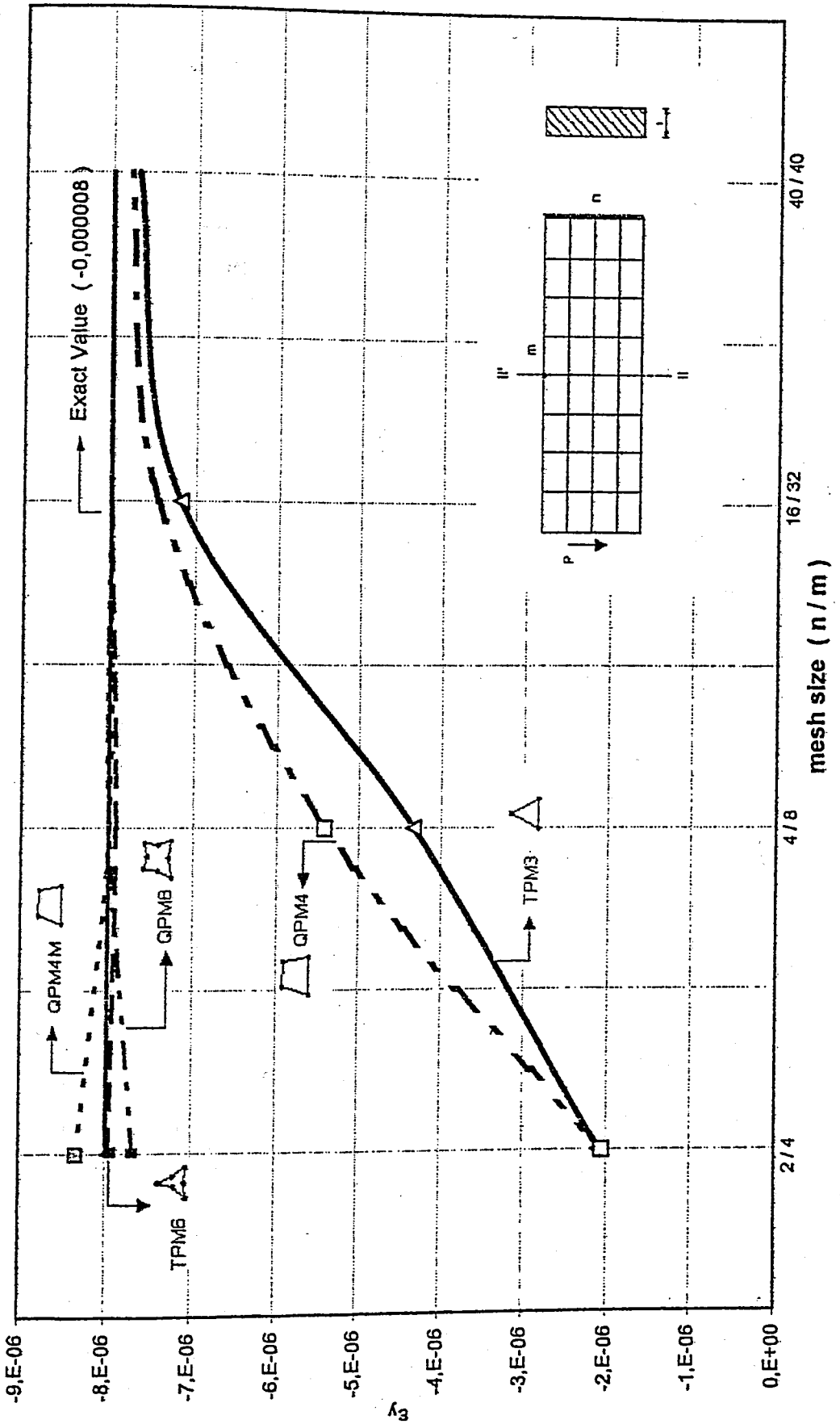


Figure B.39. Minimum strain (ϵ_y) at section II-II'

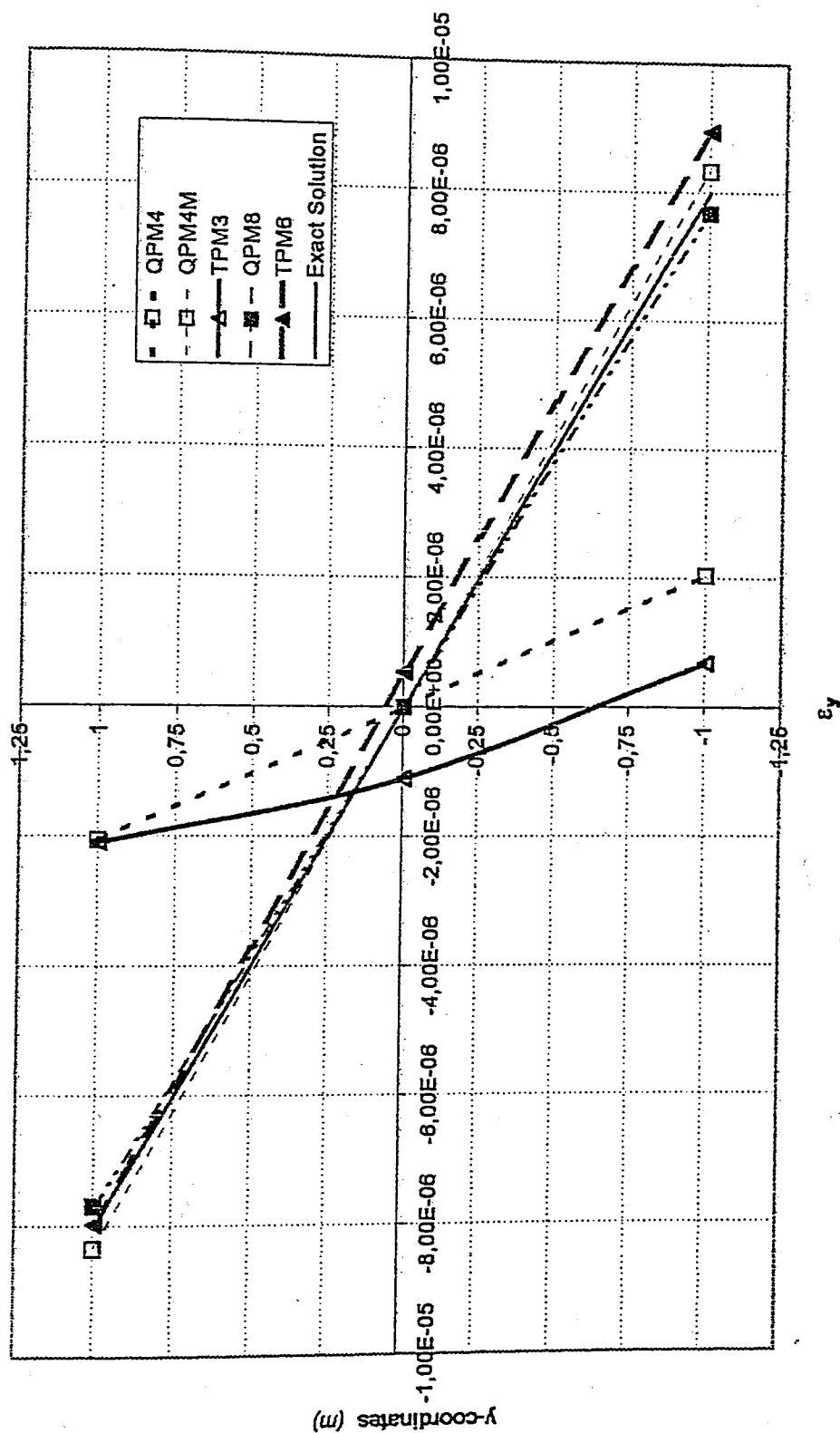


Figure B.40. Comparison of maximum strain (ϵ_y) at section II-II' (2/4 mesh size)

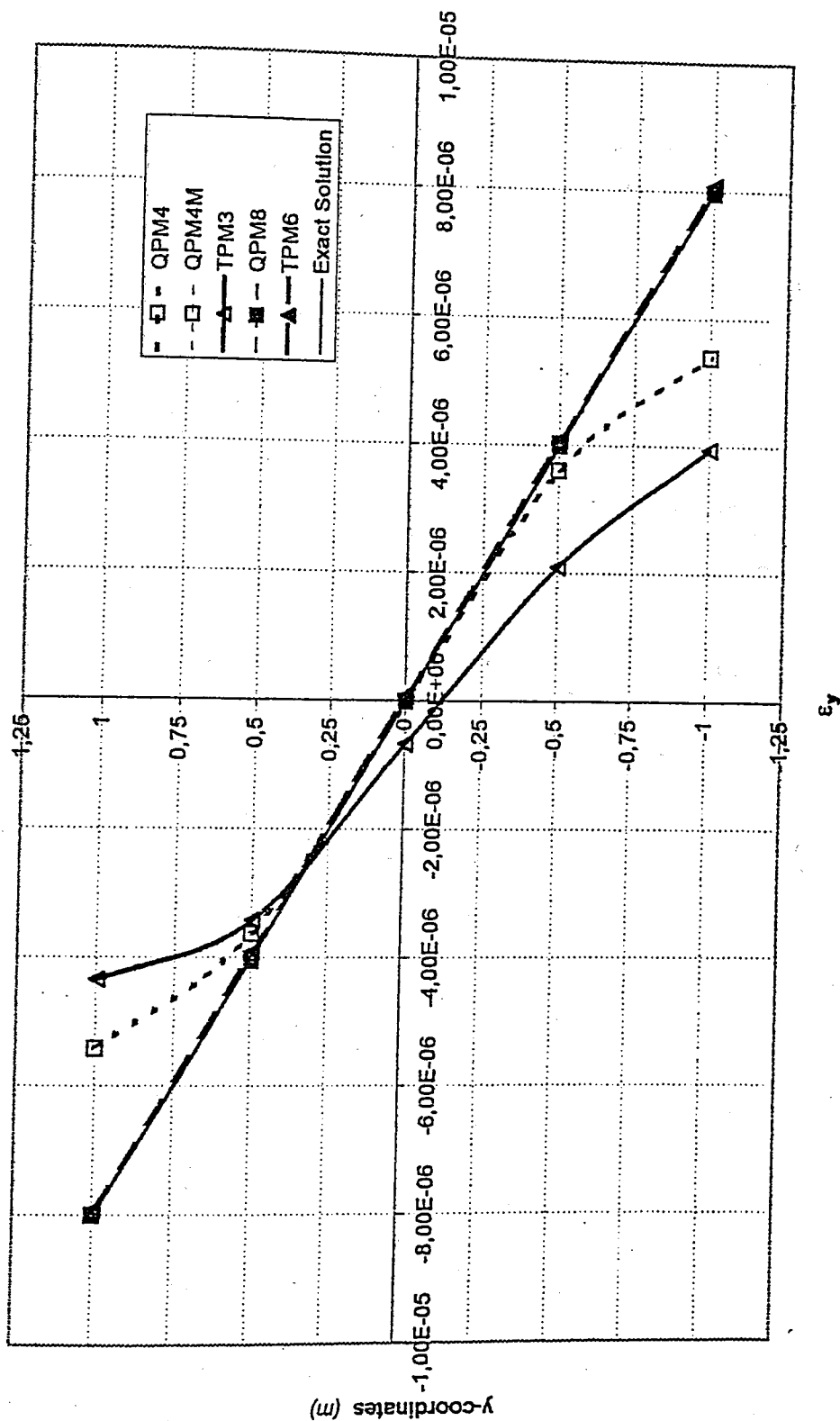


Figure B.41. Comparison of maximum strain (ϵ_y) at section II-II' (4 / 8 mesh size)

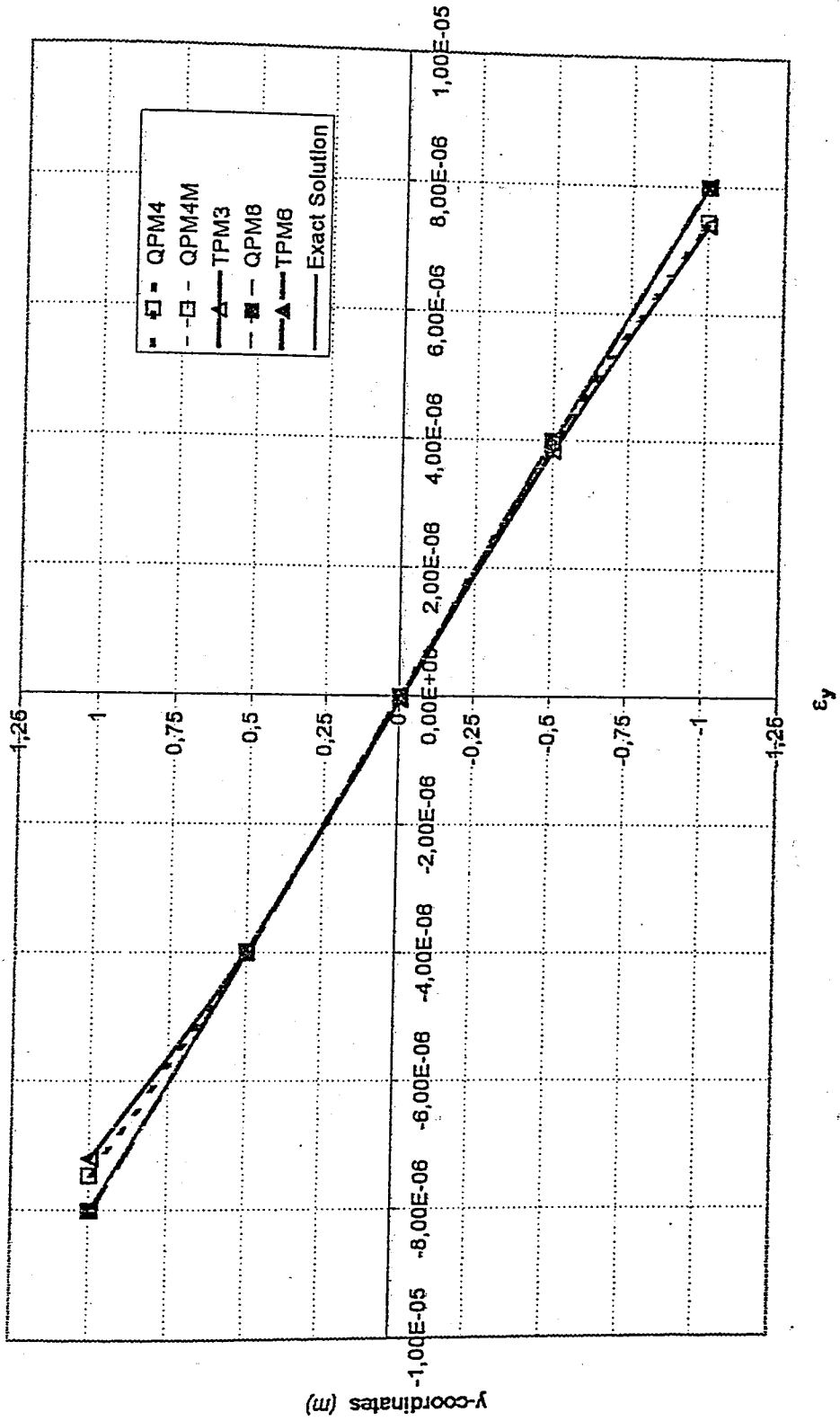


Figure B.42. Comparison of maximum strain (ϵ_y) at section II-II' (16 / 32 mesh size)

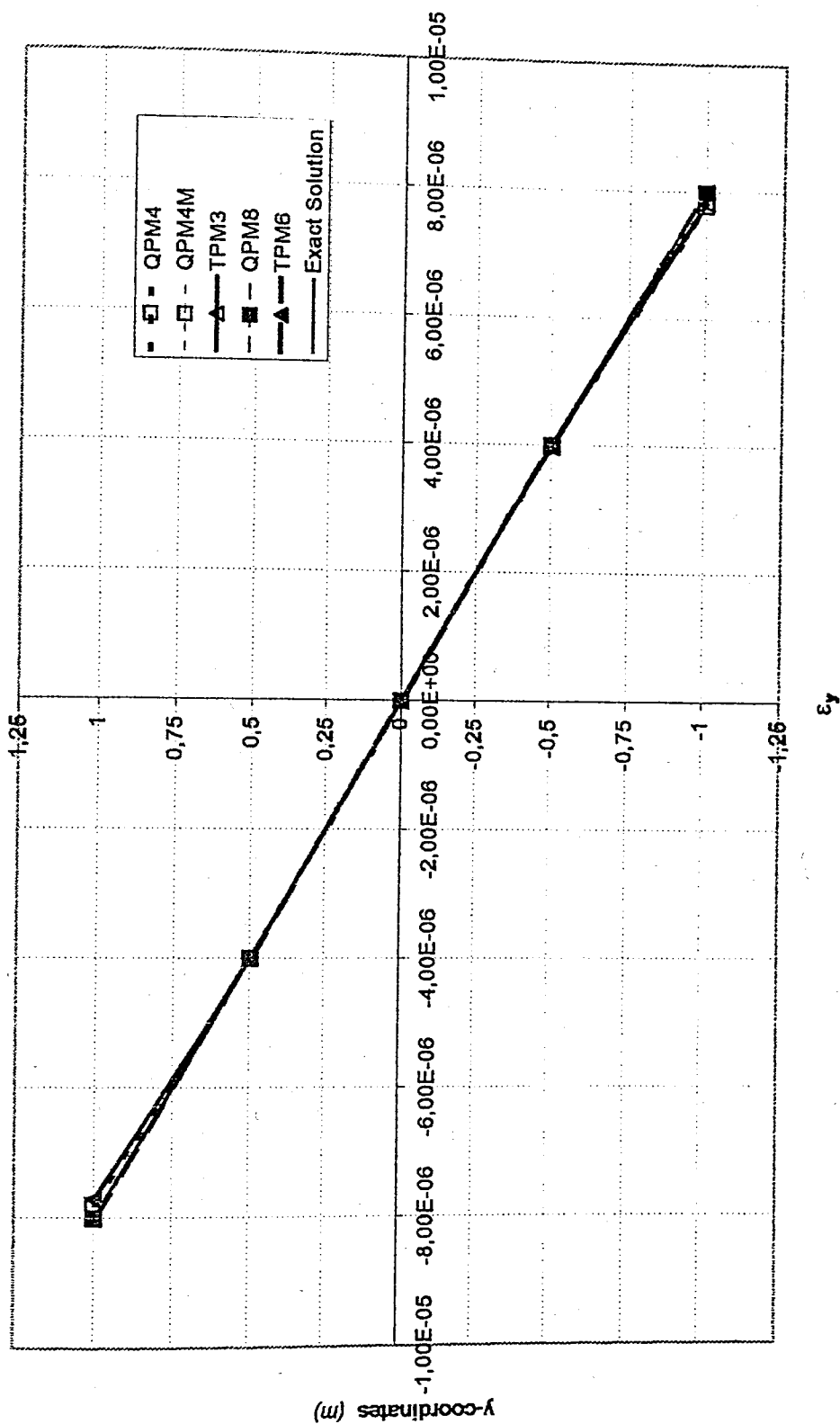


Figure B.43. Comparison of maximum strain (ϵ_y) at section II-II' (40 / 80 mesh size)

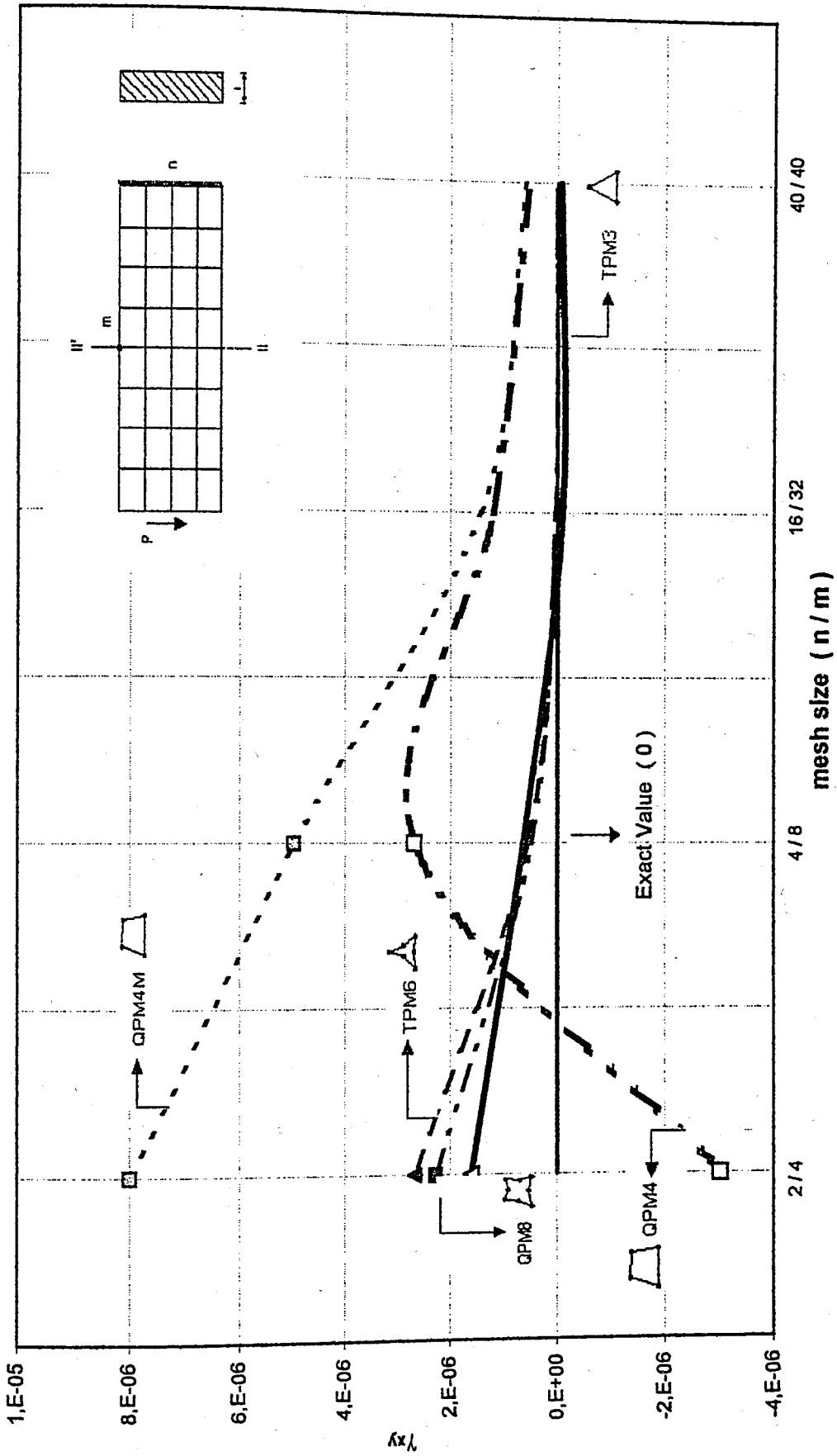


Figure B.44. Shear strain (γ_{xy}) at section II-II'

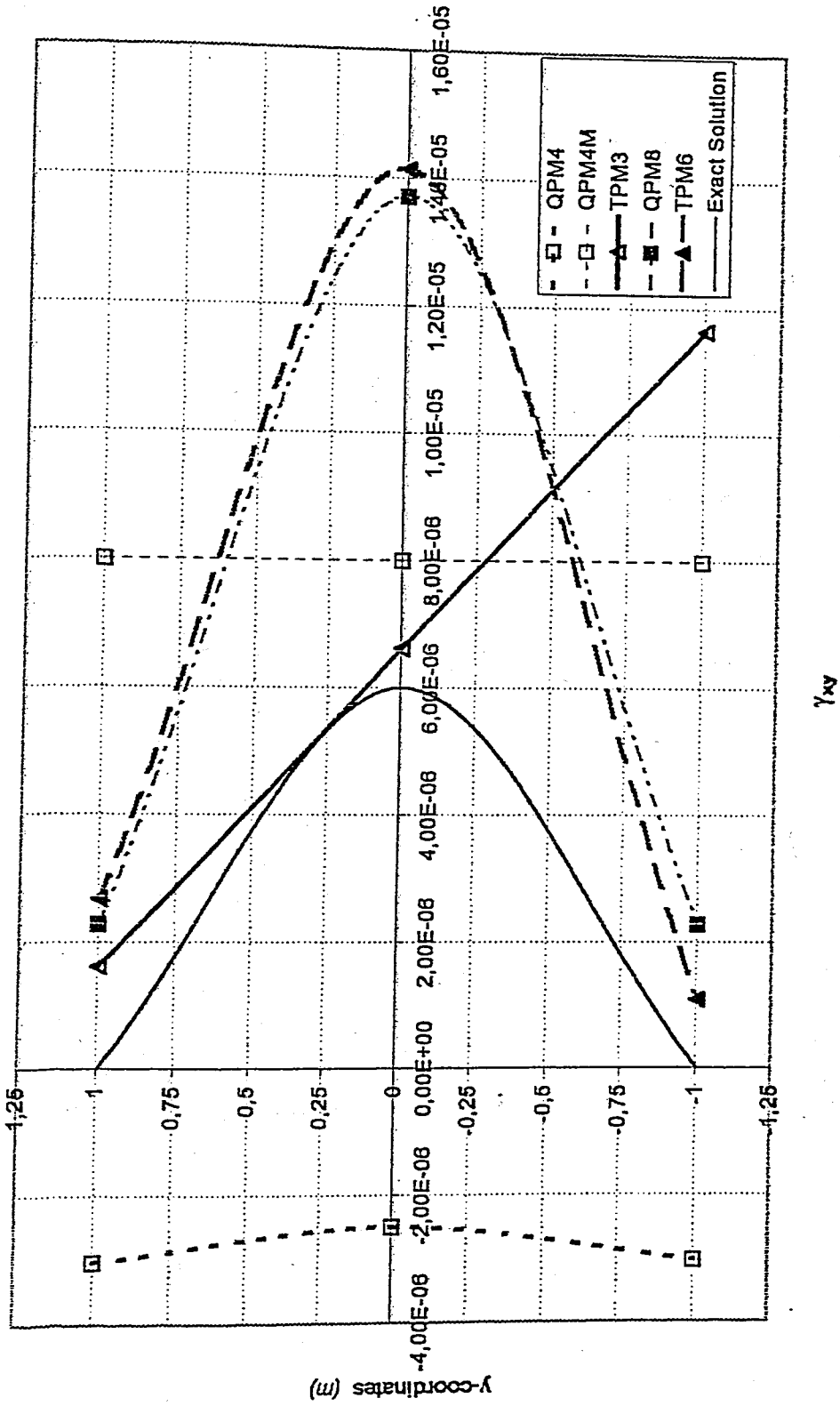


Figure B.45. Comparison of shear strain (γ_{xy}) at section II-II' (2/4 mesh size)

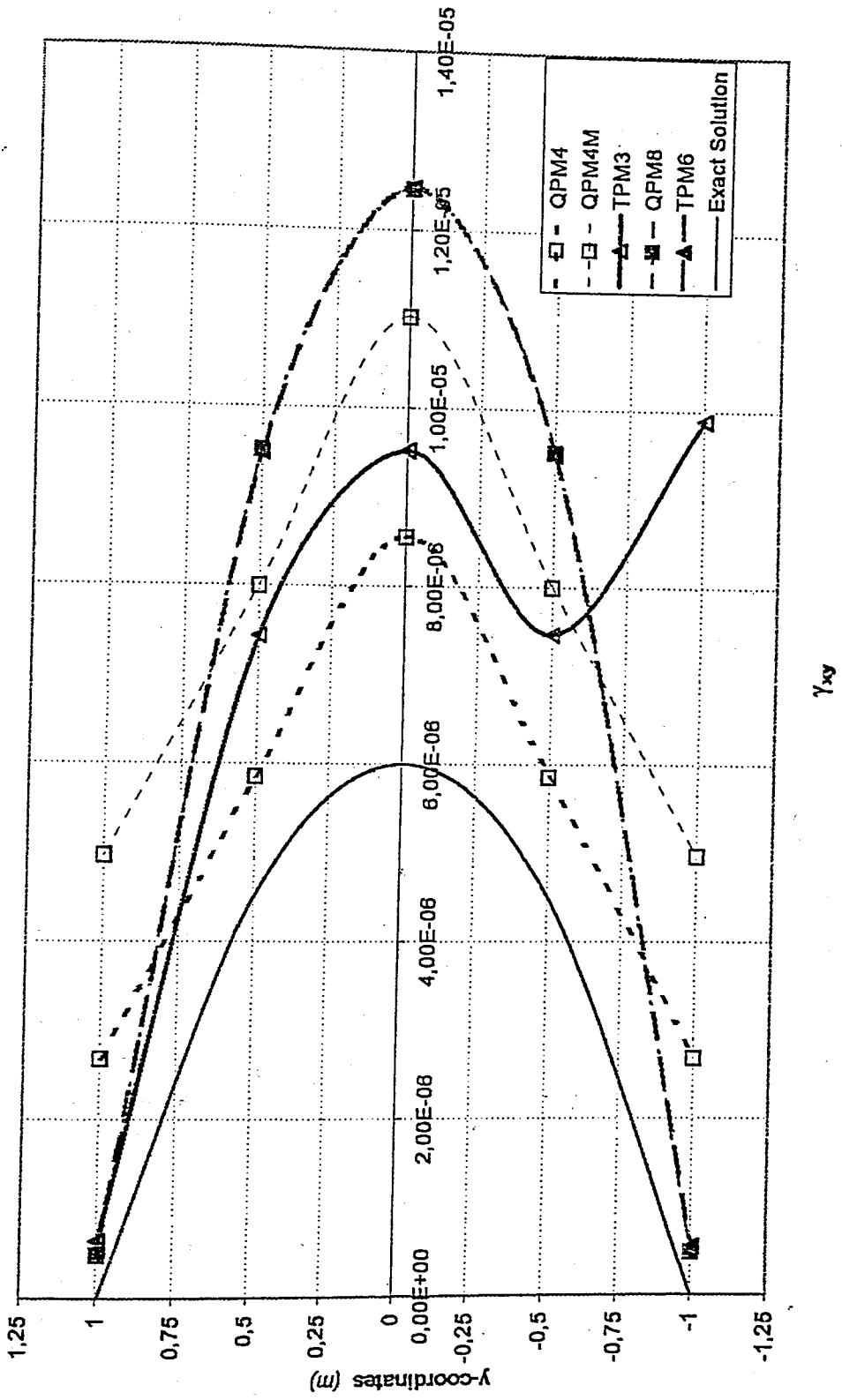


Figure B.46. Comparison of shear strain (γ_{xy}) at section II-II' (4 / 8 mesh size)

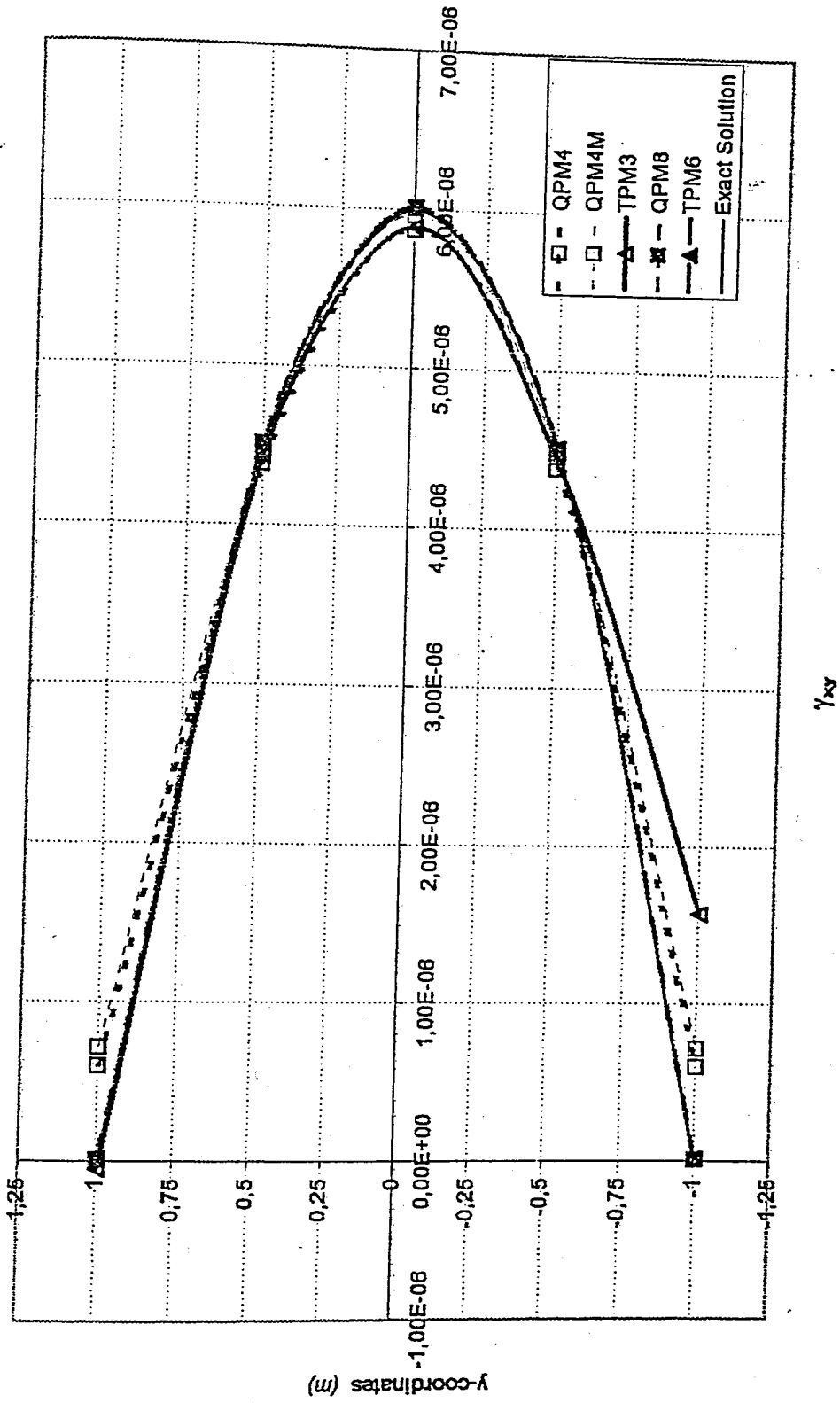


Figure B.47. Comparison of Shear Strain (γ_{xy}) at section II-II' (16 / 32 mesh size)

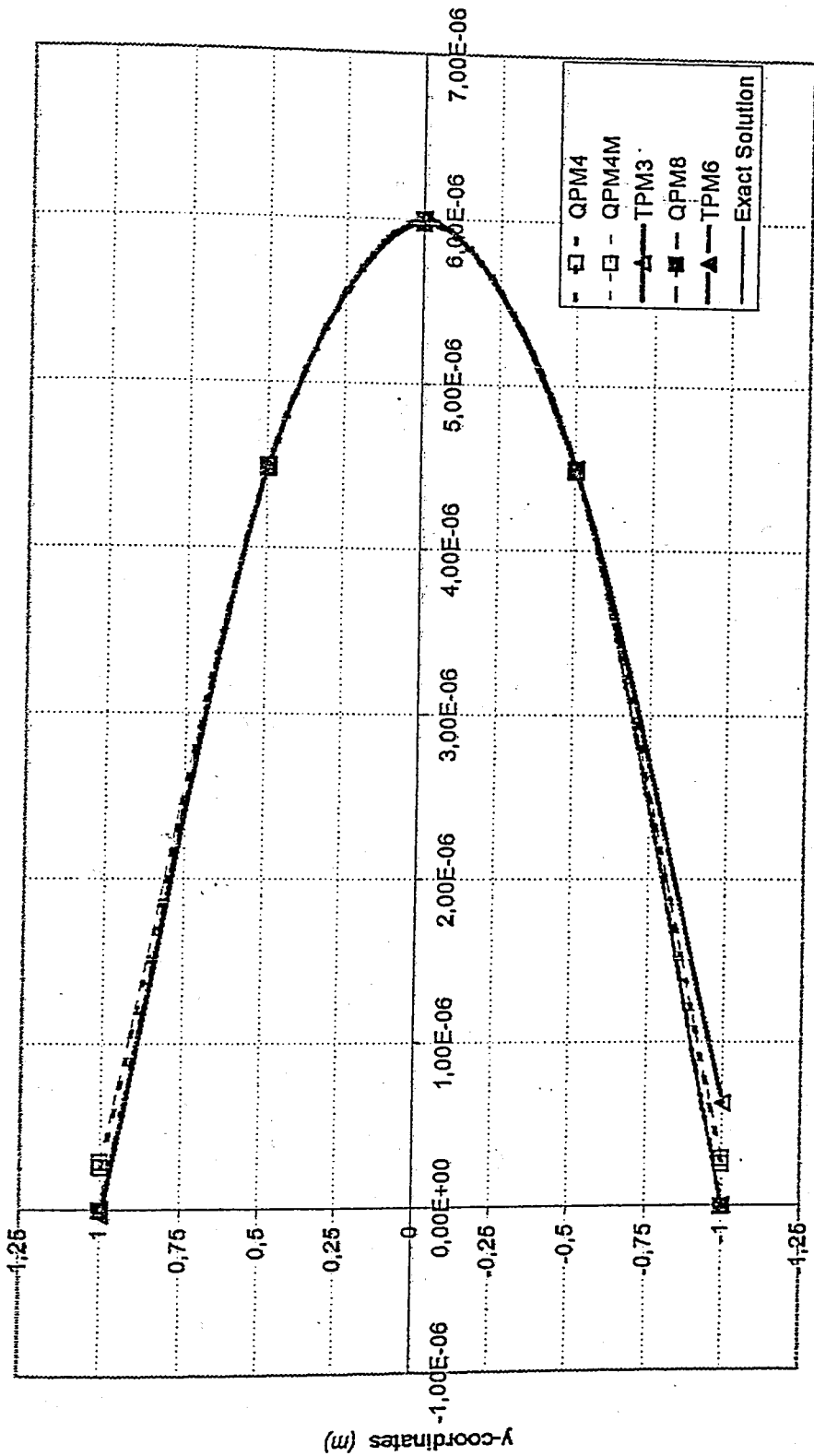
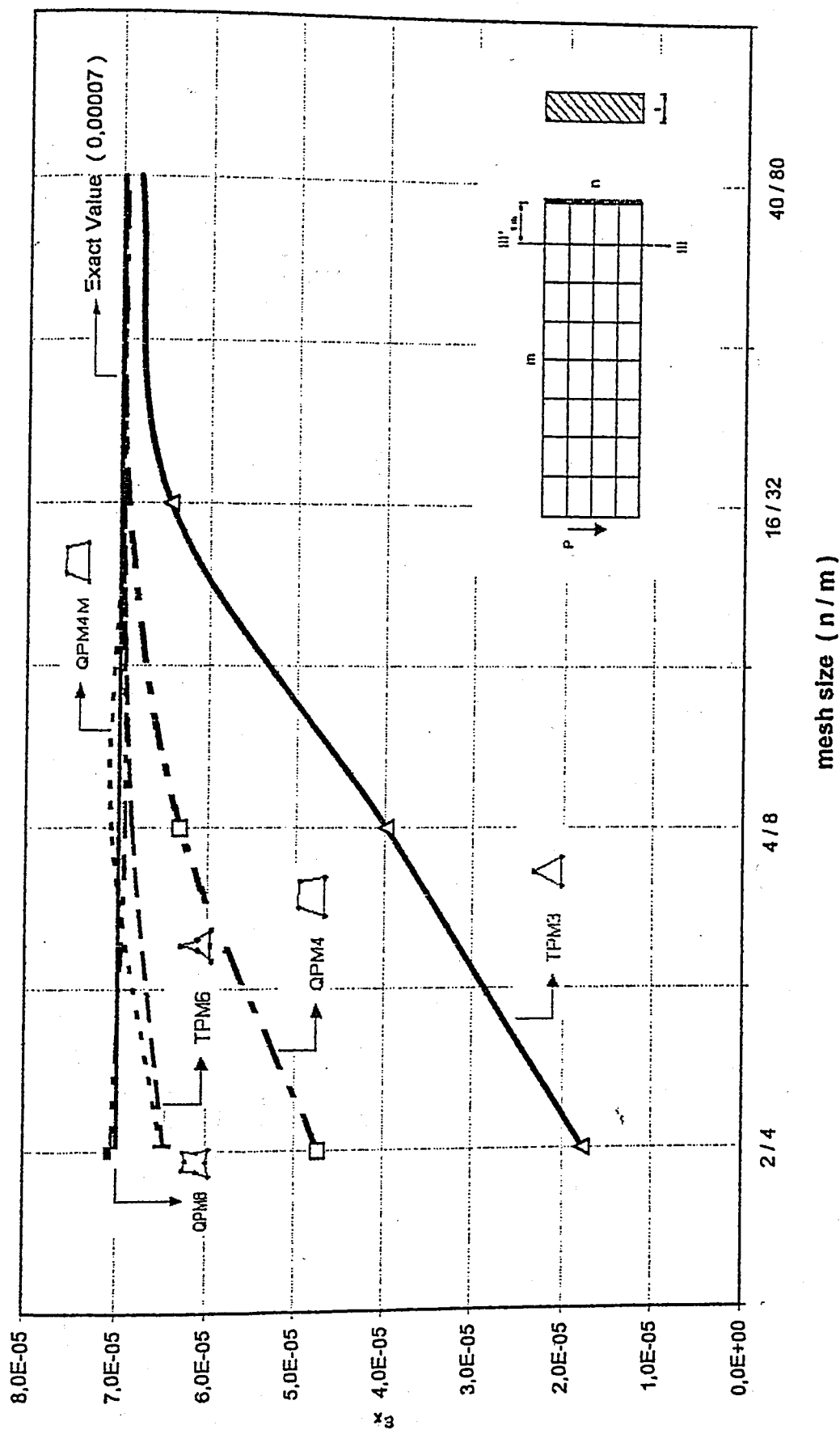


Figure B.48. Comparison of shear strain (γ_{xy}) at section II-II' (40 / 80 mesh size)

Figure B.49. Maximum strain (ϵ_x) at section III-III'

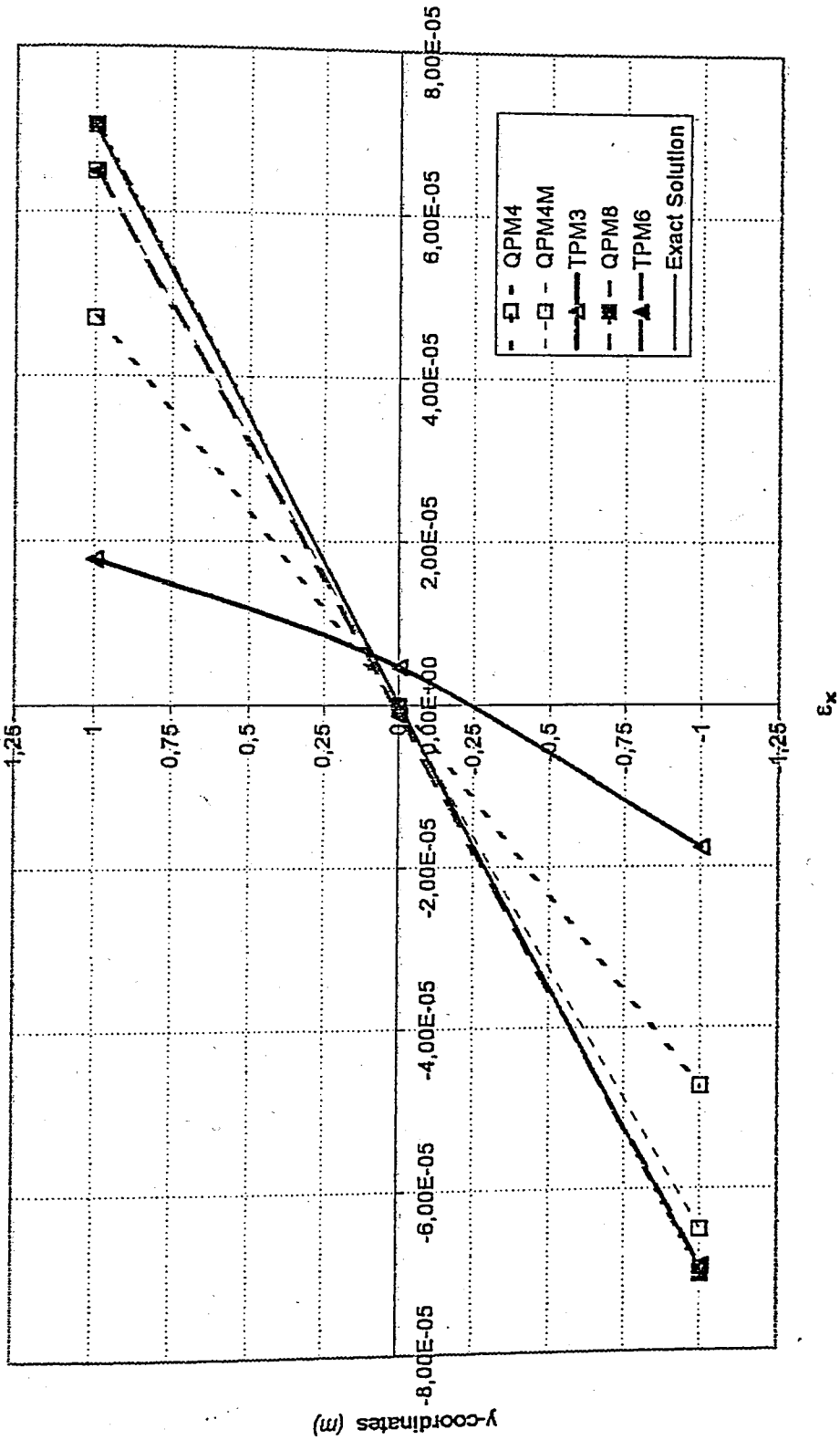


Figure B.50. Comparison of maximum strain (ϵ_x) at section III-III' (2 / 4 mesh size)

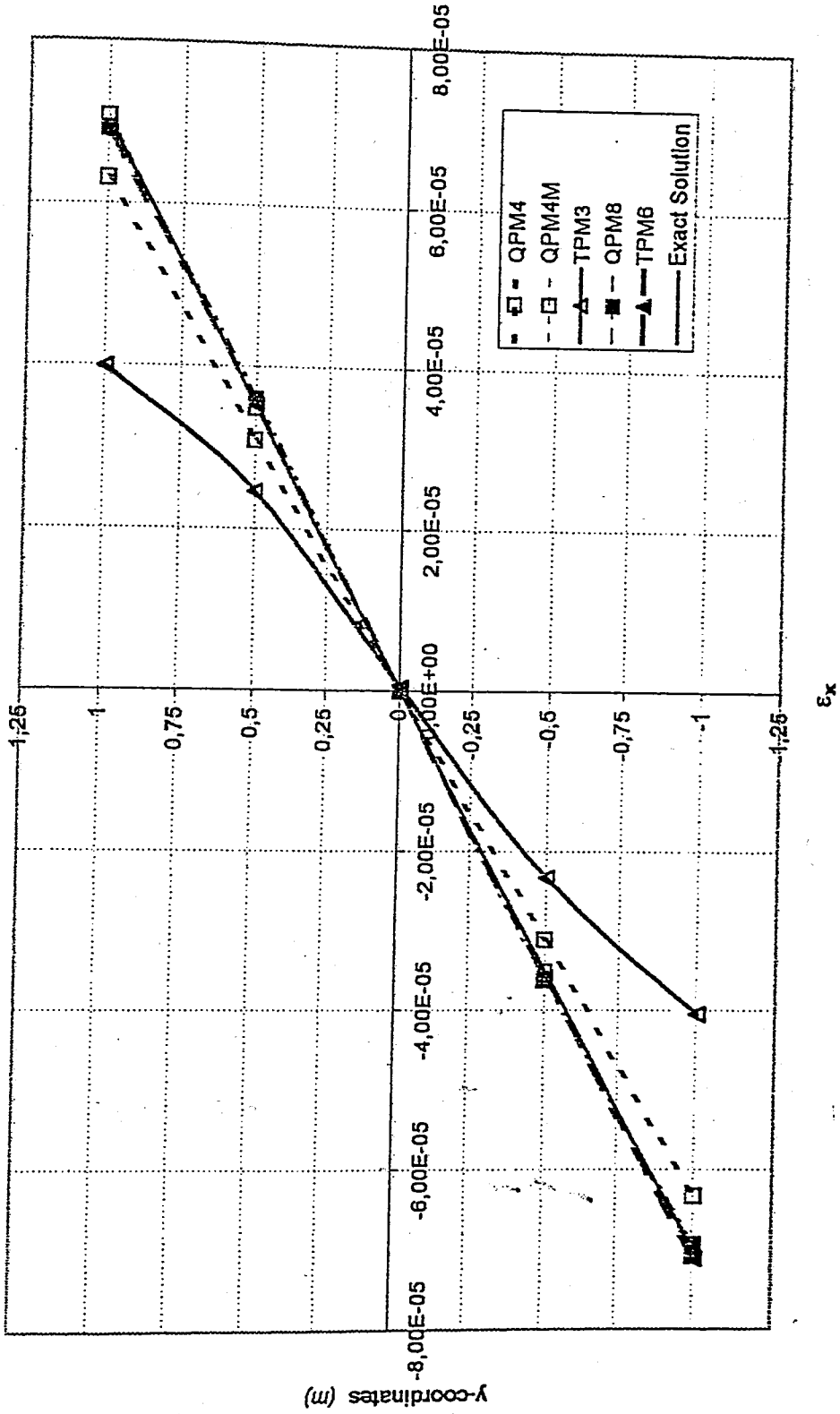


Figure B.51. Comparison of maximum strain (ϵ_x) at section III-III' (4 / 8 mesh size)

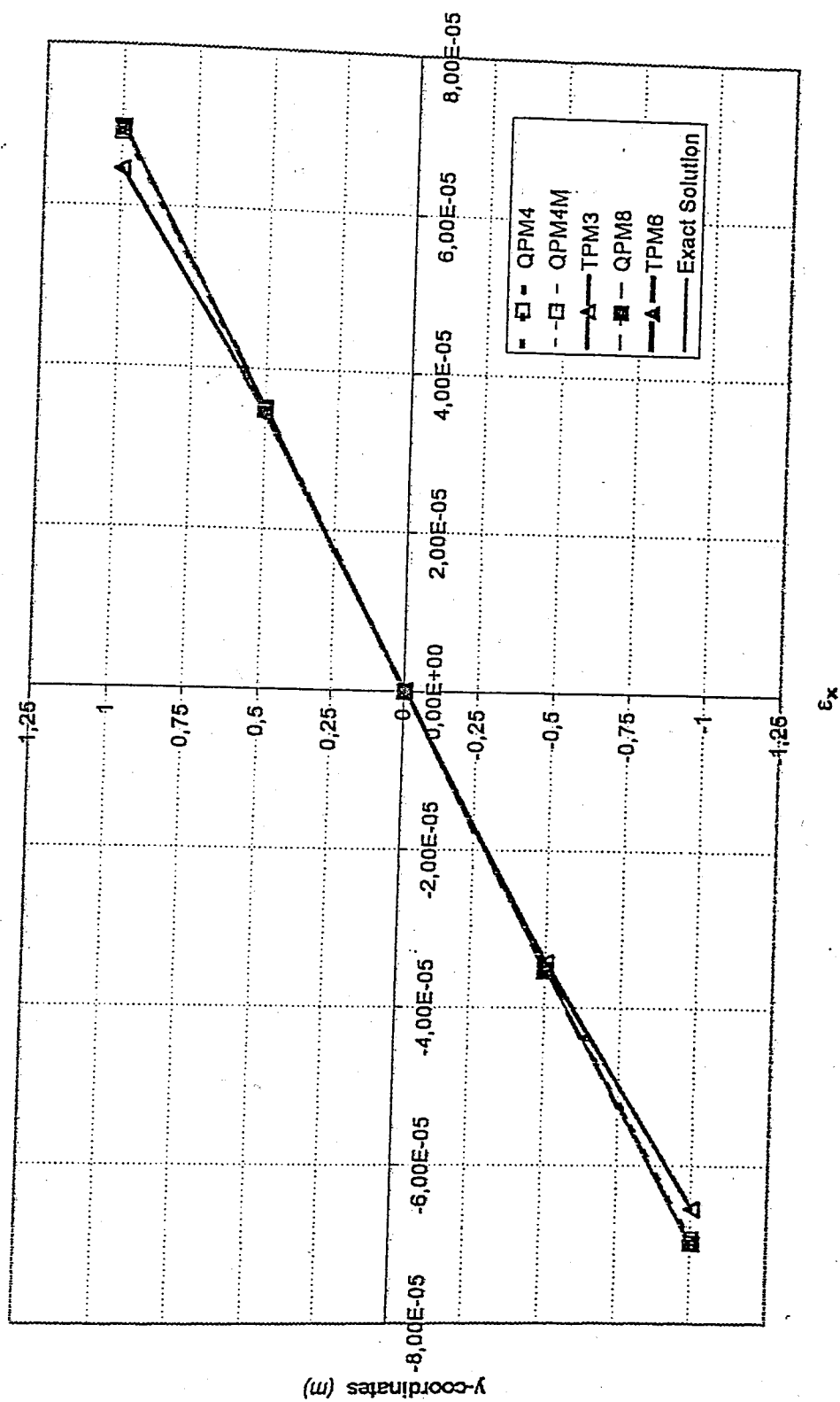


Figure B.52. Comparison of maximum strain (ϵ_x) at section III-III' (16 / 32 mesh size)

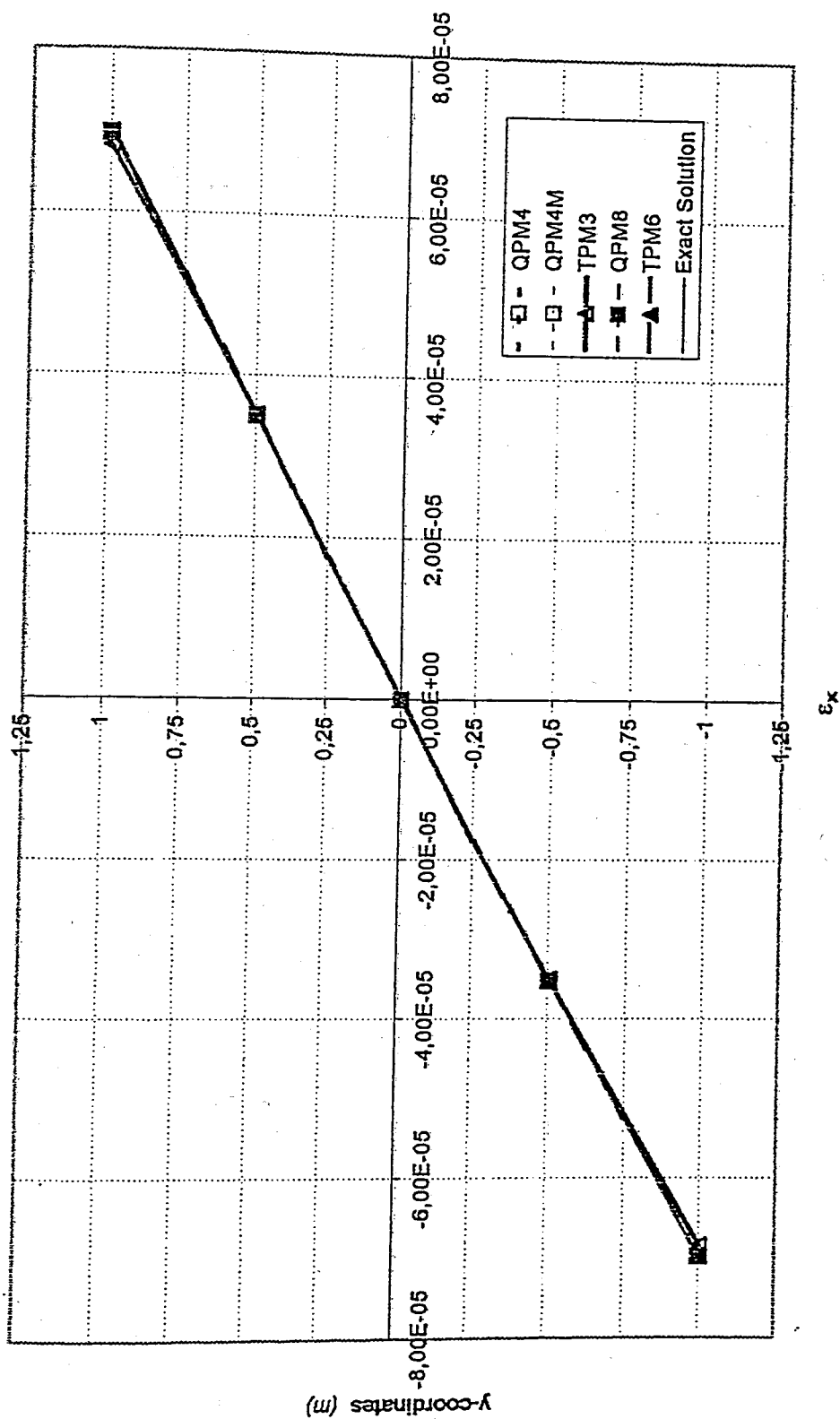
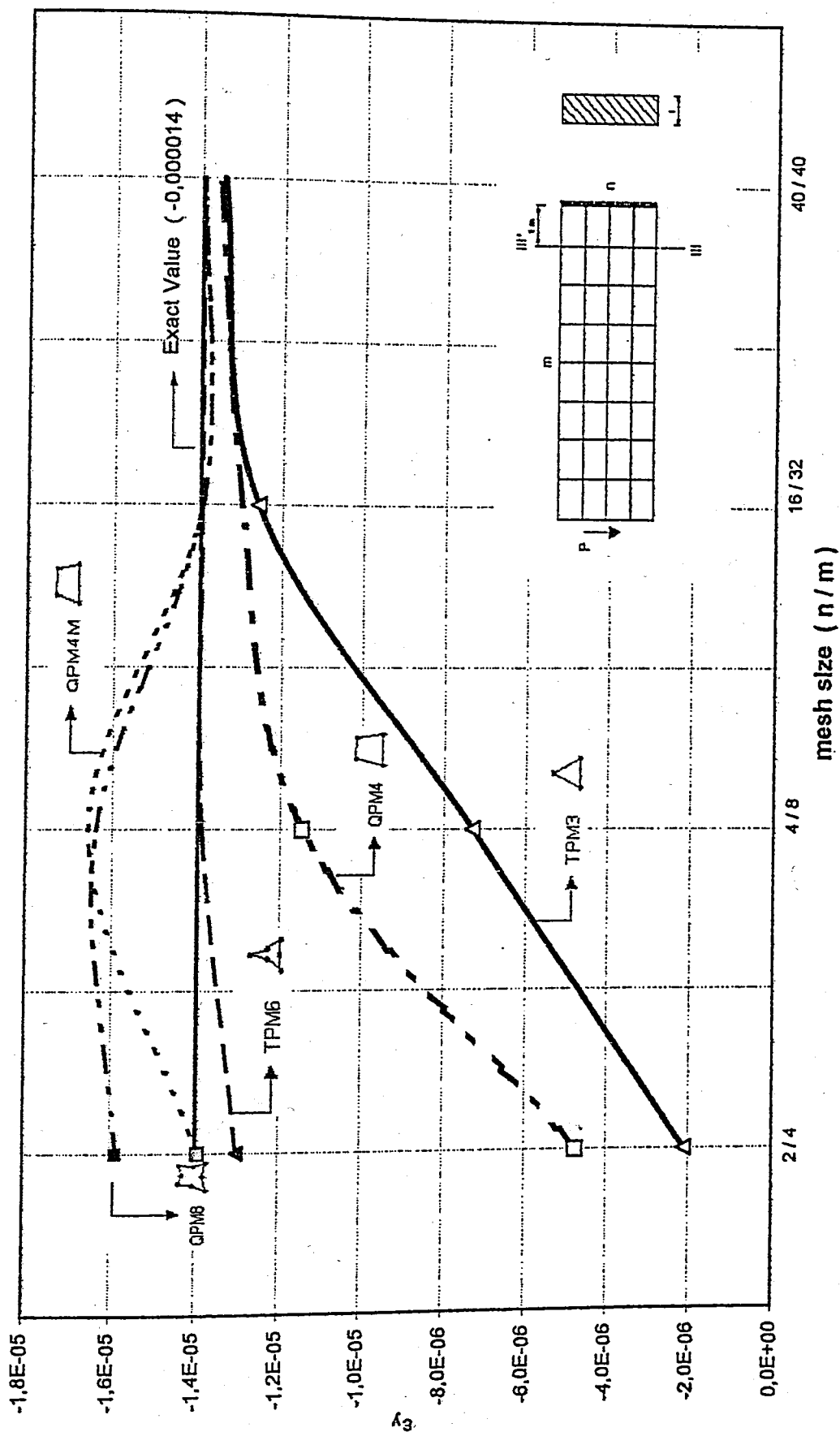
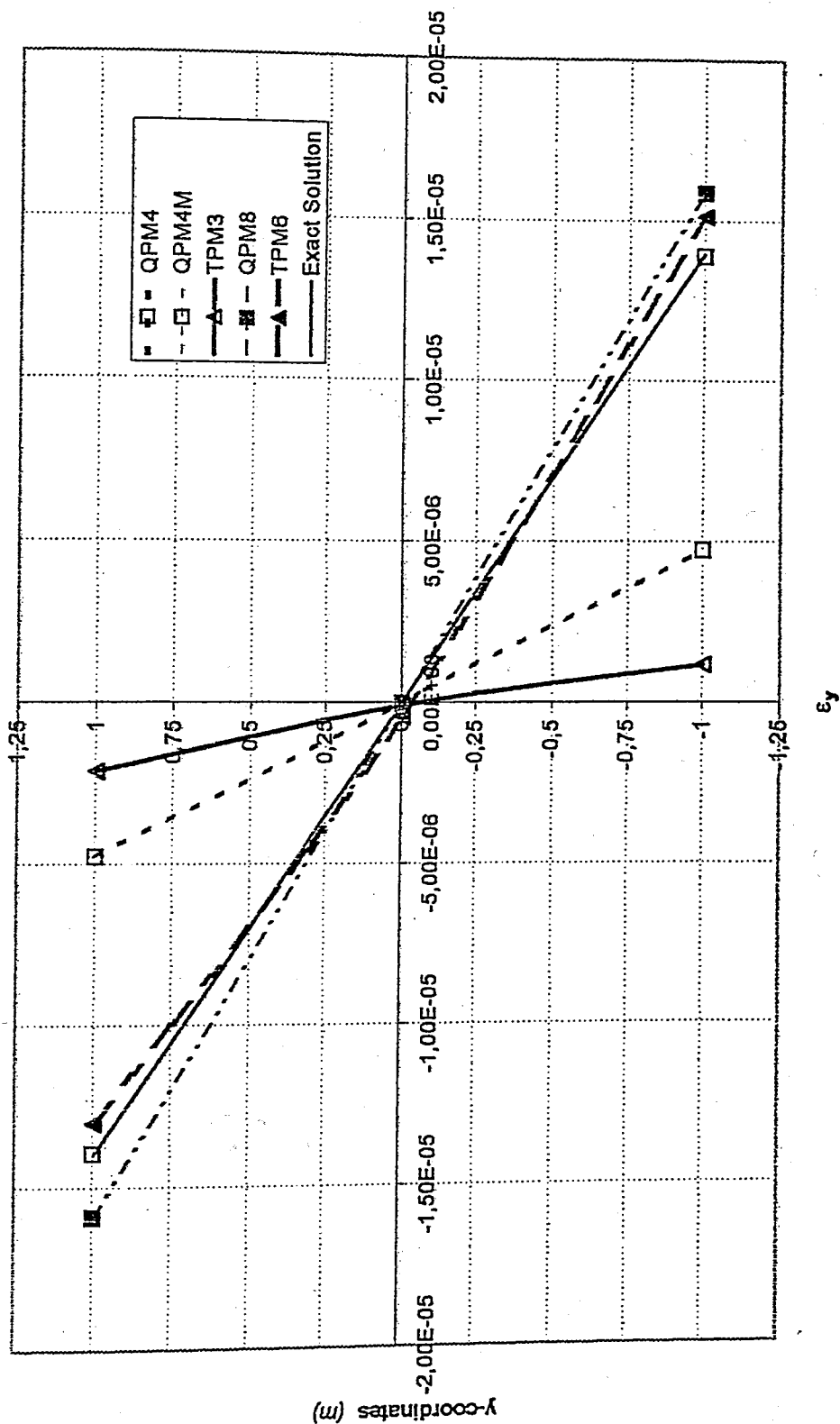


Figure B.53. Comparison of maximum strain (ϵ_x) at section III-III' (40 / 80 mesh size)

Figure B.54. Minimum strain (ϵ_y) at section III-III'

Figure B.55. Comparison of maximum strain (ϵ_y) at section III-III' (2 / 4 mesh size)

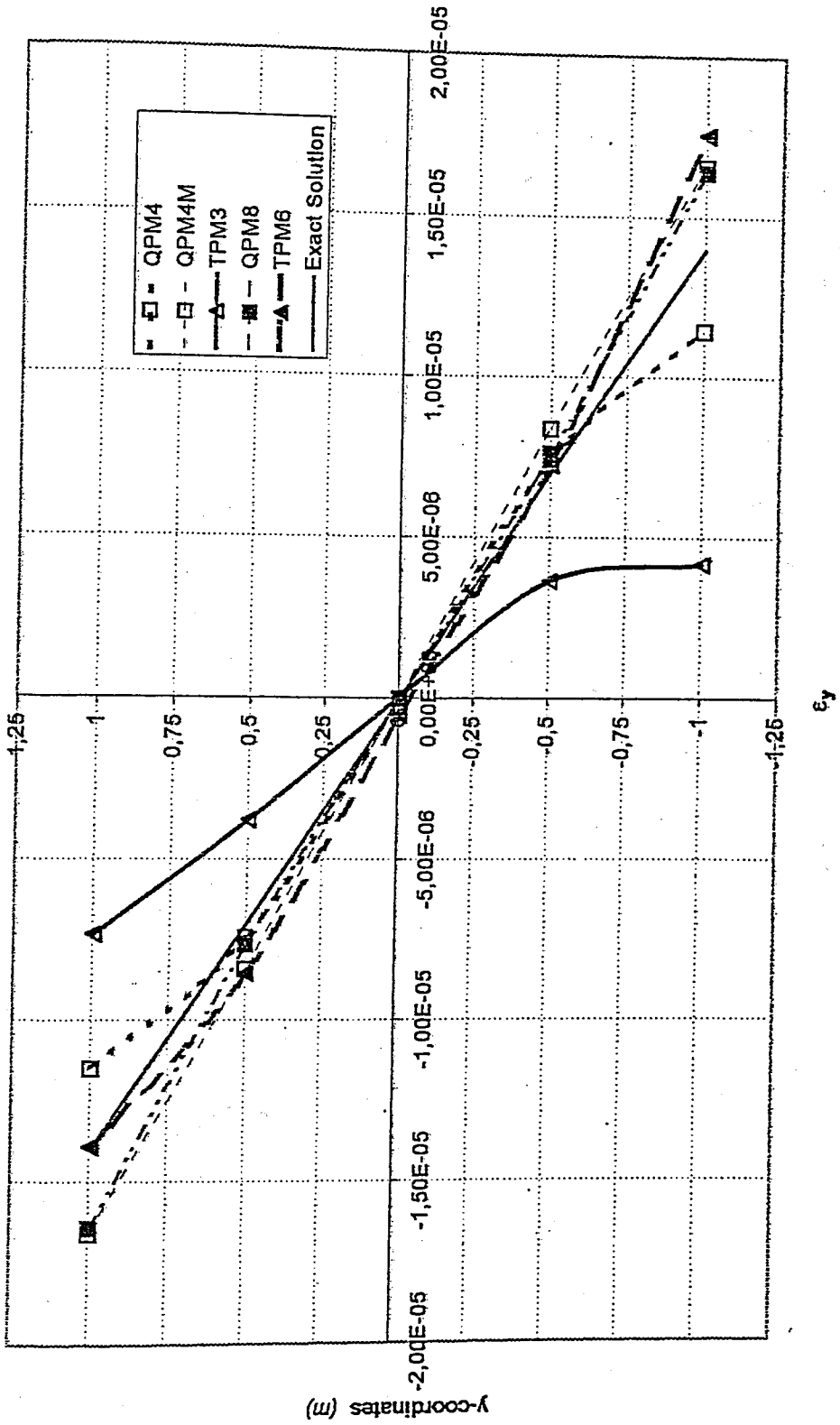


Figure B.56. Comparison of maximum strain (ϵ_y) at section III-III' (4/8 mesh size)

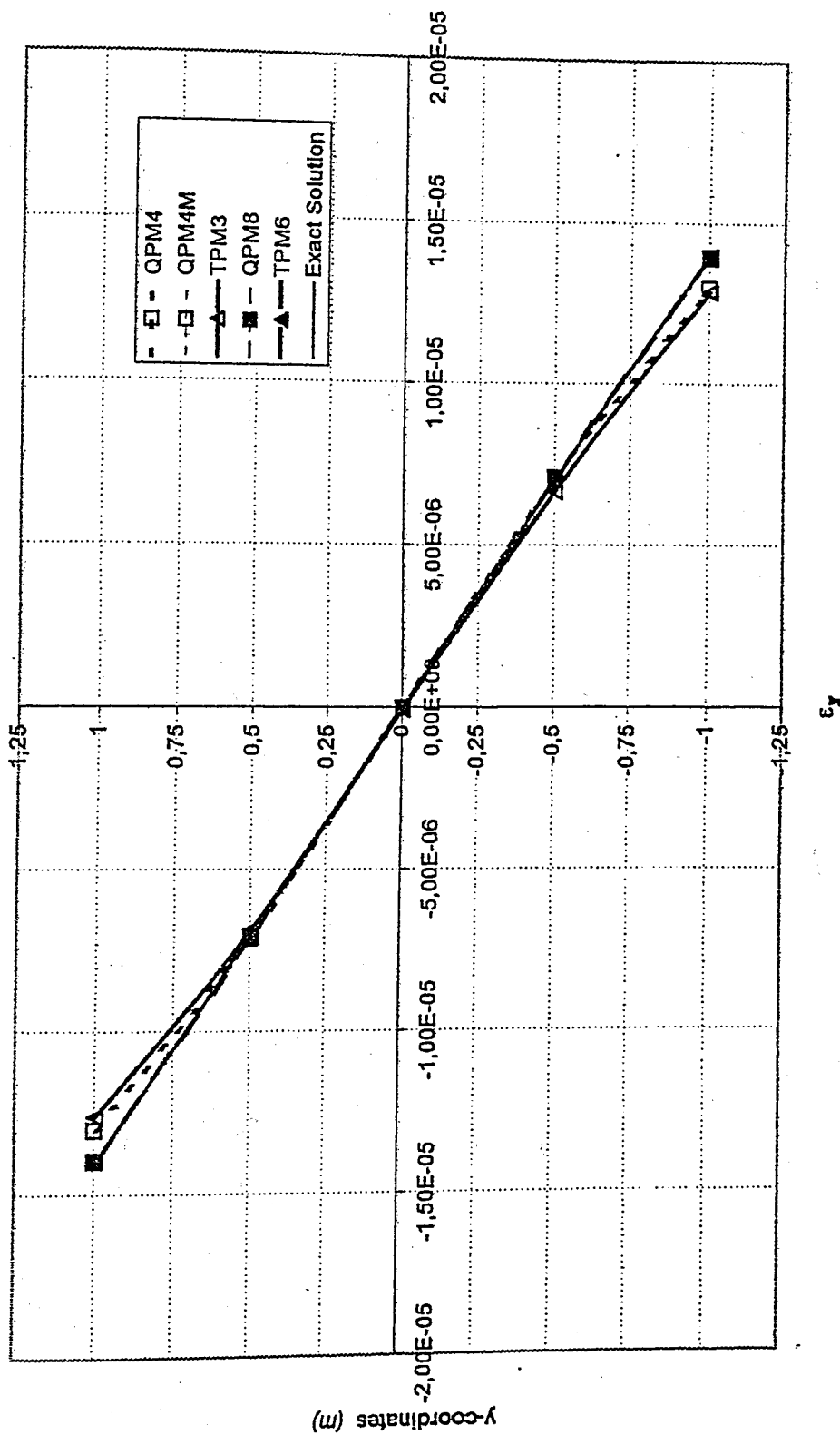


Figure B.57. Comparison of maximum strain (ϵ_y) at section III-III' (16 / 32 mesh size)

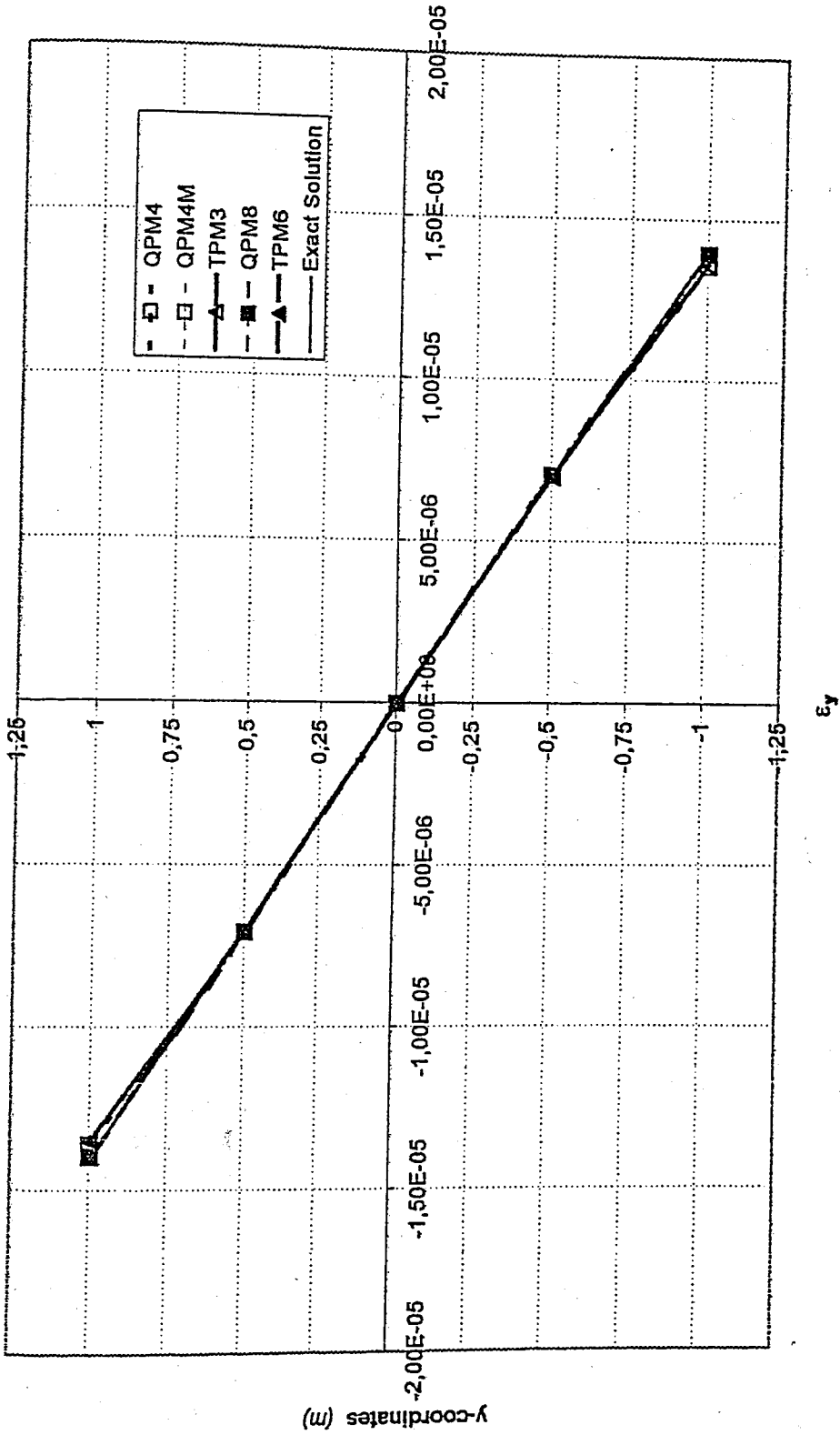
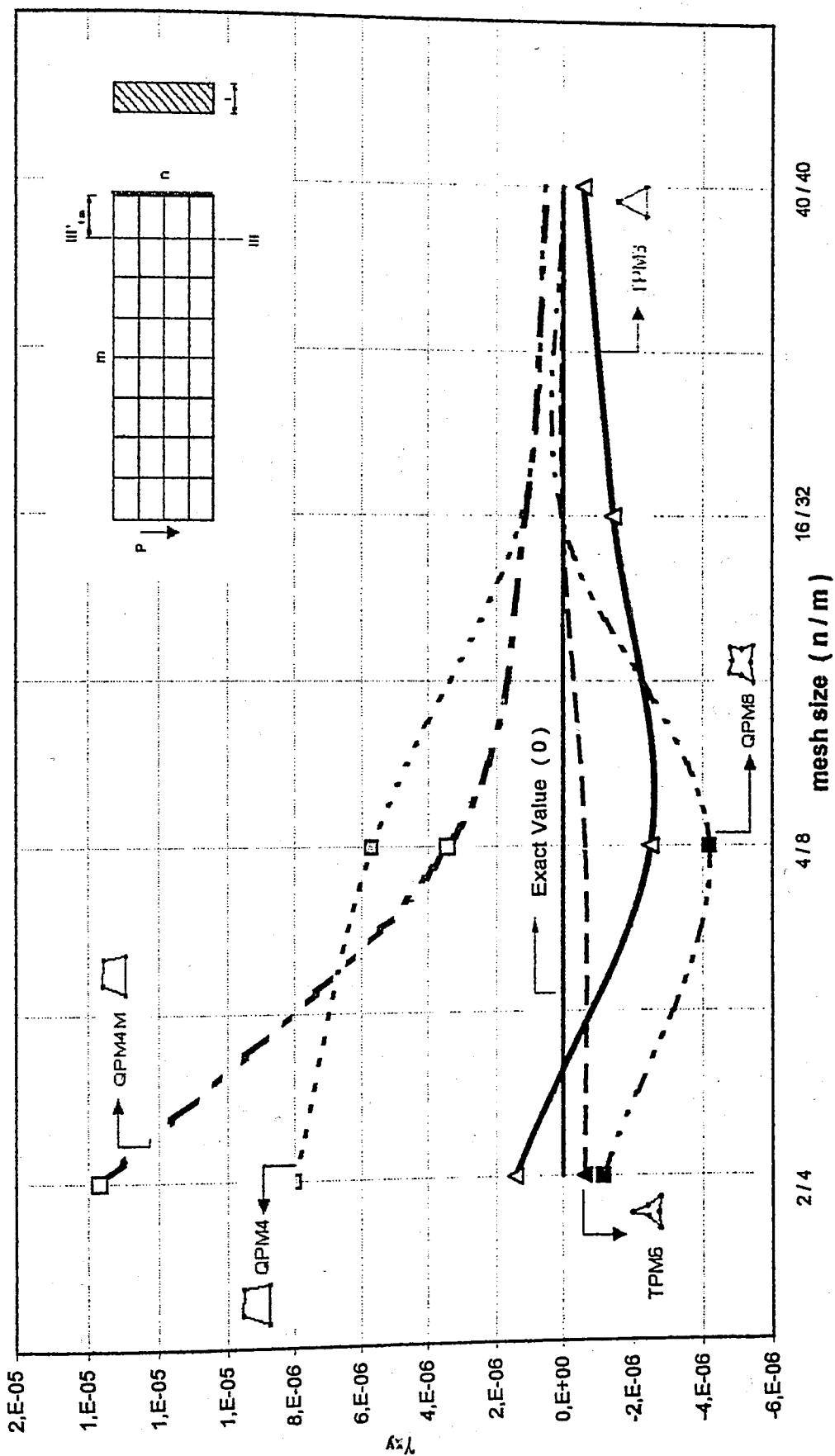


Figure B.58. Comparison of maximum strain (ϵ_y) at section III-III' (40 / 80 mesh size)

Figure B.59. Shear strain (γ_{xy}) at section III-III'

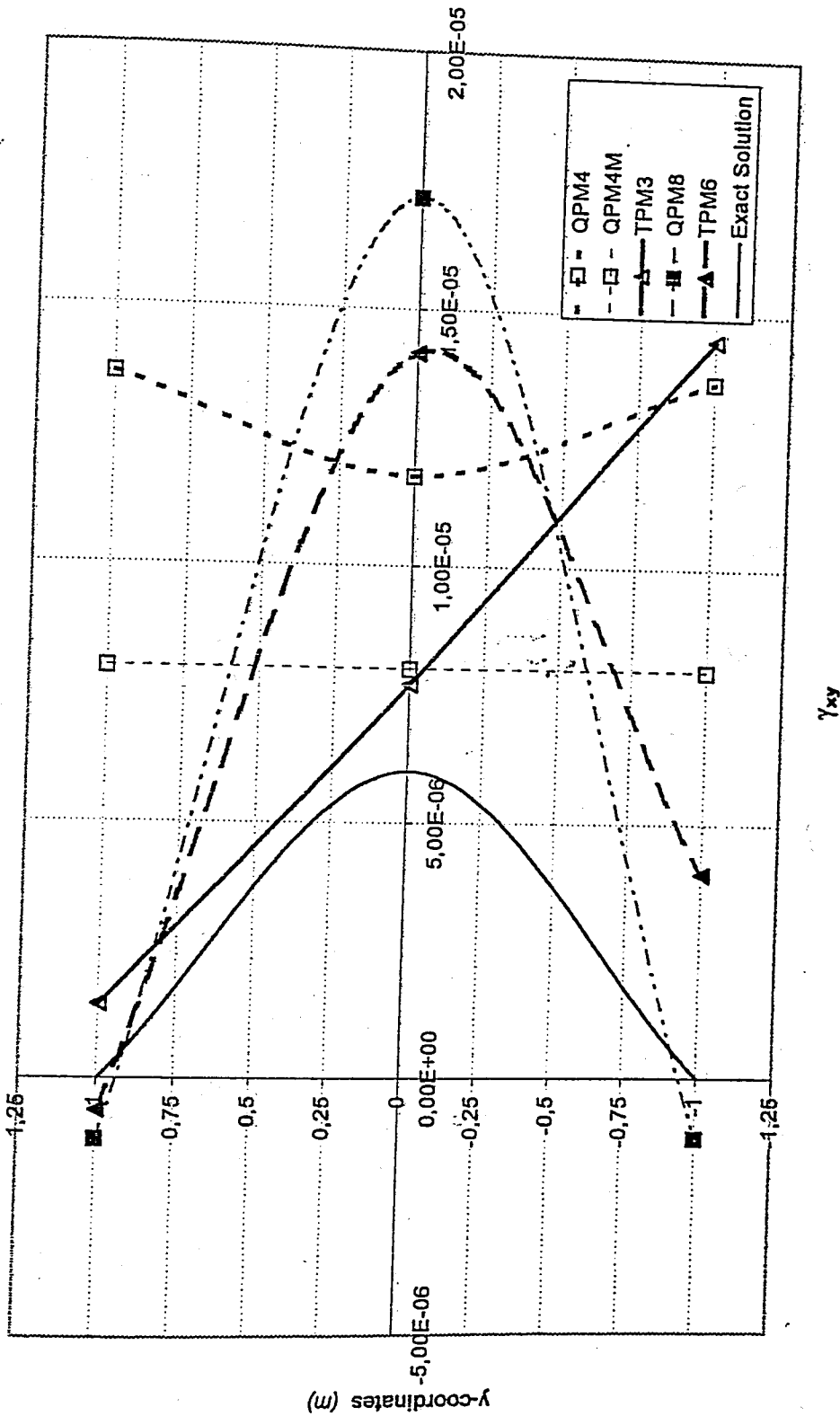


Figure B.60. Comparison of shear strain (γ_{xy}) at section III-I-III' (2 / 4 mesh size)

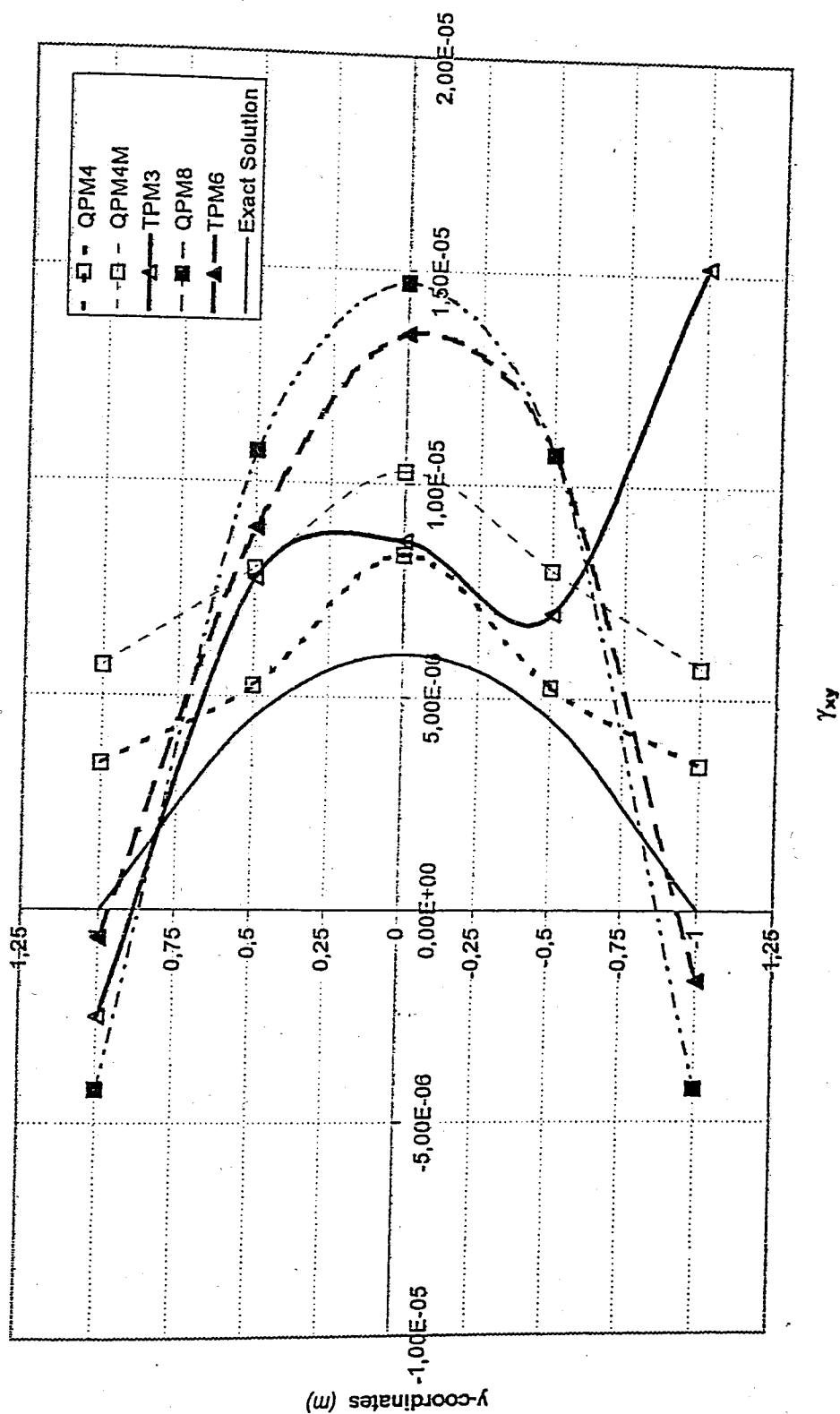


Figure B.61. Comparison of shear strain (γ_{xy}) at section III-III' (4/8 mesh size)

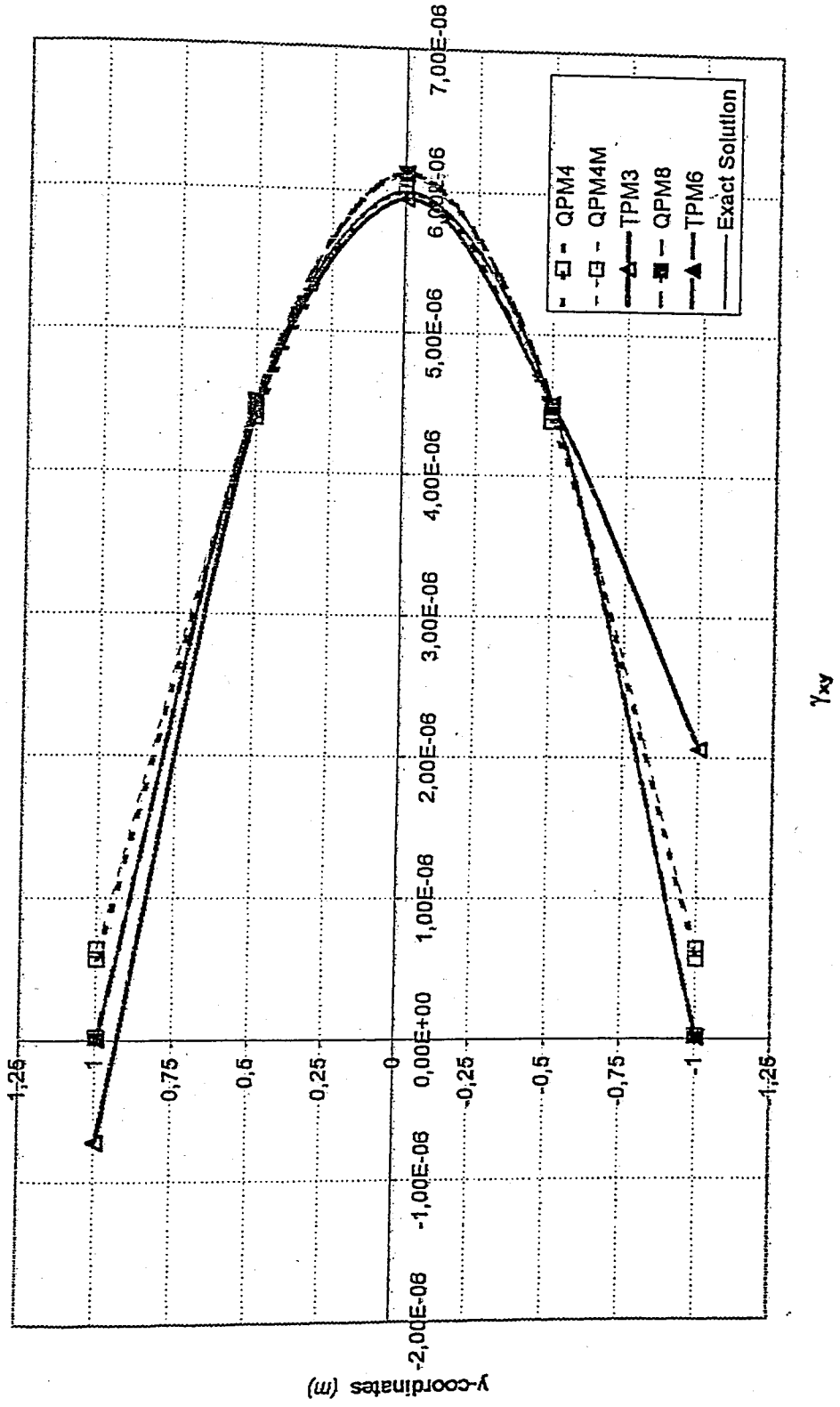


Figure B.62. Comparison of shear strain (γ_{xy}) at section III-III' (16 / 32 mesh size)

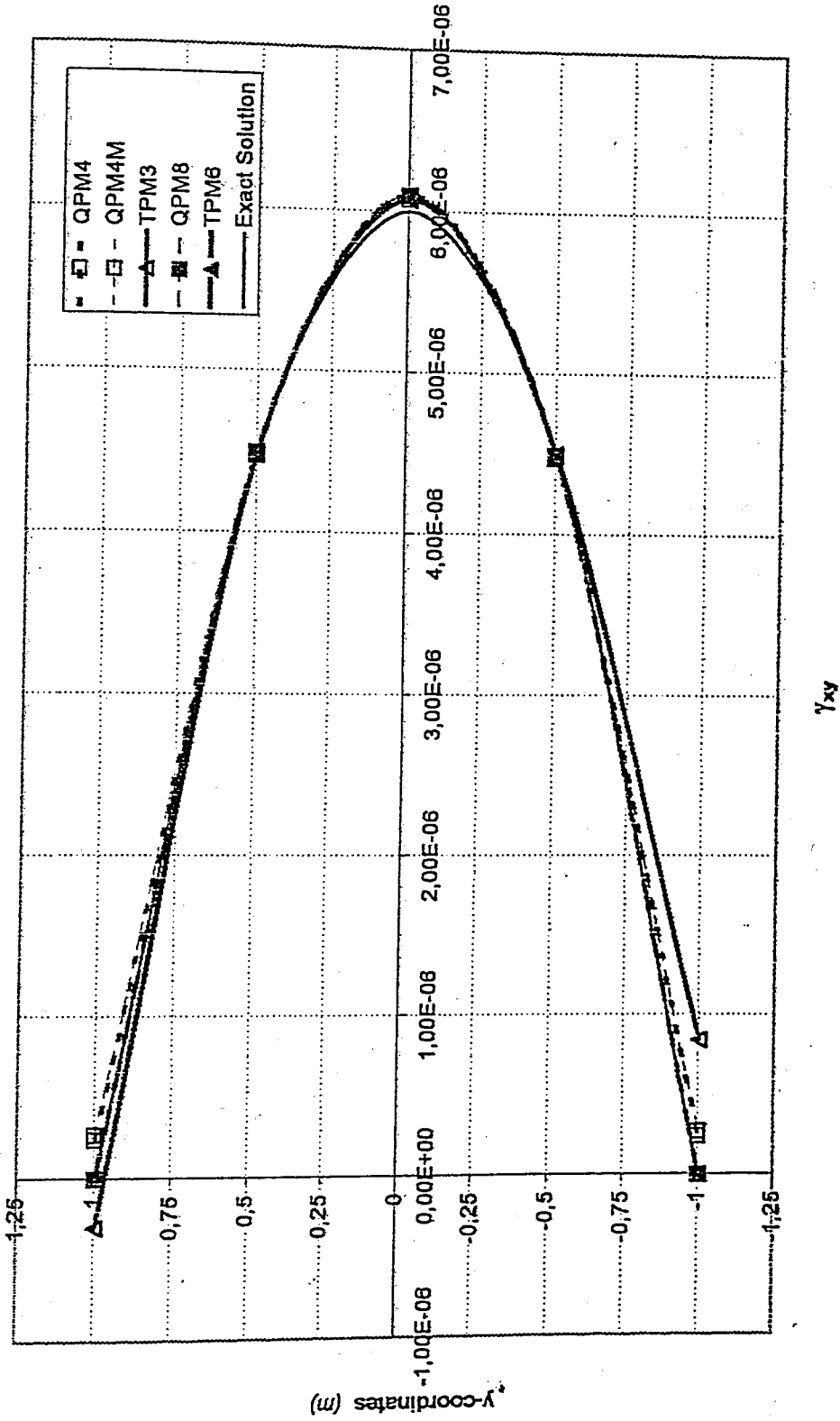


Figure B.63. Comparison of shear strain (γ_{xy}) at section III-III (40/80 mesh size)

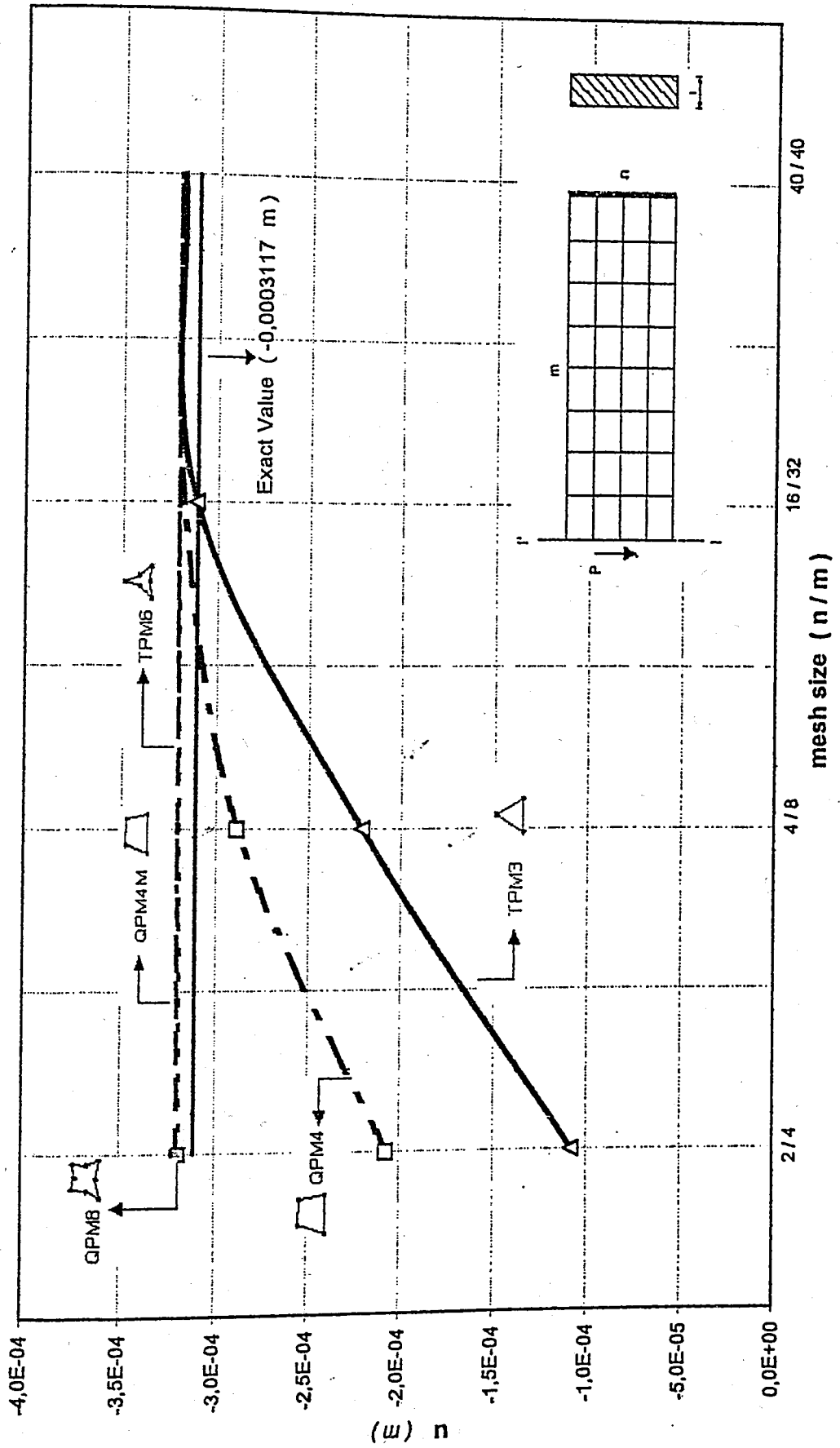
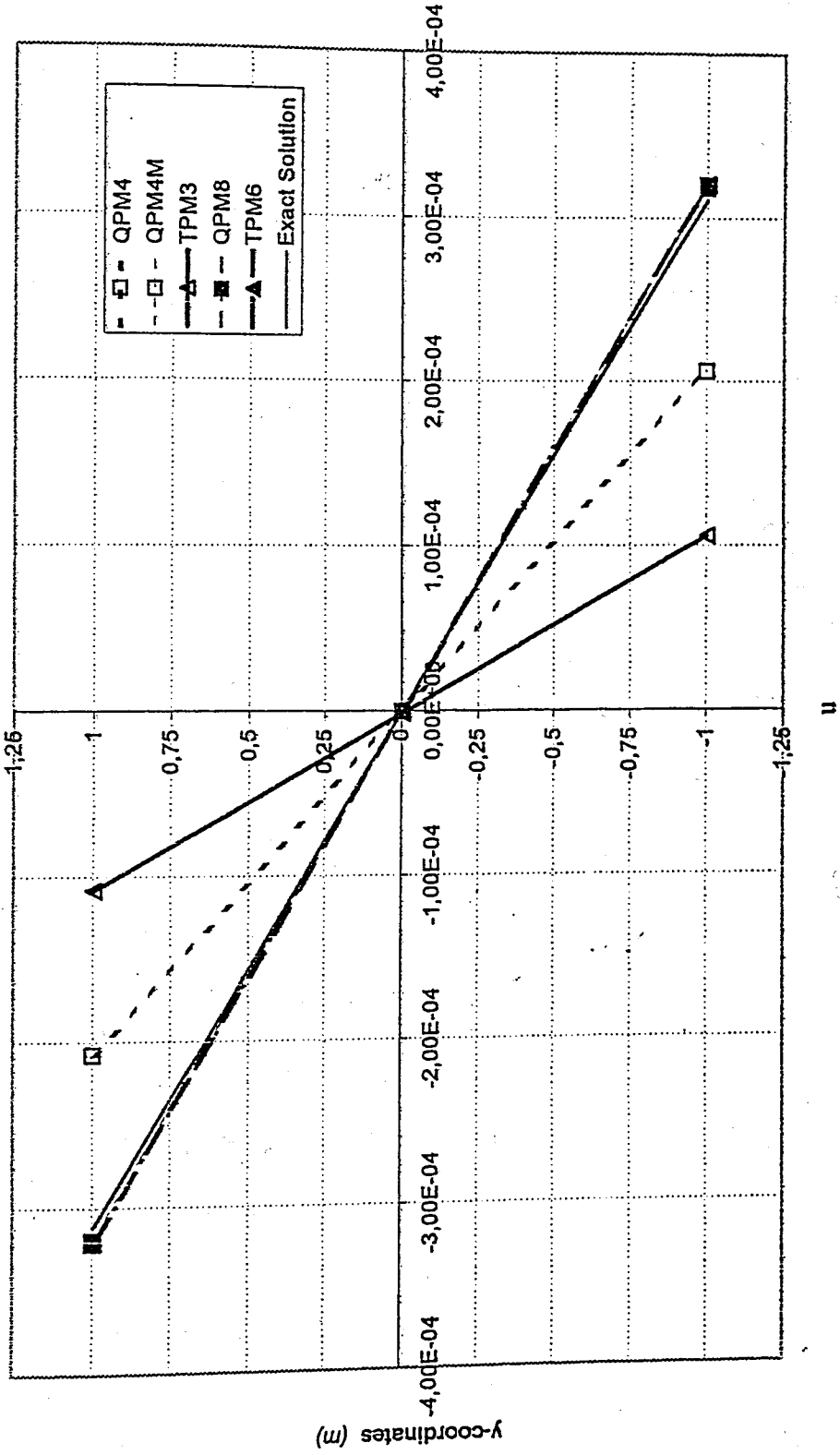


Figure B.64. Horizontal deflection (u) at section I-I'

Figure B.65. Comparison of horizontal deflection (u) at section I-I' (2 / 4 mesh size)

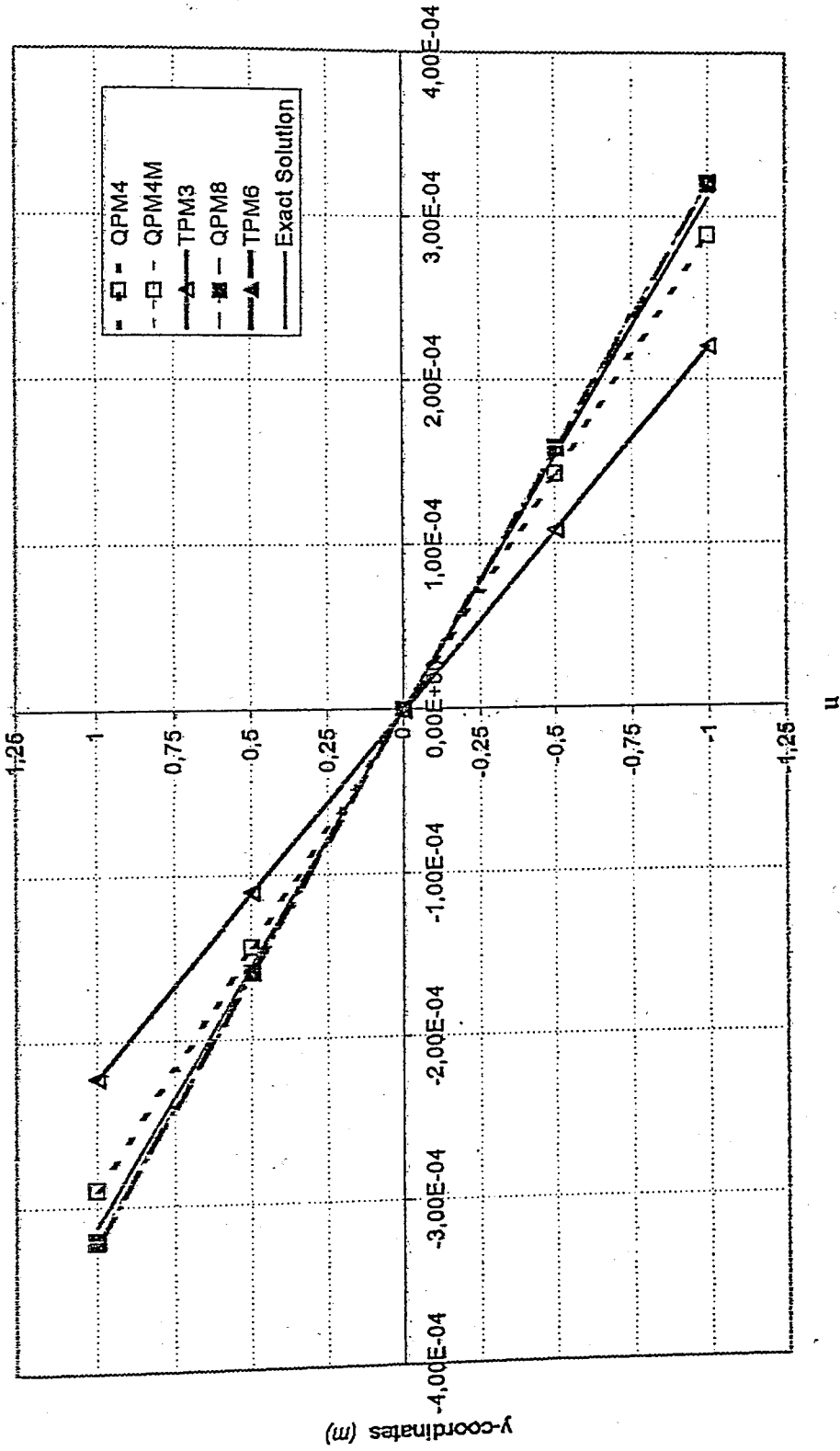


Figure B.66. Comparison of horizontal deflection (u) at section I-I' (4 / 8 mesh size)

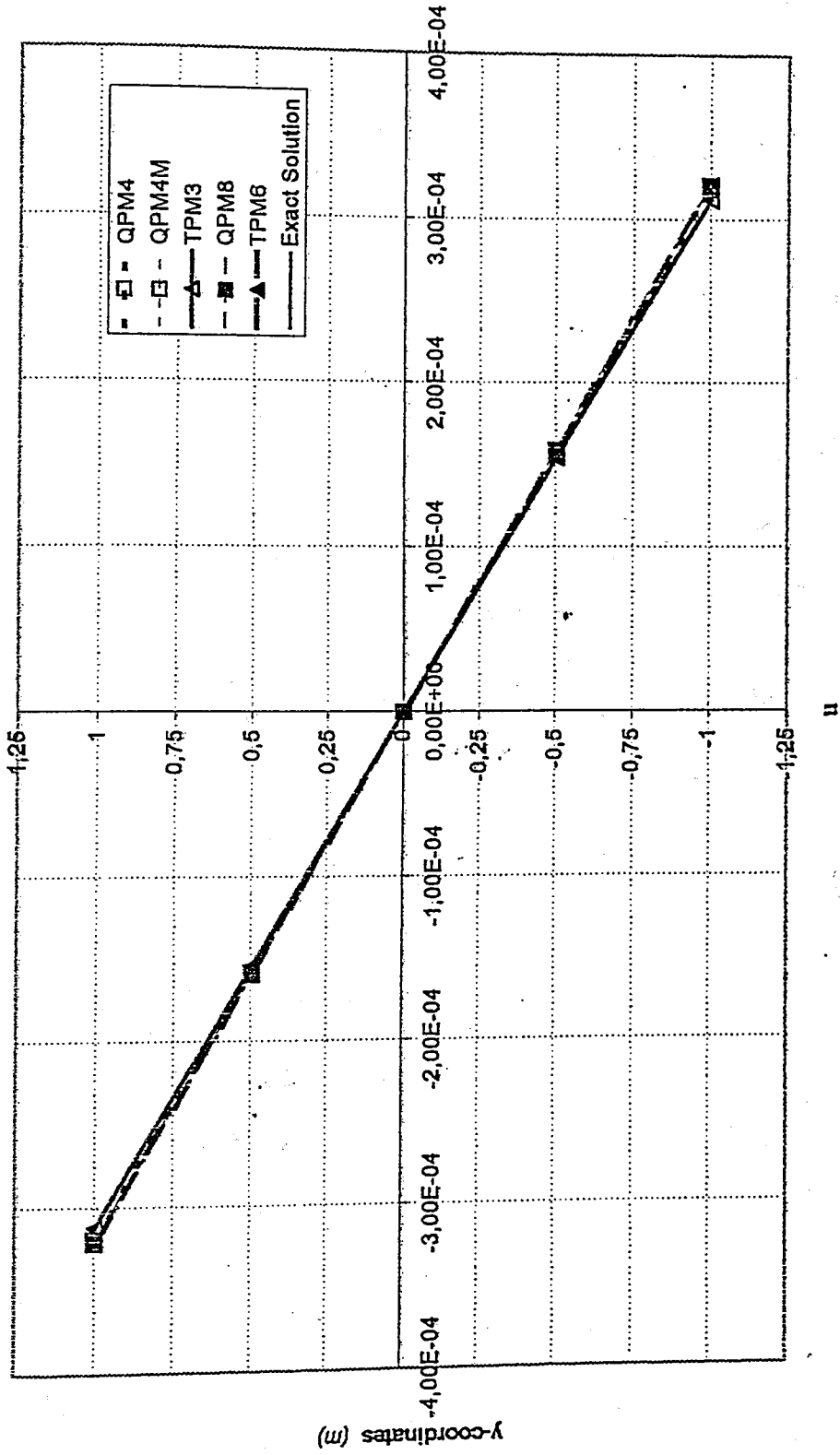


Figure B.67. Comparison of horizontal deflection (u) at section I-I' (16 / 32 mesh size)

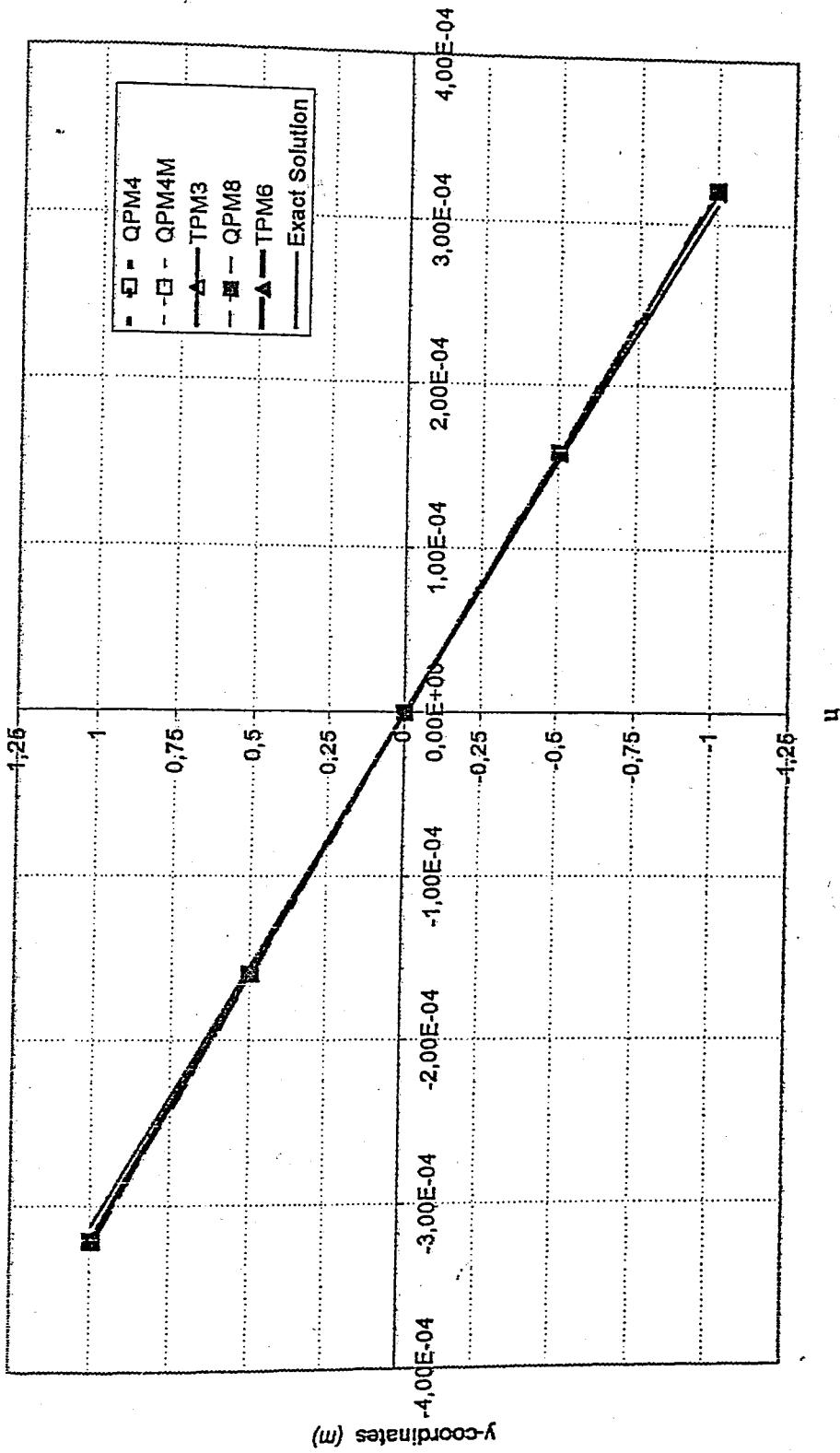


Figure B.68. Comparison of horizontal deflection (u) at section I-I' (40/80 mesh size)

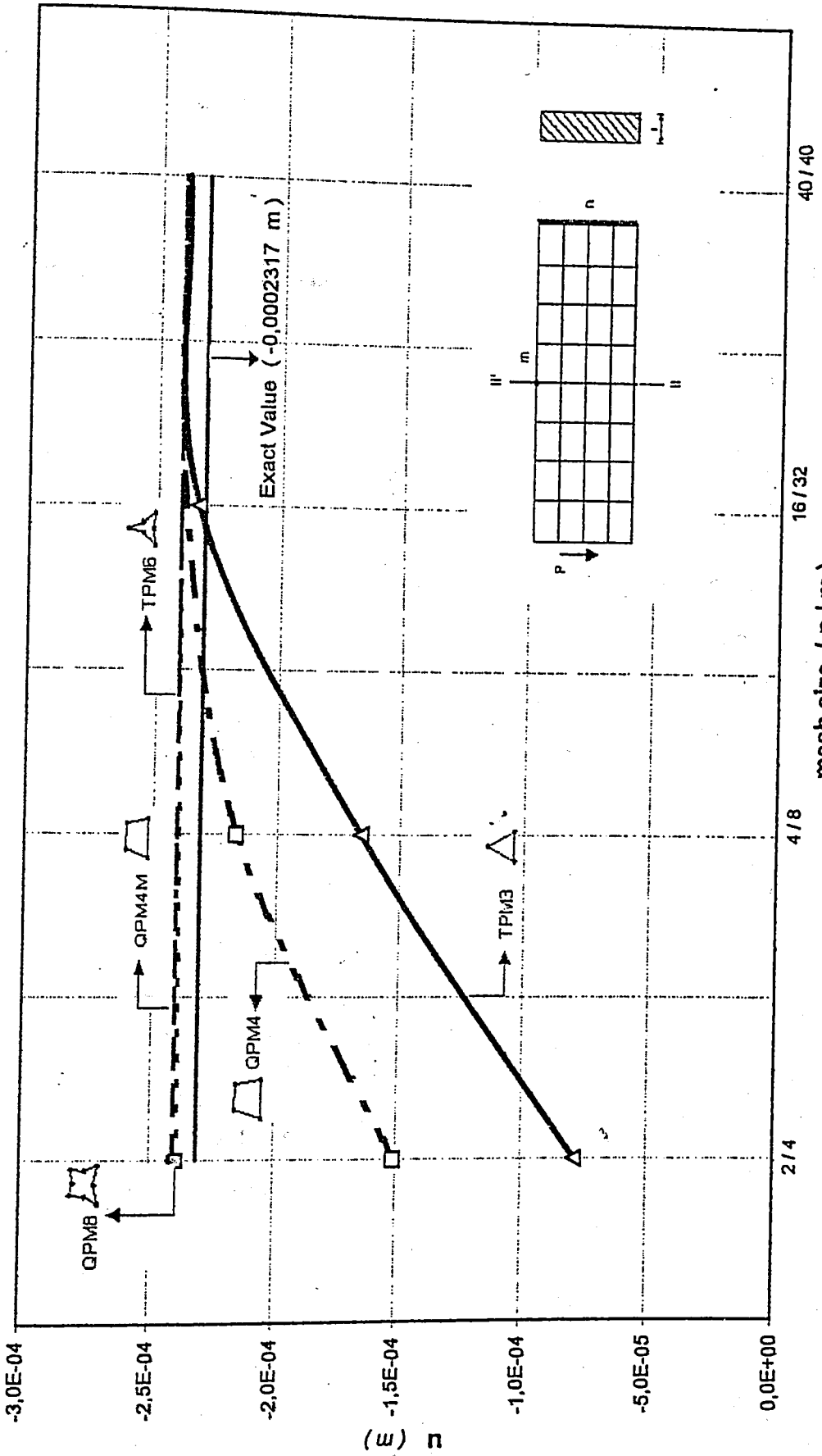


Figure B.69. Horizontal deflection (u) at section II-II'

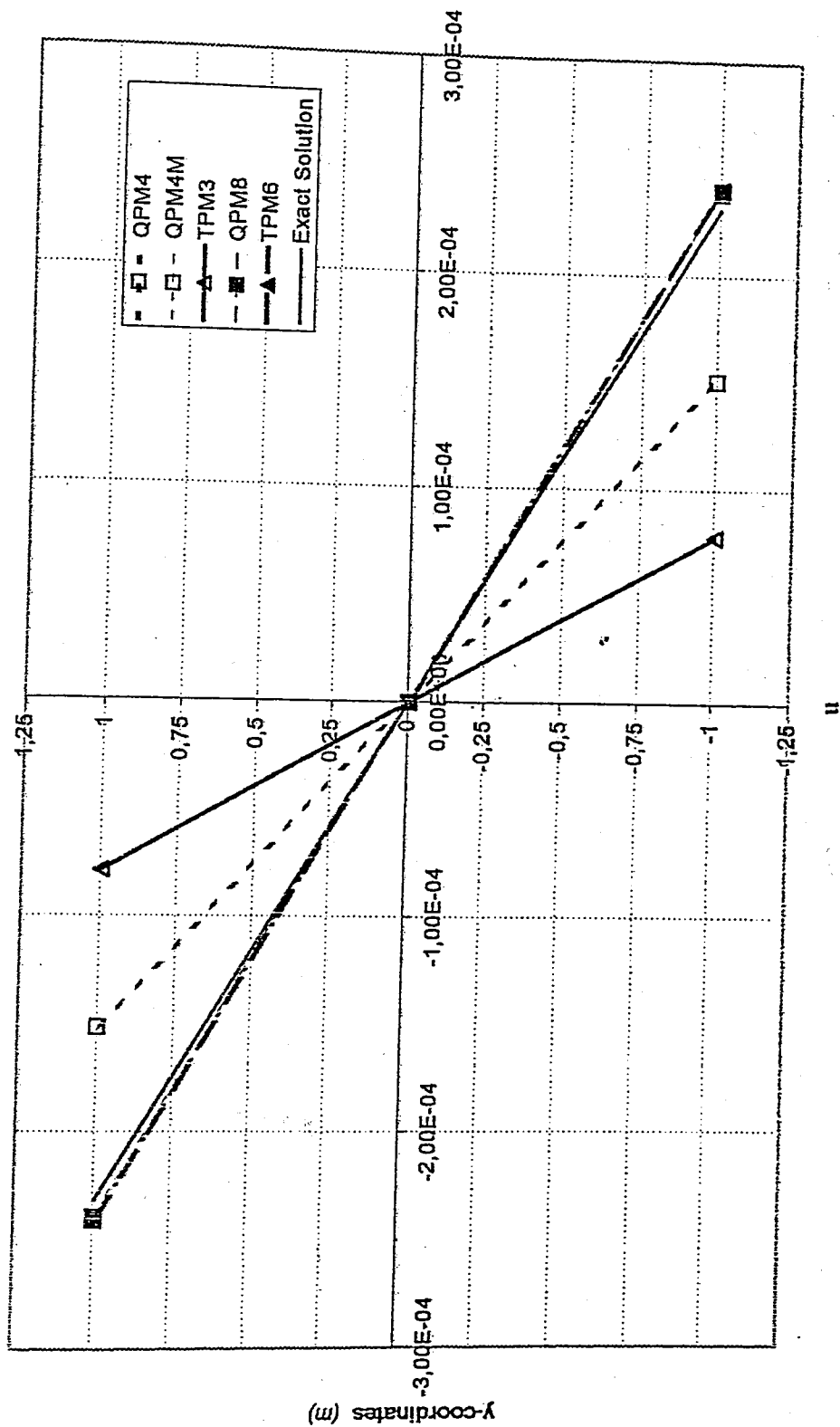


Figure B.70. Comparison of horizontal deflection (u) at section II-II' (2/4 mesh size)

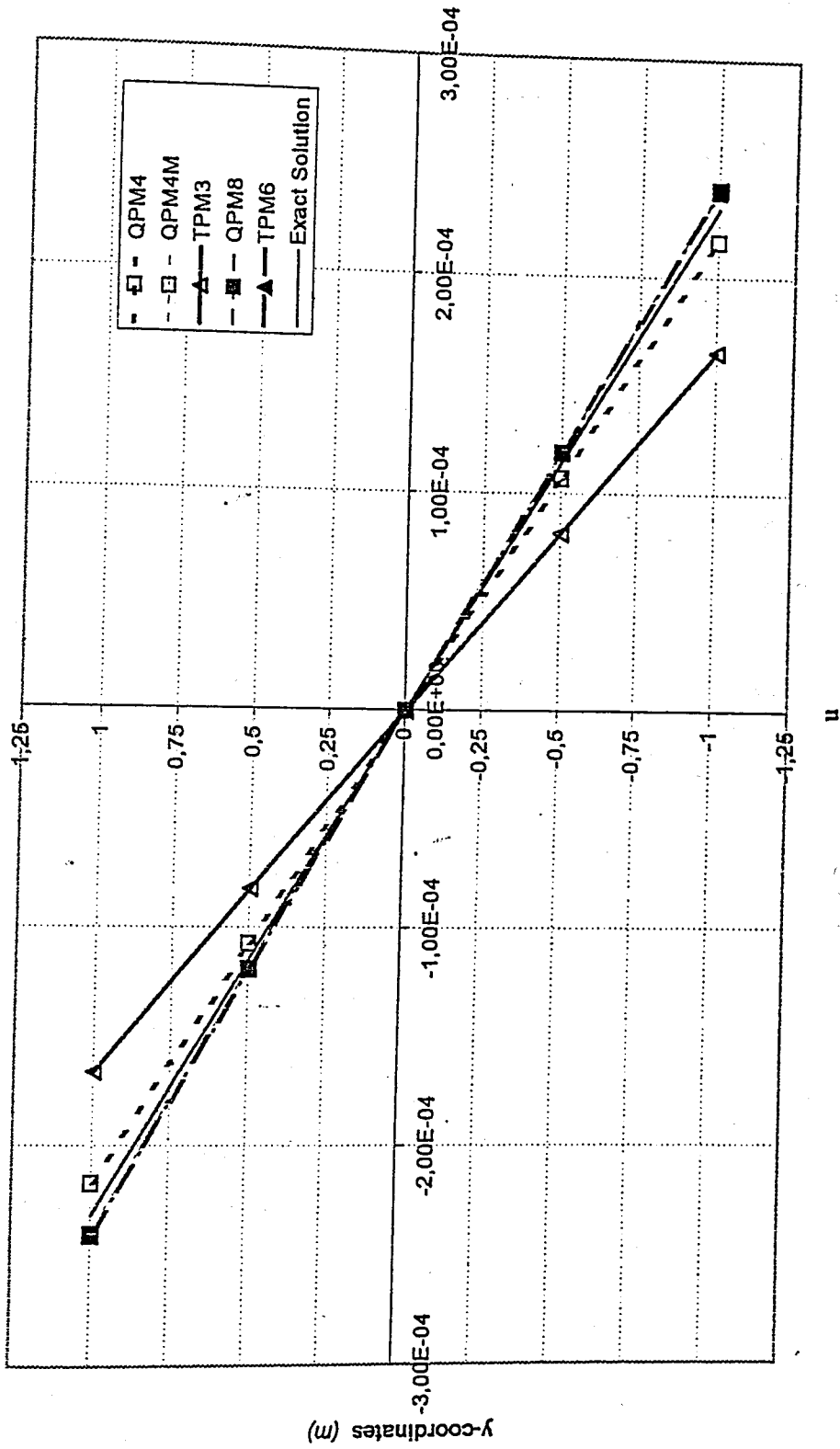


Figure B.71. Comparison of horizontal deflection (u) at section II-II' (4 / 8 mesh size)

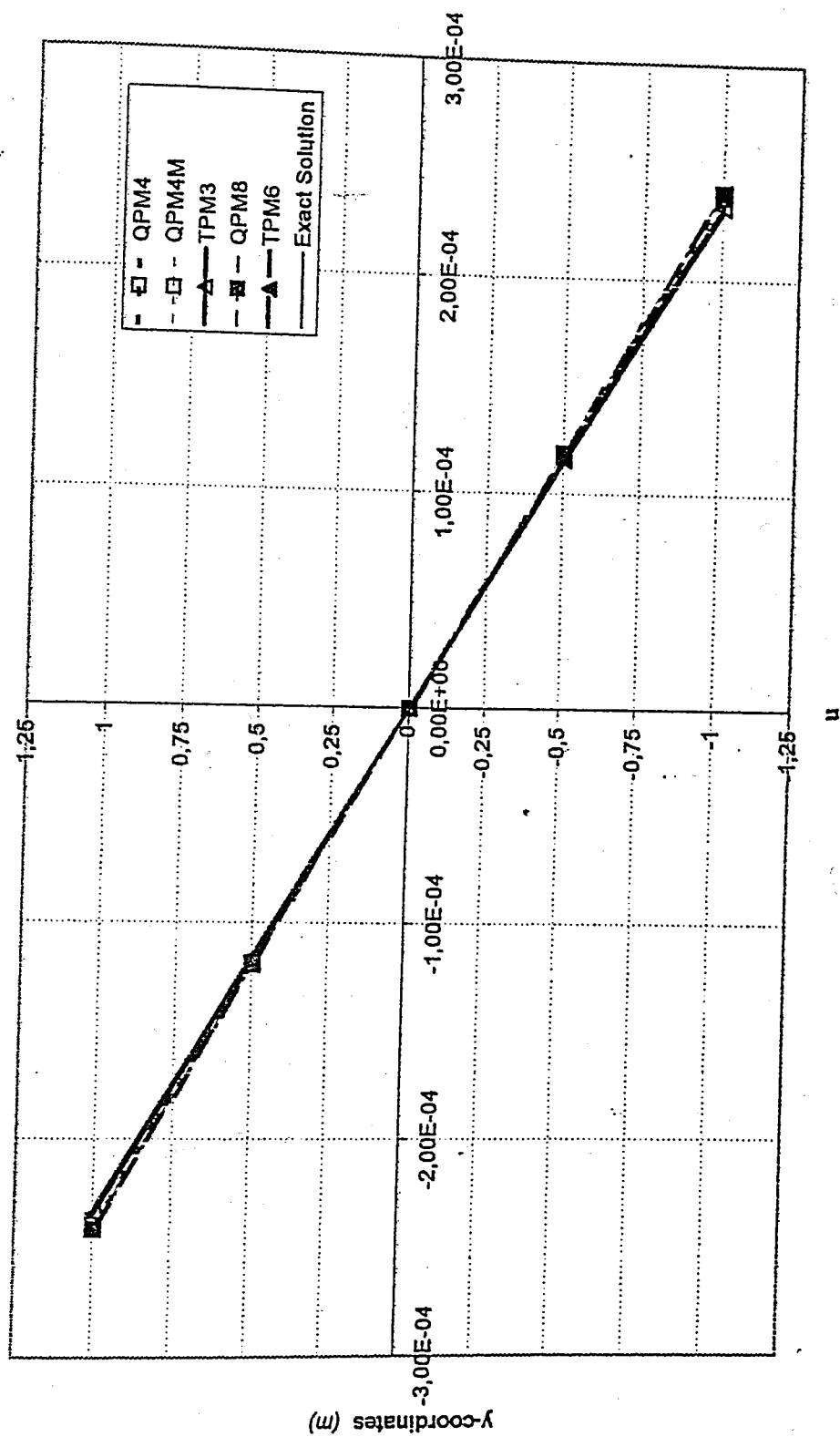


Figure B.72. Comparison of horizontal deflection (u) at section II-II' (16/32 mesh size)

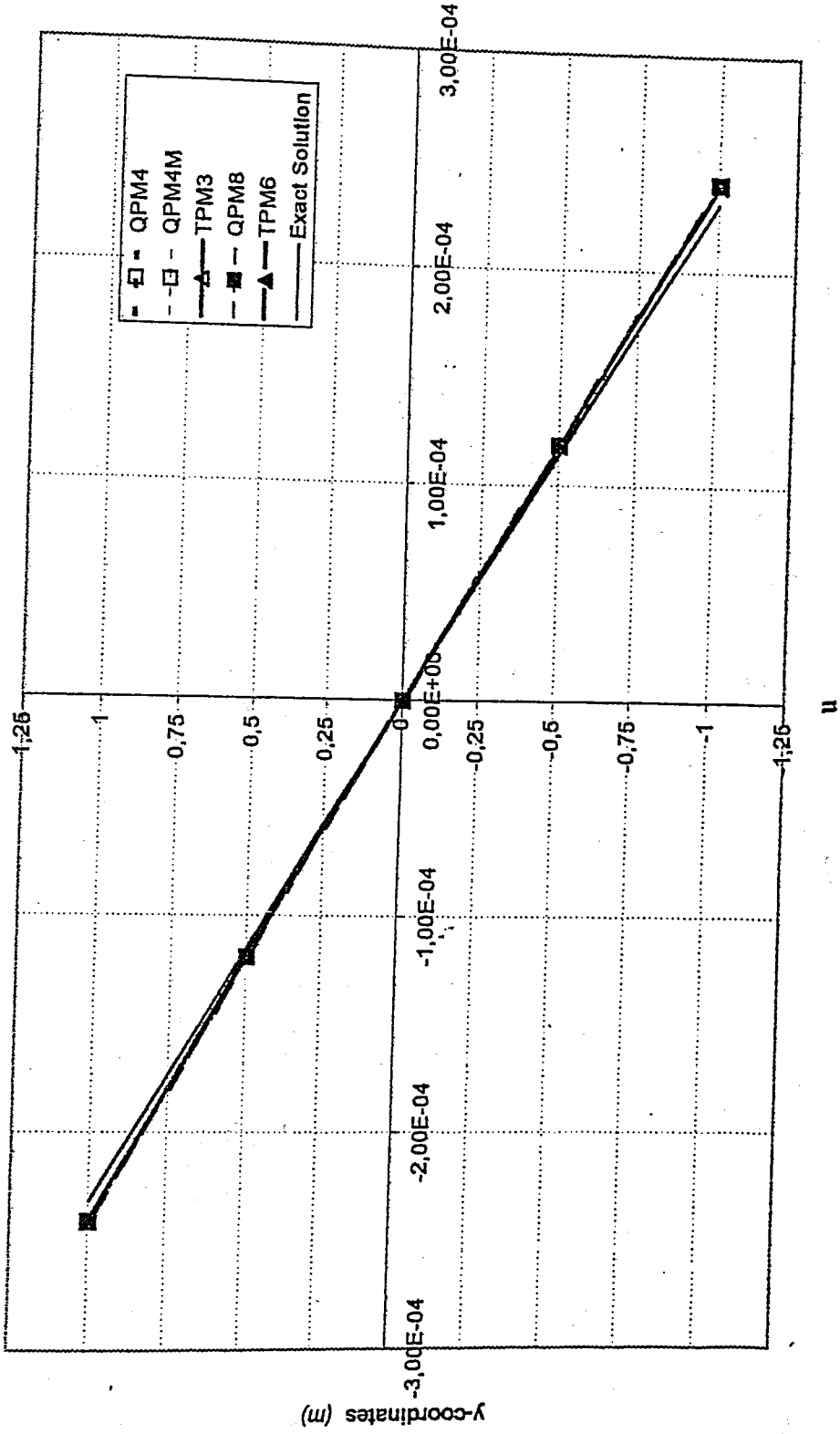


Figure B.73. Comparison of horizontal deflection (u) at section II-II' (40 / 80 mesh size)

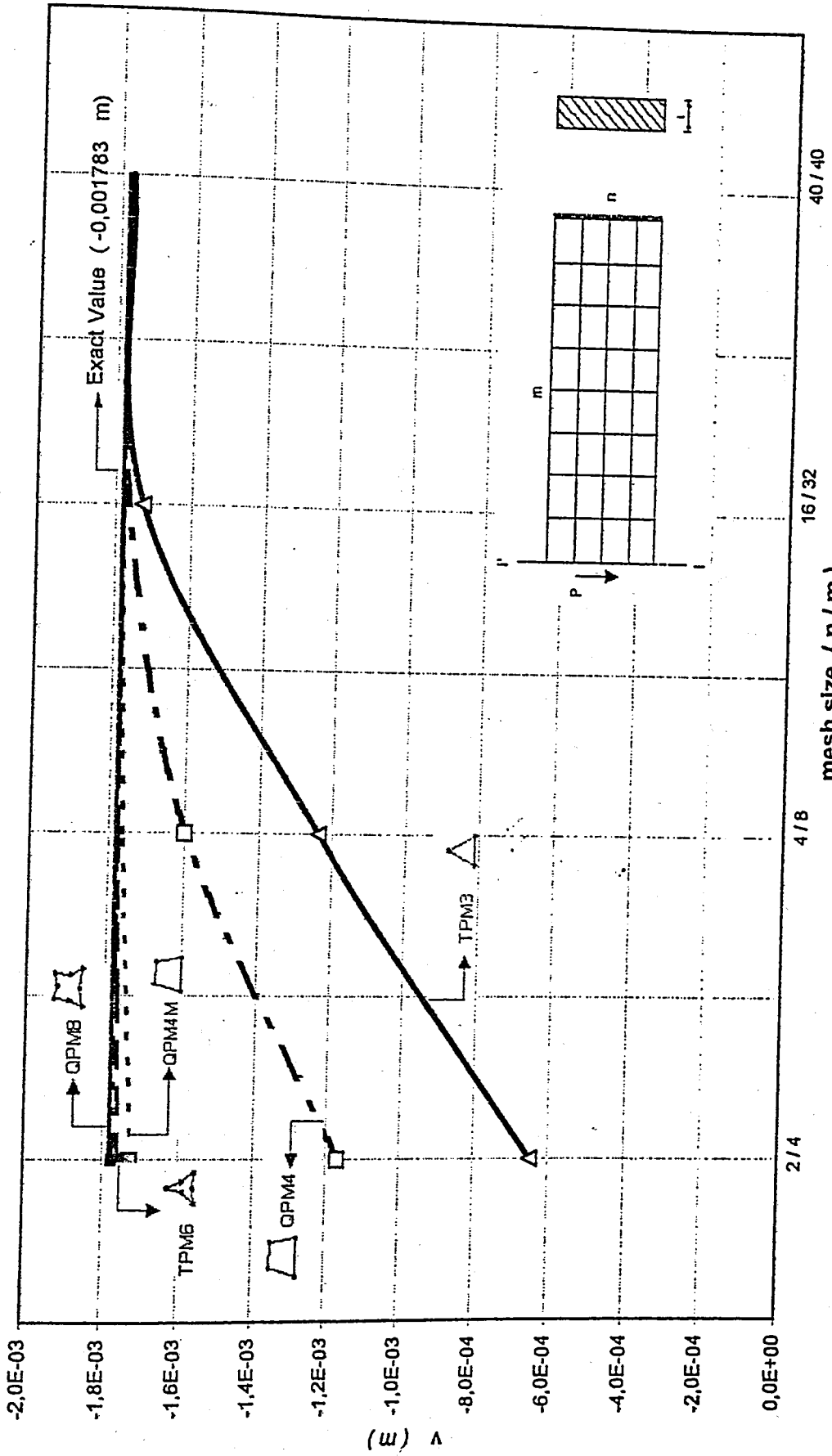


Figure B.74. Vertical deflection (v) at section I-I"

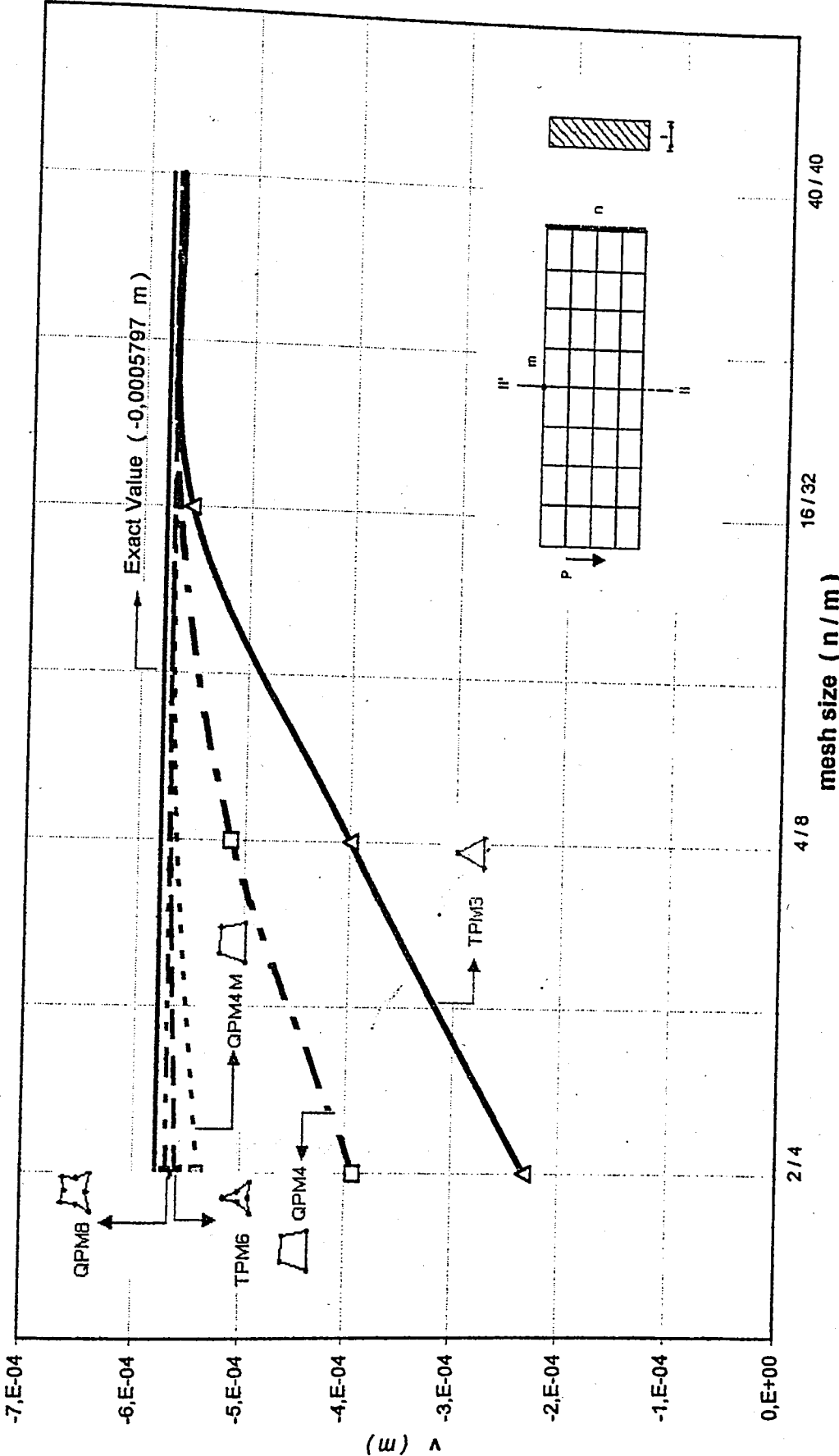


Figure B.75. Vertical deflection (v) at section II-II'

APPENDIX C : ISOFLEX THIN PLATE ELEMENTS (QF4, TF3,)

C.1. Formulation

The isoflex family of thin plate elements is formed by applying Kirchhoff constraints within elements formulated using Midlin plate assumptions w, θ_x, θ_y - at the corner nodes.

w, θ_x, θ_y - at the corner nodes,

$\Delta w, \Delta \theta_x, \Delta \theta_y$ - at the mid-side nodes of the quadrilateral,

and

$\Delta \theta_x, \Delta \theta_y$ - at the central node of the triangle.

where $\Delta \theta$ and θ are the relative (departure from linearity) and absolute rotations of the through-thickness normals after deformation. These rotations include the transverse shear deformations.

An element with thin plate performance is then produced by constraining the shear strains to zero at discrete points within the element. These constraints provide extra equations that permit certain nodal degrees of freedom to be discarded. The final nodal configurations are

w, θ_x, θ_y - at the corner nodes,

$\Delta \theta$ - at the mid-side nodes.

where $\Delta \theta$ is the relative rotation about a tangent to the element edge. This removes 8 and 11 degrees of freedom for the 6 and 3 noded triangles and 11 and 15 degrees of freedom for the 8 and 4 noded quadrilaterals respectively. This is achieved by using the following constraints, originally proposed by Irons for the Semiloof shell

$$(i) \quad \gamma_t = \frac{\partial w}{\partial x} - \theta_y = 0$$

where γ_t is the through-thickness shear strain tangential to the element edges. This provides 6 and 8 constraints respectively for the triangles and quadrilaterals which are suitable for eliminating the mid-side translation and normal rotation.

$$(ii) \quad \int_A \gamma_{xz} dA = 0, \quad \int_A \gamma_{yz} dA = 0$$

where the integral is performed using 2*2 Gauss quadrature. This provides 2 constraints for both the triangles and quadrilaterals, which are suitable for removing the rotations at the central node.

$$(iii) \quad \int_s \gamma_n dS = 0$$

where γ_n is the transverse shear strain normal to the element sides and the integral is performed using 2-point quadrature along each side. This provides one constraint suitable for removing the central translation of the quadrilaterals.

These constraints are sufficient for the higher order elements and the extra constraints required for the lower order elements are provided by enforcing a linear variation of tangential rotation along the element sides.

The infinitesimal strain-displacement relationship is derived from the 3-D continuum relationship by neglecting ϵ_z which is zero in the Midlin plate assumptions, and γ_{xz} and γ_{yz} which have been constrained to zero, so that

$$\epsilon_x = \frac{\partial U}{\partial X}$$

$$\epsilon_y = \frac{\partial V}{\partial Y} \quad (C.1)$$

$$\gamma_{xy} = \frac{\partial U}{\partial Y} + \frac{\partial V}{\partial X}$$

The continuum displacements for plates of varying thickness are related to the original degrees of freedom of the plate using

$$U = \frac{z}{t} \sum_{i=1}^n N_i(\xi, \eta) t_i \theta_{Y_i}$$

$$V = -\frac{z}{t} \sum_{i=1}^n N_i(\xi, \eta) t_i \theta_{X_i} \quad (C.2)$$

$$W = \sum_{i=1}^n N_i(\xi, \eta) W_i$$

where t and t_i are the thick nesses of the plate at the integration and nodal points respectively, and $N_i(\xi, \eta)$ are the element shape functions. Therefore the discretised, generalized, flexural strain-displacement relationship is

$$\begin{bmatrix} \Psi_X \\ \Psi_Y \\ \Psi_{XY} \end{bmatrix} = \sum_{i=1}^n \begin{bmatrix} \frac{1}{t} \frac{\partial t}{\partial X} \frac{\partial N_i}{\partial X} & 0 & \frac{t_i}{t} \frac{\partial N_i}{\partial X} \\ \frac{1}{t} \frac{\partial t}{\partial Y} \frac{\partial N_i}{\partial Y} & -\frac{t_i}{t} \frac{\partial N_i}{\partial Y} & 0 \\ \frac{1}{t} \frac{\partial t}{\partial Y} \frac{\partial N_i}{\partial X} + \frac{1}{t} \frac{\partial t}{\partial X} \frac{\partial N_i}{\partial Y} & -\frac{t_i}{t} \frac{\partial N_i}{\partial Y} & \frac{t_i}{t} \frac{\partial N_i}{\partial Y} \end{bmatrix} \begin{bmatrix} W_i \\ \theta_{X_i} \\ \theta_{Y_i} \end{bmatrix} \quad (C.3)$$

where the terms involving $\frac{\partial t}{\partial X}$ and $\frac{\partial t}{\partial Y}$ are the small strain contributions due to thickness variations. For flat plates

$$\begin{bmatrix} \Psi_X \\ \Psi_Y \\ \Psi_{XY} \end{bmatrix} = \begin{bmatrix} -\frac{\partial^2 W}{\partial X^2} \\ -\frac{\partial^2 W}{\partial Y^2} \\ -2\frac{\partial^2 W}{\partial X \partial Y} \end{bmatrix} \quad (C.4)$$

The isotropic and orthotropic elastic resultant modulus or rigidity matrices are

$$\text{Isotropic} \quad \hat{\bar{D}} = \frac{Et^3}{12(1-\nu)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{(1-\nu)}{2} \end{bmatrix} \quad (C.5)$$

$$\text{Orthotropic} \quad \hat{\bar{D}} = \frac{t^3}{12} \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & 0 \\ 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix}^{-1} \quad (C.6)$$

where ν_{xy} has been set $\nu_{xy}E_x/E_x$ to maintain symmetry.

C.2. Cases on Usage

This element may be used to analyze any 2-D plate type structures where transverse shear effects do not influence the solution, e.g. thin cantilever plates. Generally, the 4-noded QF4 element is most effective, however the 8-noded QF8 element may be more appropriate for structures with curved boundaries.

APPENDIX D : ISOPARAMETRIC THICK MINDLIN PLATE (QTF8, TTF6)

D.1. Formulation

The QTF8 and TTF6 elements are isoparametric plate elements formulated using Mindlin plate theory, which assumed that

- (i) Normal stress in the transverse stress is negligible in comparison with the in plane stresses,
- (ii) Normals to the mid-surface remain straight but not necessarily normal to the mid-surface after deformation.

Thus the element account for the transverse shear effects associated with thicker plate elements. The theory also permits treatment of lateral displacement and rotations as independent variables, producing elements which only require $C(0)$ continuity.

The nodal degrees of freedom are

$$w, \theta_x, \theta_y \quad \text{at the corner nodes}$$

where θ_x, θ_y are the rotation of the normals to the mid-surface and include the effects of shear deformations.

The infinitesimal, generalized, flexural strain-displacement relationship is derived from the 3-D continuum strain-displacement relationship by neglecting the out of plane strain, so that

$$\Psi_X = \frac{\partial \theta_Y}{\partial X}$$

$$\Psi_Y = -\frac{\partial \theta_Y}{\partial Y}$$

$$\Psi_{XY} = \frac{\partial \theta_Y}{\partial Y} - \frac{\partial \theta_Y}{\partial X} \quad (D.1)$$

$$\gamma_{YZ} = \frac{\partial W}{\partial Y} - \theta_X$$

$$\gamma_{XZ} = \frac{\partial W}{\partial Y} + \theta_Y$$

The elastic resultant modulus or rigidity matrix is defined as

$$\hat{\underset{\sim}{D}} = \begin{bmatrix} \hat{\underset{\sim}{D}}_{\approx b} & 0 \\ 0 & \hat{\underset{\sim}{D}}_{\approx s} \end{bmatrix} \quad (D.2)$$

where, for isotropic materials

$$\hat{\underset{\sim}{D}}_{\approx b} = \frac{Et^3}{12(1-\nu)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{(1-\nu)}{2} \end{bmatrix} \quad (D.3)$$

and

$$\hat{\underset{\sim}{D}}_{\approx s} = \frac{Et}{2.4(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \quad (D.4)$$

and for orthotropic materials

$$\hat{\mathbf{D}}_{\approx b} = \frac{t^3}{12} \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & 0 \\ 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix}^{-1} \quad (\text{D.5})$$

and

$$\hat{\mathbf{D}}_{\approx s} = \frac{t}{1.2} \begin{bmatrix} G_{yz} & 0 \\ 0 & G_{xz} \end{bmatrix} \quad (\text{D.6})$$

where ν_{xy} is equal to $\nu_{xy} E_x / E_x$ to maintain symmetry.

D.2. Cases on Usage

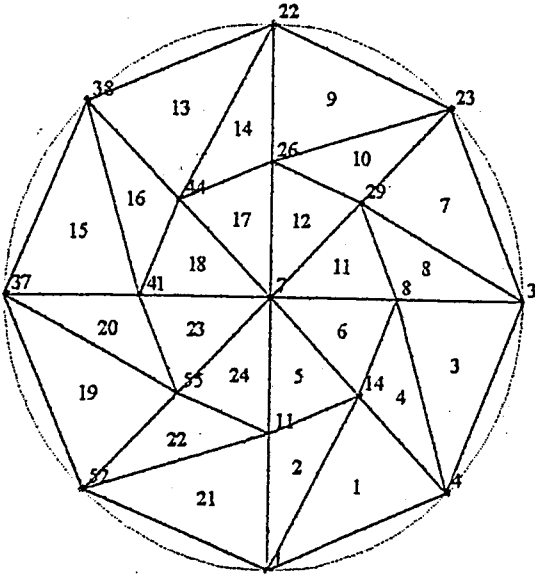
These elements may be used to analyze any 2-D plate type structures where transverse shear effects influence the solution, e.g. perforated thick plates.

Note

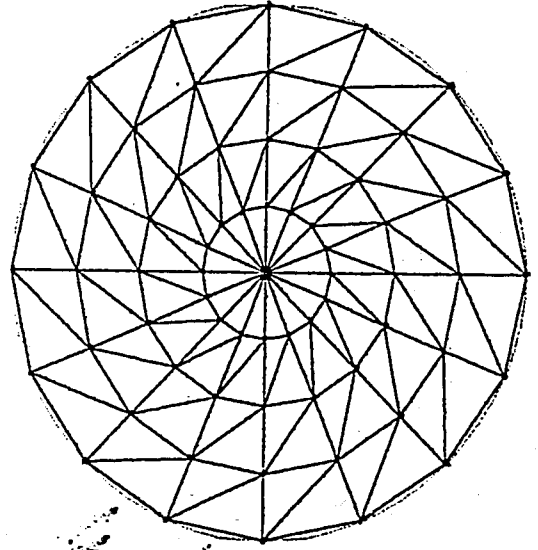
1. These elements are usually more effective than the QSC4 thick shell element.
2. The elements tend to lock as the plate thickness approaches the thin plate limit since shear strain energy dominates the element stiffness. Therefore, a thin plate or shell element should be used when the depth / span ratio exceeds 1/50.
3. 3-D solid elements should be used if the zero normal stress hypothesis may be invalid.
4. Gauss quadrature is utilized to integrate the element matrices.
5. The QTF8 element with 2*2 Gauss quadrature contains one zero energy mode. However, the 2*2 rule is generally more effective than the 3*3 rule.
6. The through-thickness integration is performed explicitly.

APPENDIX E: MESH SIZES OF CIRCULAR PLATE BENDING PROBLEM

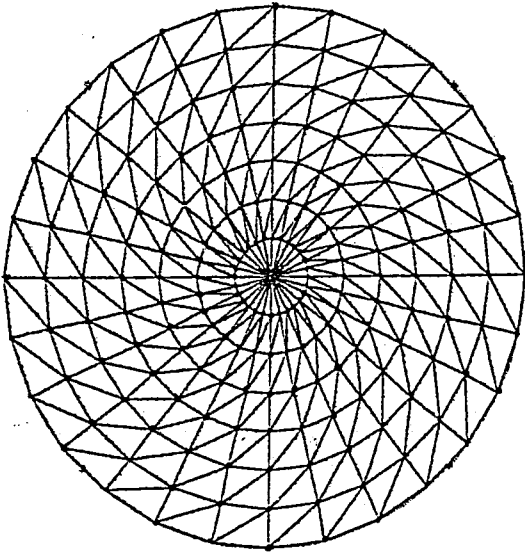
The geometry of the mesh sizes of plate bending elements TF3, TTF6, QF4 and QTF8 are shown.



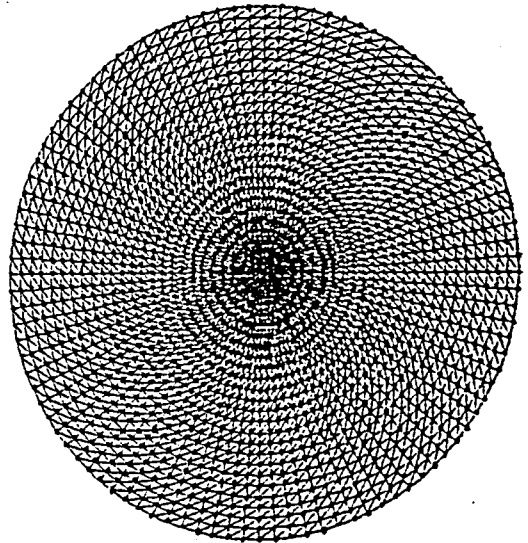
2 / 8 Mesh size



4 / 16 Mesh size

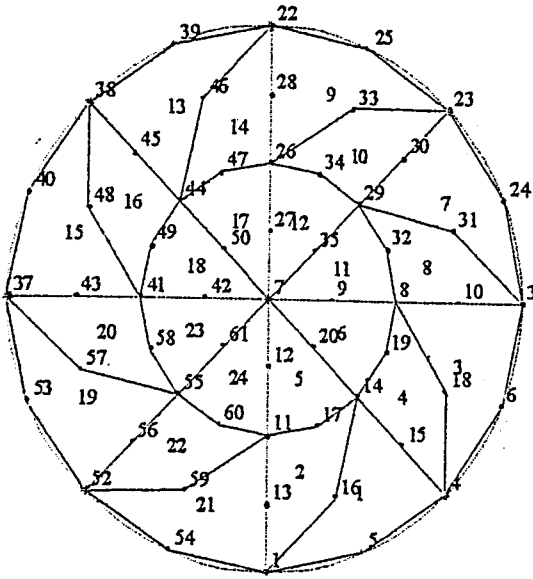


7 / 28 Mesh size

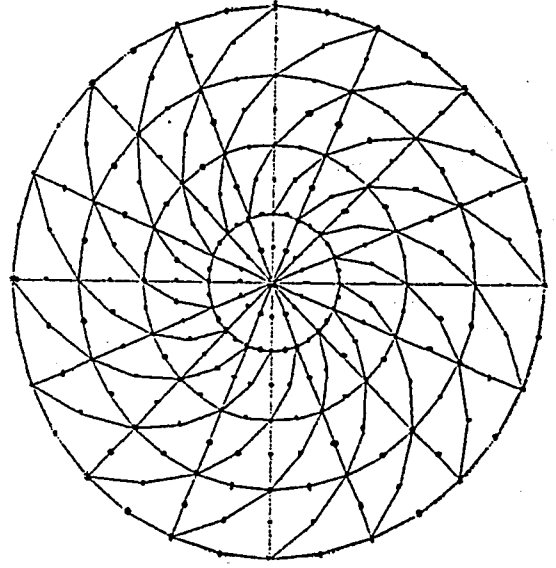


25 / 100 Mesh size

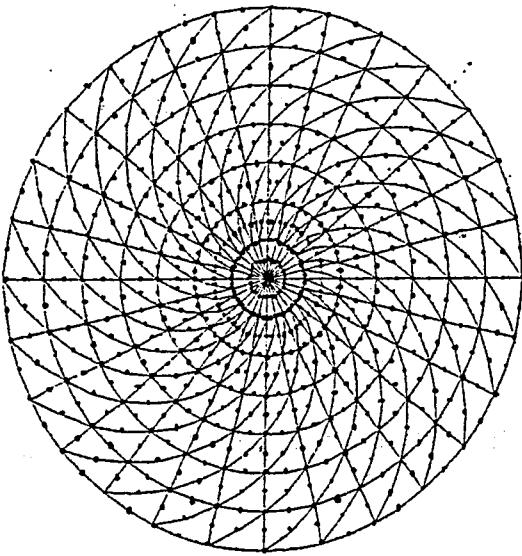
Figure E.1. Mesh sizes of plate element (TF3)



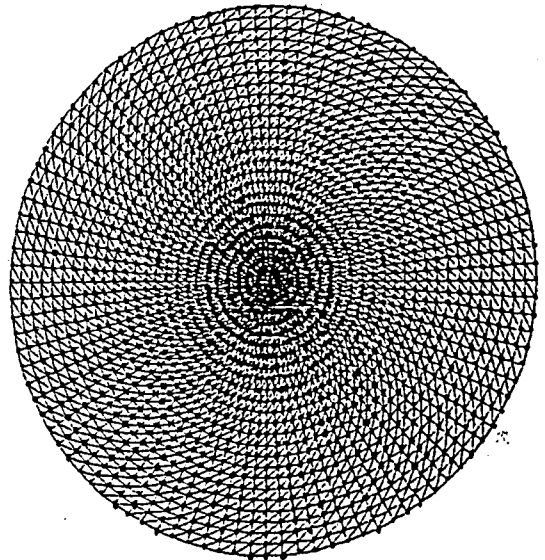
2 / 8 Mesh size



4 / 16 Mesh size

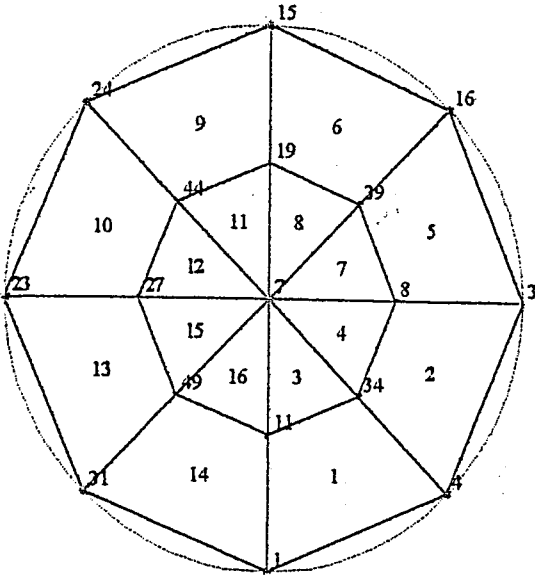


7 / 28 Mesh size

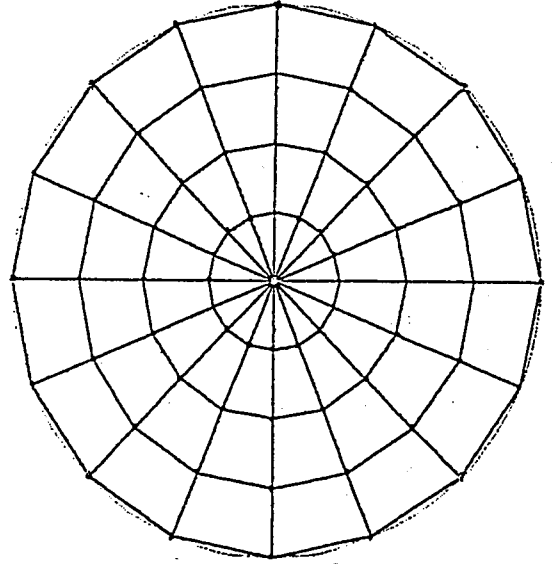


25 / 100 Mesh size

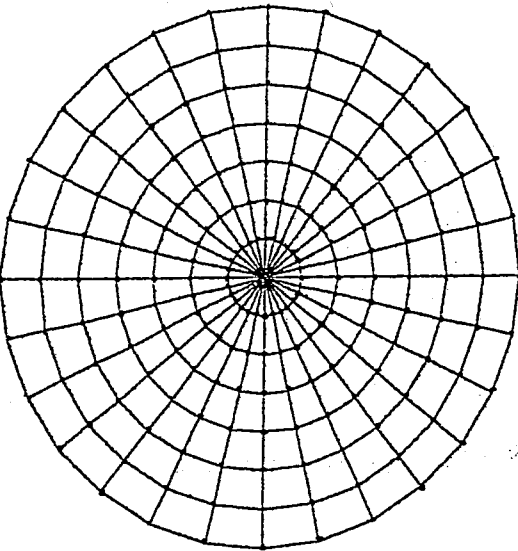
Figure E.2. Mesh sizes of plate element (TTF6)



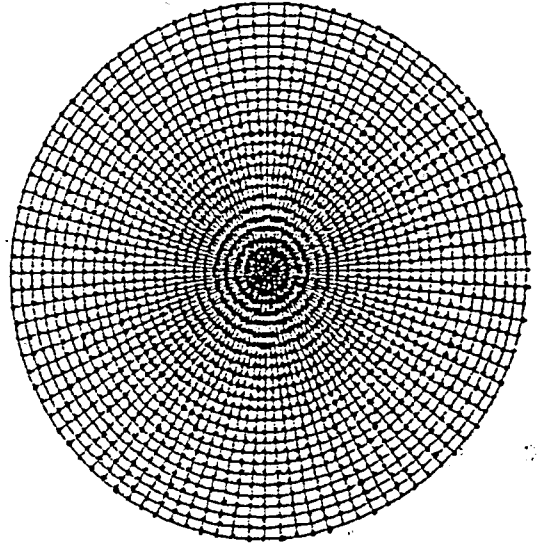
2 / 8 Mesh size



4 / 16 Mesh size

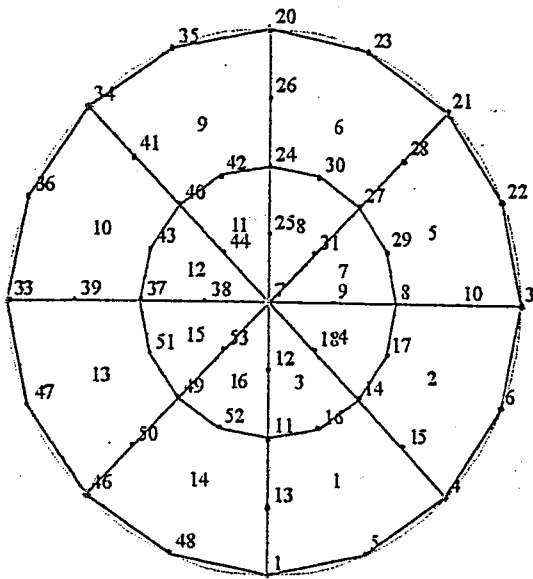


7 / 28 Mesh size

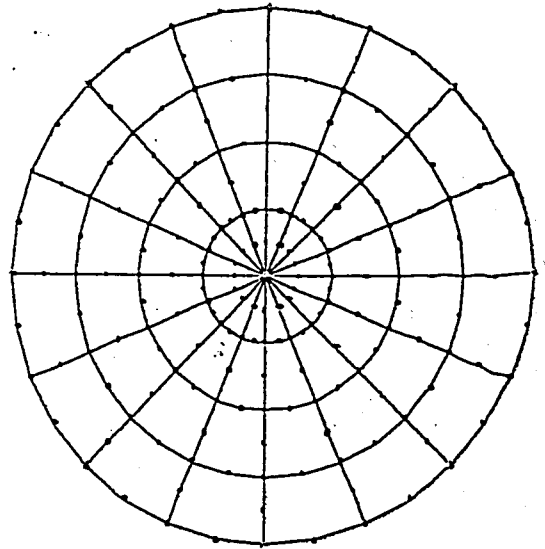


25 / 100 Mesh size

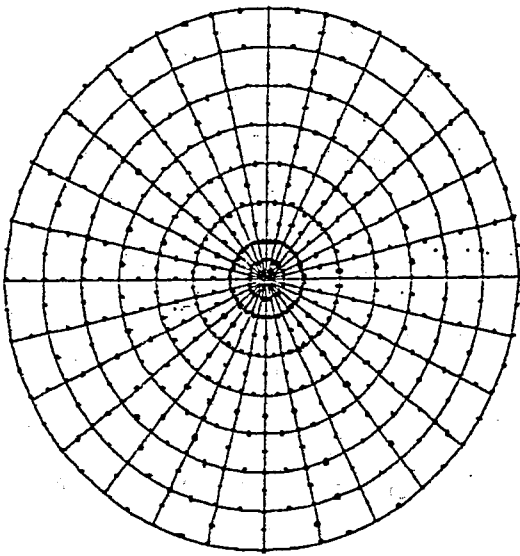
Figure E.3. Mesh sizes of plat bending element (QF4)



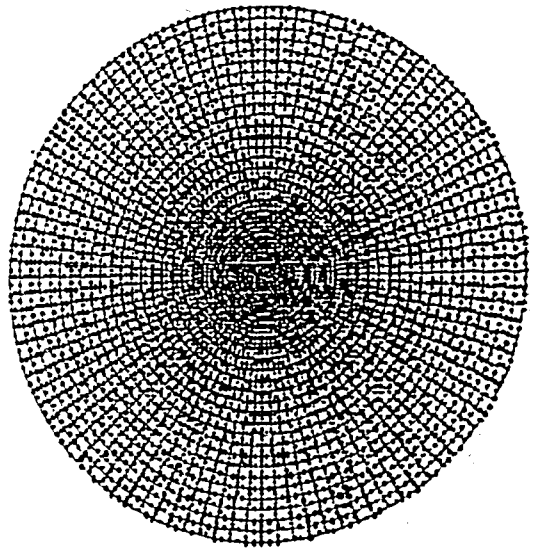
2 / 8 Mesh size



4 / 16 Mesh size



7 / 28 Mesh size

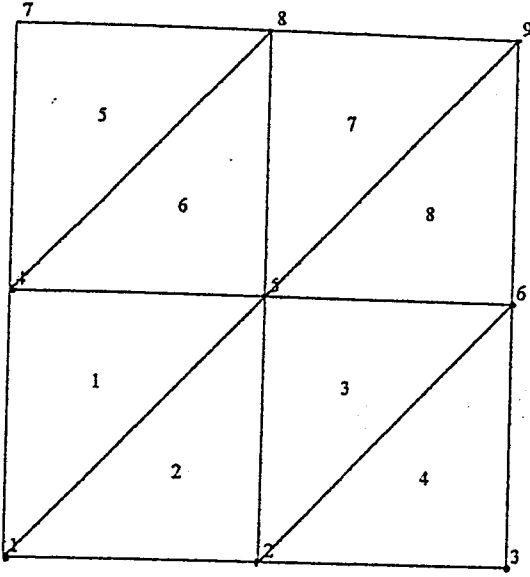


25 / 100 Mesh size

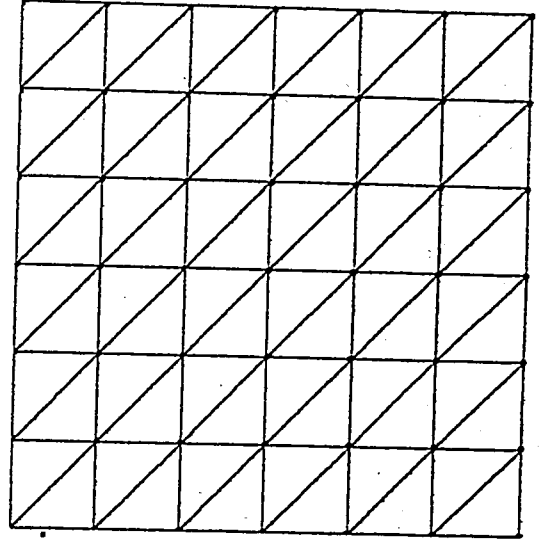
Figure E.4. Mesh sizes of plat bending element (QTF8)

APPENDIX F: RECTANGULAR PLATE BENDING PROBLEM UNDER CENTRAL LOAD

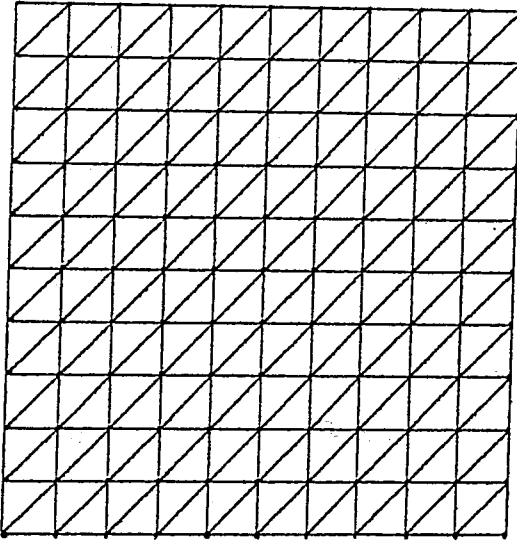
In this section, the mesh sizes, input and output data of the rectangular plate bending problem under central load are included.



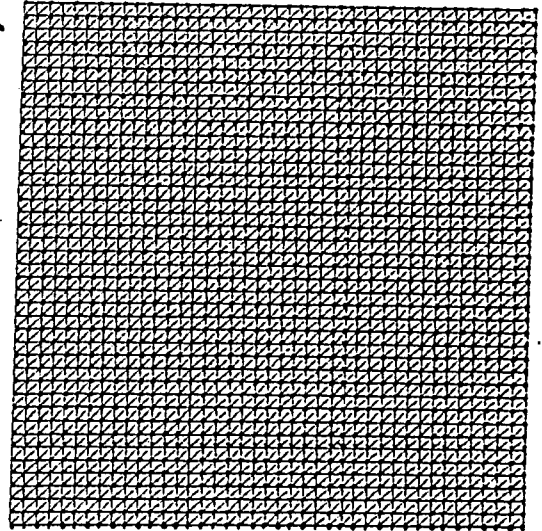
2 / 2 Mesh size



6 / 6 Mesh size

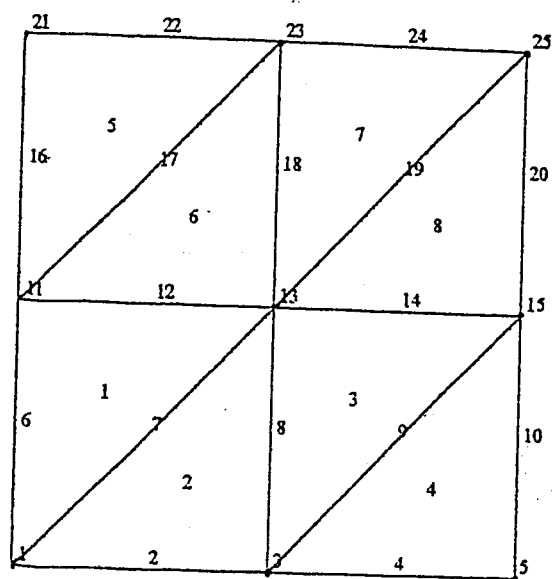


10 / 10 Mesh size

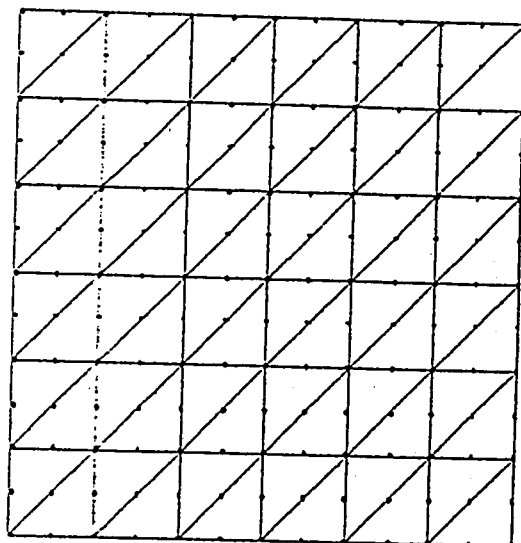


40 / 40 Mesh size

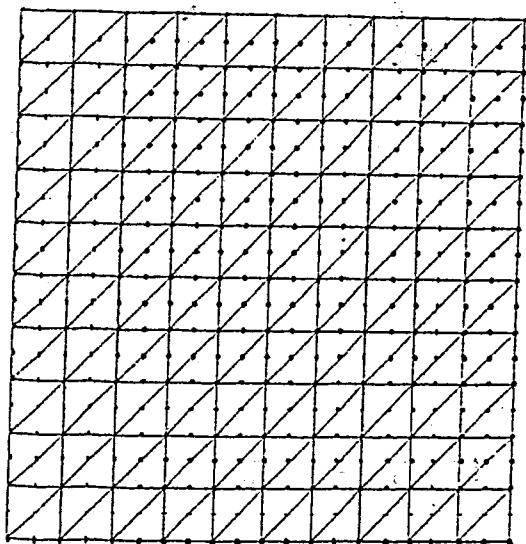
Figure F.1. Mesh sizes of plate bending element (TF3)



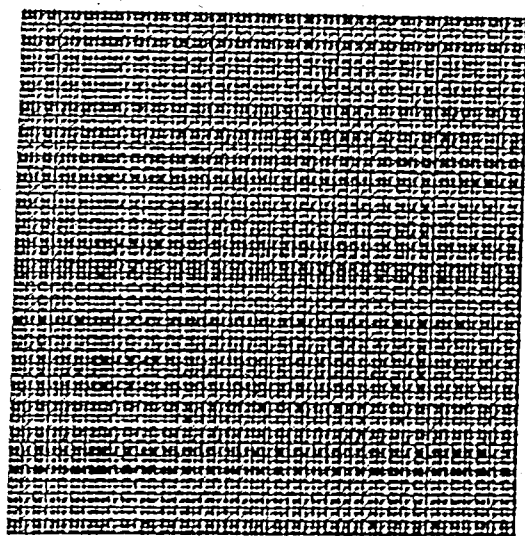
2 / 2 Mesh size



6 / 6 Mesh size



10 / 10 Mesh size



40 / 40 Mesh size

Figure F.2. Mesh sizes of plate bending element (TTF6)

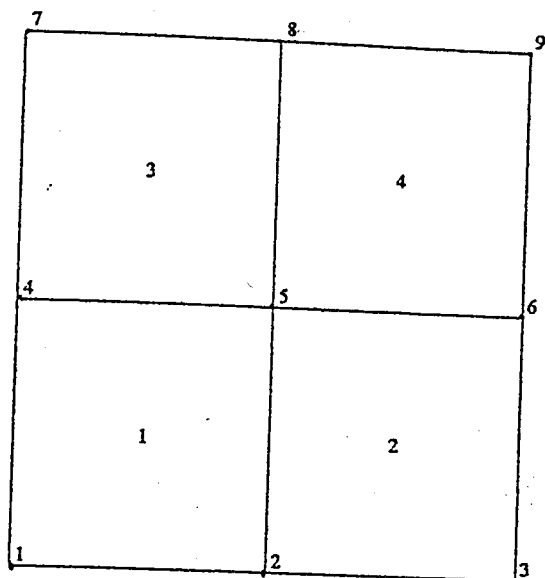
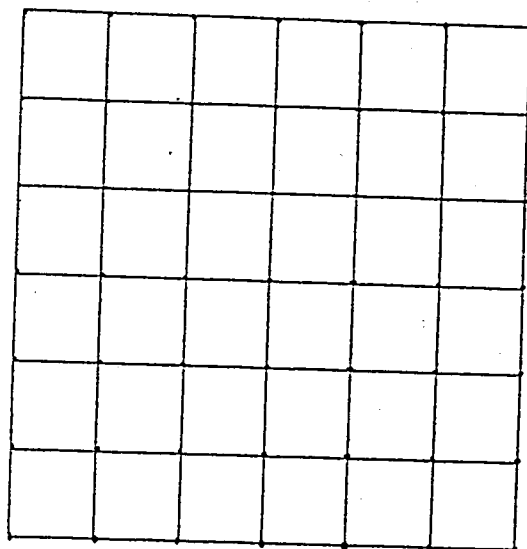
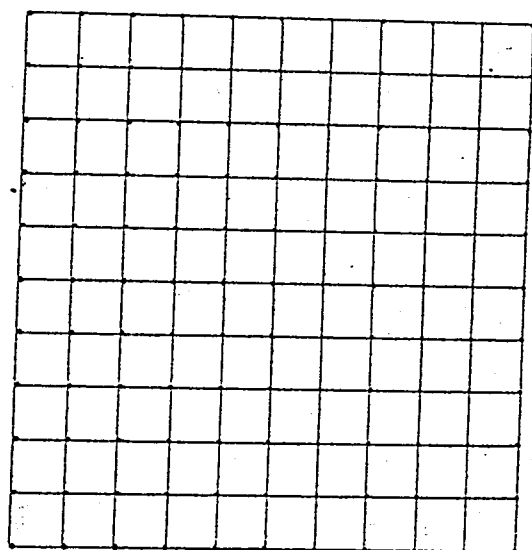
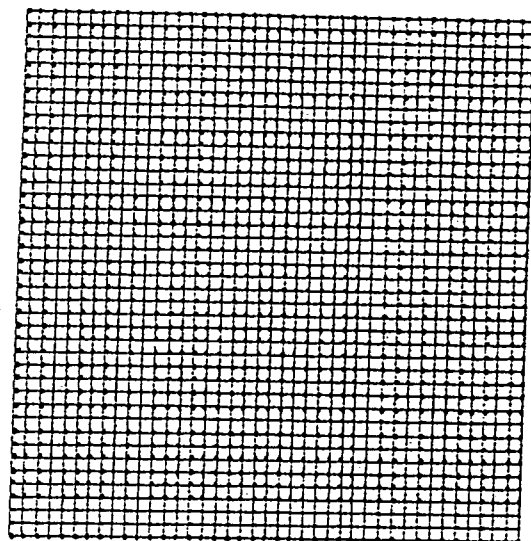
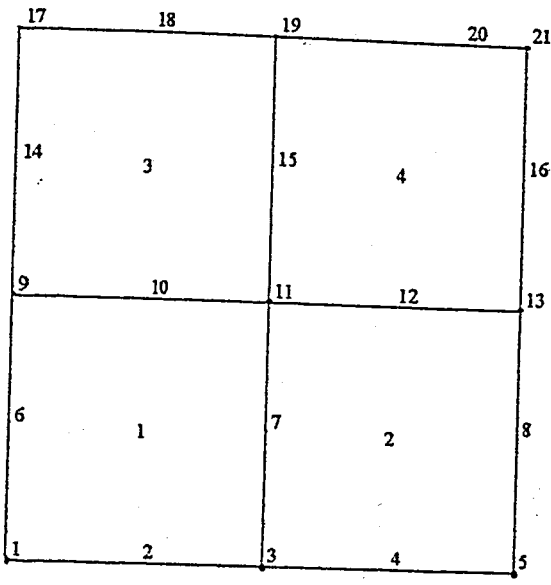
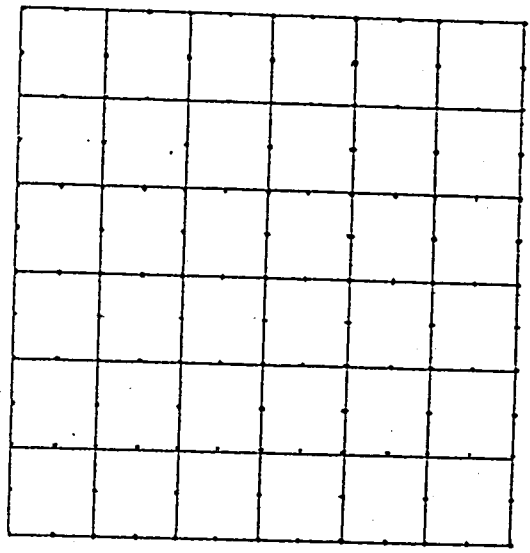
*2 / 2 Mesh size**6 / 6 Mesh size**10 / 10 Mesh size**40 / 40 Mesh size*

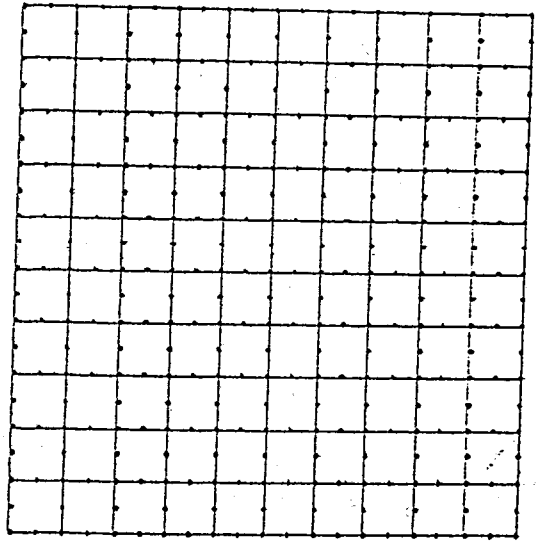
Figure F.3. Mesh sizes of plate bending element (QF4)



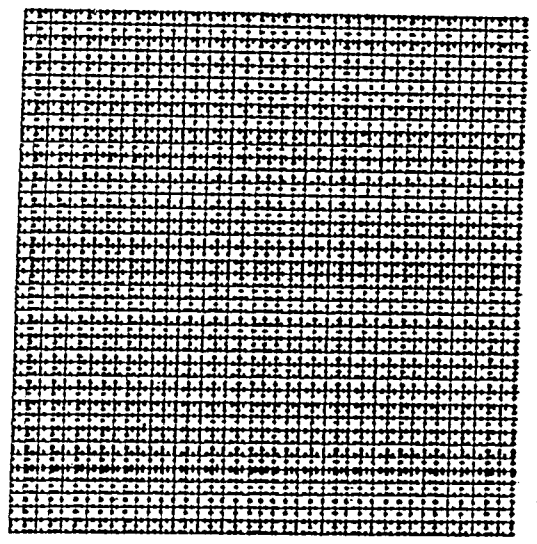
2 / 2 Mesh size



6 / 6 Mesh size



10 / 10 Mesh size



40 / 40 Mesh size

Figure F.4. Mesh sizes of plate bending element (QTF8)

F.1. Input Data for Plate Bending Element QF4 (2/4 mesh size)

C Version 13.4 Date 30-01-02 Time 11:19

PROBLEM TITLE Plate Bending

OPTION 2 48 61 117 118 123 146 157 235 250 ...
253 278 319 324

UNITS kN m KG s C

QF4 ELEMENT TOPOLOGY

1	1	2	5	4
2	2	3	6	5
3	4	5	8	7
4	5	6	9	8

SOLUTION ORDER AUTOMATIC 4

NODE COORDINATES

1	0.000000000E+00	0.000000000E+00	0.000000000E+00
3	4.000000000E+00	0.000000000E+00	0.000000000E+00
7	0.000000000E+00	4.000000000E+00	0.000000000E+00
9	4.000000000E+00	4.000000000E+00	0.000000000E+00

SPACING

FIRST	1	7	3	2*2
INC	2	2	(2)	
FIRST	1	3	1	2*2
INC	3	3	(3)	

QF4 GEOMETRIC PROPERTIES

TITLE Thickness

2	1.400000000E-01	1.400000000E-01	1.400000000E-01 ...
	1.400000000E-01		

GEOMETRIC ASSIGNMENTS

C

C Lusasm Dataset 2 Has Title Thickness

1	4	1	2	2
---	---	---	---	---

MATERIAL PROPERTIES

4	3.000000000E+07	3.000000000E-01	2.548400000E+00 ...
	1.000000000E-05	0.000000000E+00	0.000000000E+00 ...
	0.000000000E+00		

MATERIAL ASSIGNMENTS

C

C Lusasm Dataset 4 Has Title BS25 (elastic)

1	4	1	4	0	0	0	0	4
---	---	---	---	---	---	---	---	---

0

0

FREEDOM TEMPLATE

W THX THY

SUPPORT NODES

1	3	1	R R R
4	0	0	R R R
6	0	0	R R R
7	9	1	R R R

LOAD CASE 1 TITLE Loadcase 1

CL

5	0	0	-1.000000000E+02	0.000000000E+00	0.000000000E+00
---	---	---	------------------	-----------------	-----------------

ELEMENT OUTPUT

1	4	1	0
---	---	---	---

NODE OUTPUT

1	9	1	0
---	---	---	---

PLOT FILE

END

F.2. Displacement Components for Plate Bending Element QF4 (2 / 4 mesh size)

LINEAR/DYNAMIC ANALYSIS

Current Results File = C:\Lusas13\Projects\qf4-disp.mys ID 0

Current Selected Load ID = 1

"Loadcase 1"

Displacement Components In Global Axes

Node	DZ	THX	THY	RSLT
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
5	-0.1327E-02	0.1011E-18	0.5054E-19	0.1327E-02
6	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
7	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
8	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
9	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

F.3. Moments for Plate Bending Element QF4 (2 / 4 mesh size)

LINEAR/DYNAMIC ANALYSIS

Current Results File = C:\Lusas13\Projects\qf4-momonet.mys ID 0

Current Selected Load ID = 1

"Loadcase 1"

Averaged Stress Resultant Components In Global Axes

Node	MX	MY	MXY
1	0.7754E-09	0.7754E-09	1.750
2	4.500	15.00	0.4441E-15
3	0.7754E-09	0.7754E-09	-1.750
4	15.00	4.500	0.6661E-15
5	-19.50	-19.50	0.3331E-15
6	15.00	4.500	-0.7772E-15
7	0.7754E-09	0.7754E-09	-1.750
8	4.500	15.00	0.8882E-15
9	0.7754E-09	0.7754E-09	1.750

APPENDIX G: MESH SIZES OF RECTANGULAR PLATE BENDING PROBLEM UNDER UNIFORM LOAD

The geometry of the mesh sizes of plate bending elements TF3, TTF6, QF4 and QTF8 are shown.

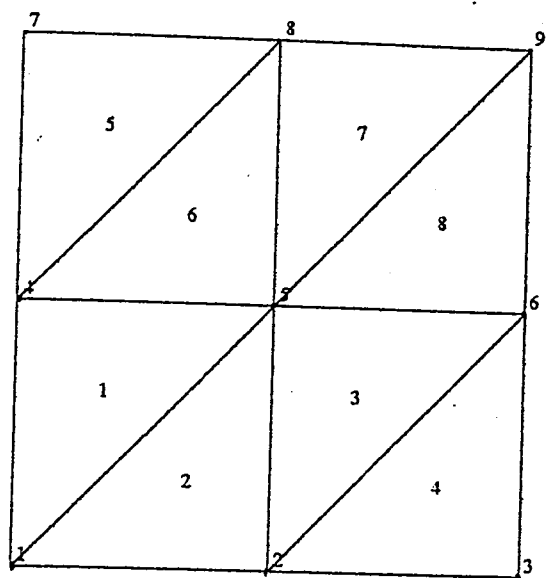
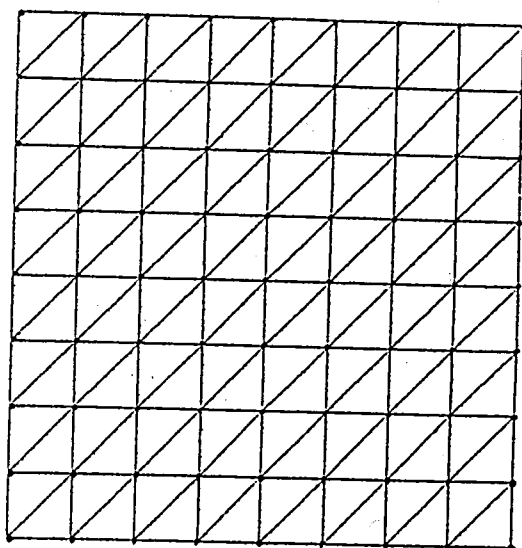
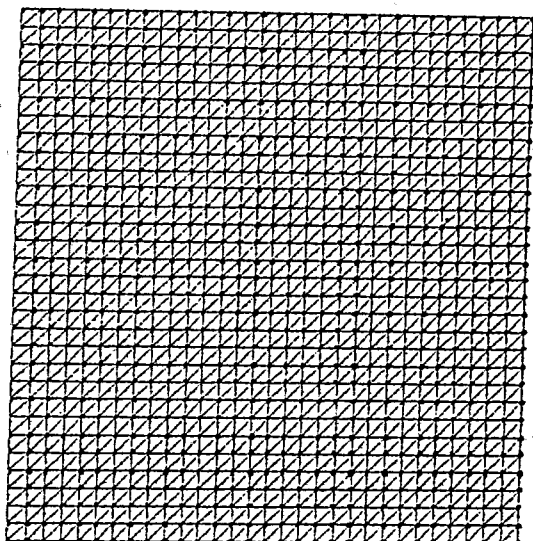
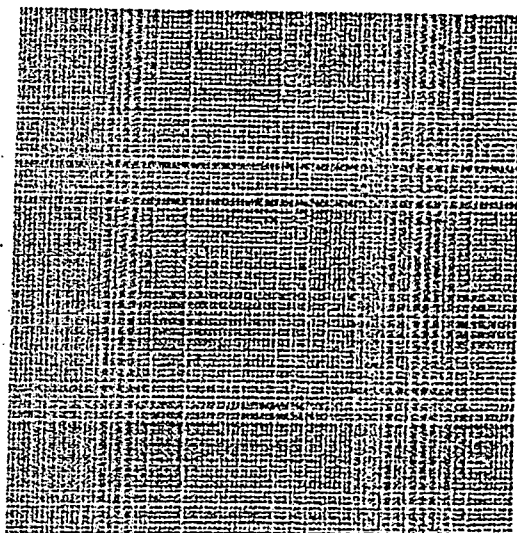
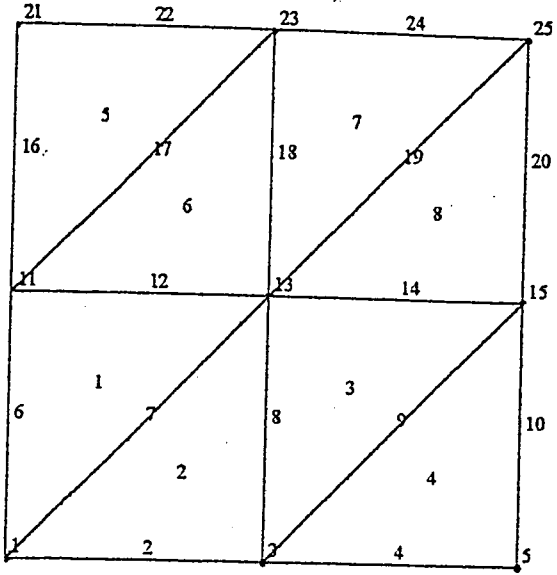
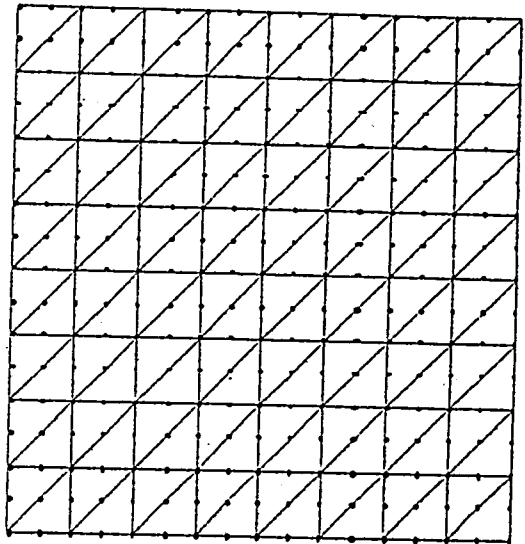
*2 / 2 Mesh size**8 / 8 Mesh size**30 / 30 Mesh size**100 / 100 Mesh size*

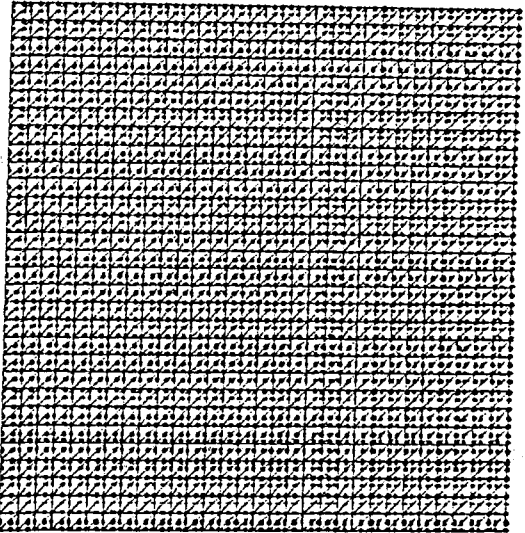
Figure G.1. Mesh sizes of plate bending element (TF3)



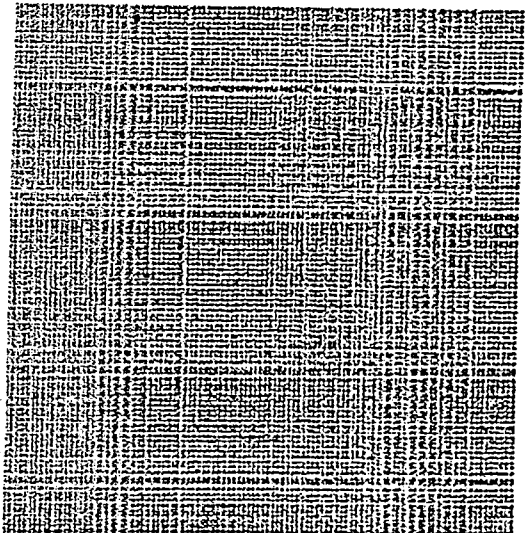
2 / 2 Mesh size



8 / 8 Mesh size

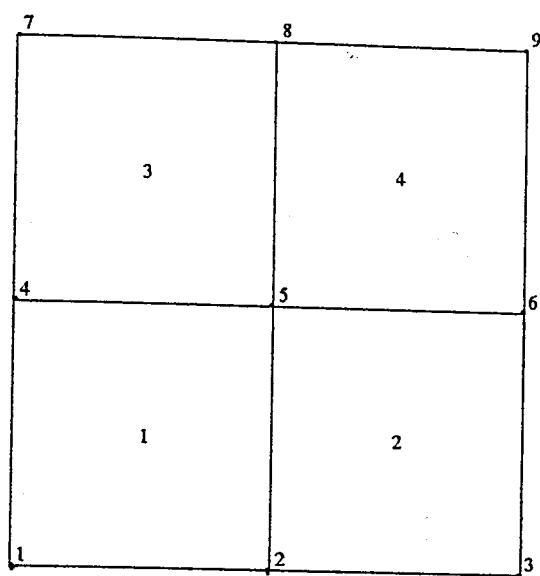
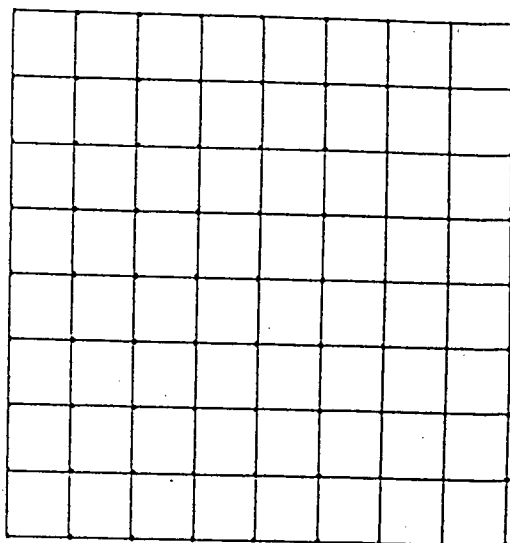
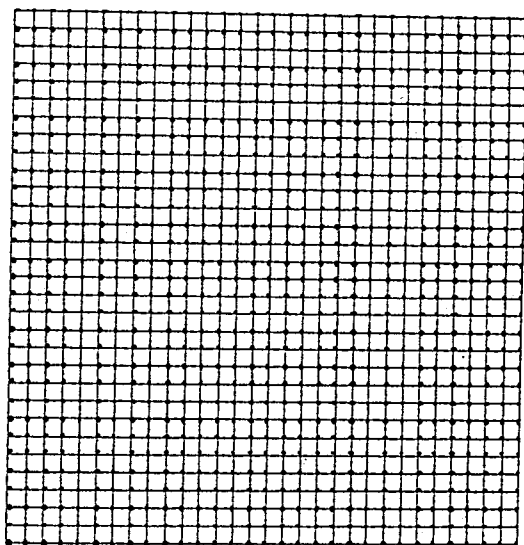
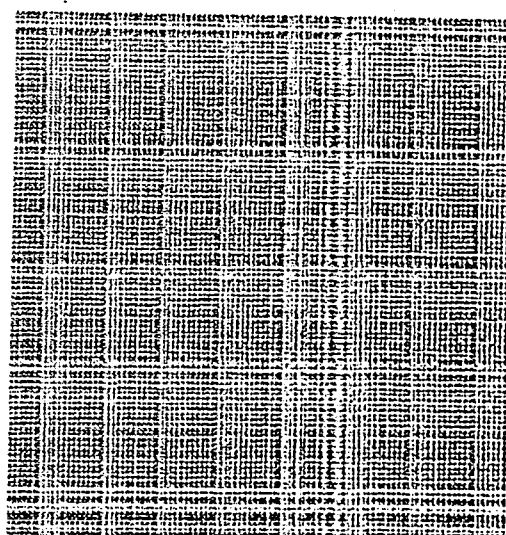


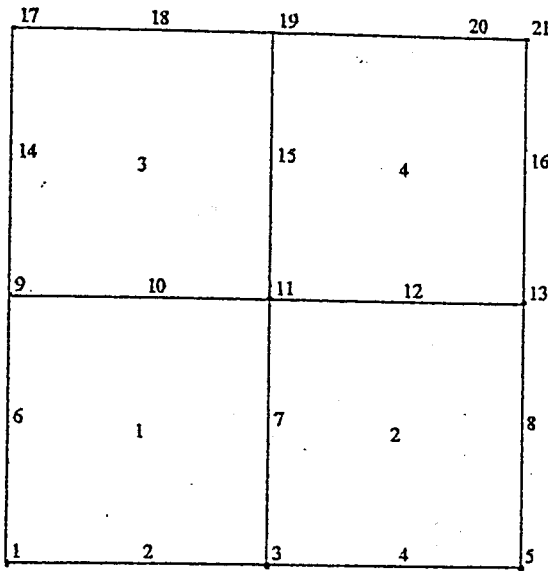
30 / 30 Mesh size



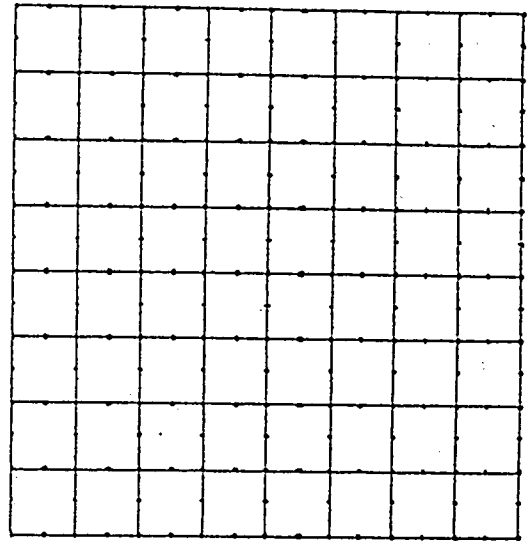
100 / 100 Mesh size

Figure G.2. Mesh sizes of plate bending element (TTF6)

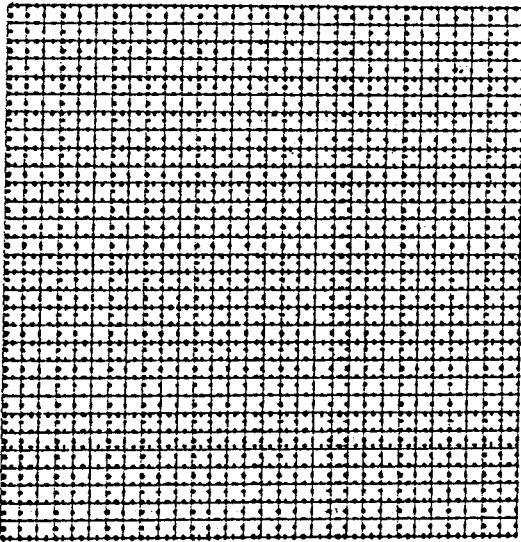
*2 / 2 Mesh size**8 / 8 Mesh size**30 / 30 Mesh size**100 / 100 Mesh size***Figure G.3. Mesh sizes of plate bending element (QF4)**



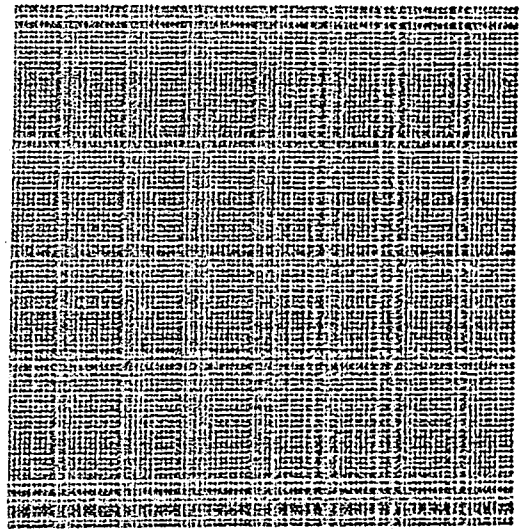
2 / 2 Mesh size



8 / 8 Mesh size



30 / 30 Mesh size



100 / 100 Mesh size

Figure G.4. Mesh sizes of plate bending element (QTF8)

REFERENCES

1. Rao, S. S., *The Finite Element Method in Engineering*, Pergamon Press, Oxford, 1989.
2. Ottosen, N. and H. Petterson, *Introduction to the Finite Element Method*, Prentice Hall, UK, 1992.
3. Chandrupatla, T. R. and A. D. Belegundu, *Introduction to Finite Elements in Engineering*, 2nd edu., Prentice Hall, New Jersey, 1997.
4. Hrenikoff, A., "Solution of Problems in Elasticity by the Framework Method", *Journal of Applied Mechanics*, Vol. 8, pp. 169-175, 1941.
5. Courant, R., "Variational Methods for the Solution of Problems of Equilibrium and Vibrations", *Bulletin of American Mathematical Society*, Vol. 49, pp. 1-23, 1943.
6. Turner, M. J., R. W. Clough, H. C. Martin and L. J. Topp, "Stiffness and Deflection Analysis of Complex Structures", *Journal of Aeronautical Sciences*, Vol. 23, pp. 805-824, 1956.
7. Clough, R. W., "The Finite Element Method in Plane Stress Analysis", *Proceeding of the second ASCE Conference on Electronic Computation*, Pittsburgh, September 1960, pp. 345-379, 1960.
8. Argyris, J. H. and S. Kelsey, *Energy Theorems and Structural Analysis*, Butterworth, London, 1965.
9. Zienkiewicz, O. C. and Y. K. Cheung, *The Finite Element Method in Structural and Continuum Mechanics*, McGraw-Hill, London, 1967.
10. Oden, J. T., *Finite Elements of Nonlinear Continua*, McGraw-Hill, New York, 1972.

11. Buchanan, G. R., *Schaum's outline of Theory and Problems of Finite Element Analysis*, McGraw-Hill, New York, 1995.
12. Bathe, K. J., *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, New Jersey, 1982.
13. Bathe, K. J., *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, New Jersey, 1996.
14. Tezcan, S. S., *Lecture Notes on Finite Element Method*, Boğaziçi University, 2000.
15. Cook R. D., *Concepts and Applications of Finite Element Analysis*, John Wiley, Canada, 1971.
16. Tenchev, R. T., "A Study of the Accuracy of some FEM stress recovery schemes for 2D Stress Concentration Problems", *Finite Elements in Analysis and Design*, Vol. 29, pp.105-119, 1998.
17. Pourazady, M. and M. Radhakrishnan, "Optimization of a Triangular Mesh", *Computers & Structures*, Vol. 40, No. 3, pp.795-804, 1991.
18. Martinez, R. and A. Samartin, "Two-dimensional Mesh Optimization in the Finite Element Method", *Computers & Structures*, Vol. 40, No. 5, pp. 1169-1175, 1991.
19. Peraire, J., M. Vahdati, K. Morgan and O. C. Zienkiewicz, "Adaptive Remeshing for Compressible Flow Computations", *Journal of Computational Physics*, Vol. 72, pp. 449-466, 1987.
20. Zhu, J. Z., E. Hinton and O. C. Zienkiewicz, "Adaptive Finite Element Analysis with Quadrilaterals", *Computers & Structures*, Vol. 40, No. 5, pp. 1097-1104, 1991.
21. Segerlind, L. J., *Applied Finite Analysis*, John Wiley & Sons, Inc., New York, 1976.

22. Zienkiewicz, O. C., *The Finite Element Method in Engineering Science*, 2nd edu., McGraw-Hill, London, 1971.
23. Smith, I. M., "Are there any new elements?", in E. Onate, J. Periaux and A. Samuelsson, *The Finite Element Method in the 1990's*, pp. 109-118, Artes Gráficas Torres S.A., Barcelona, 1991.
24. Cook R. D., *Finite Element Modeling for Stress Analysis*, John Wiley, U.S.A., 1995.
25. Hiller, J. F., and K. J. Bathe, "On Higher-order-accuracy Points in Isoparametric Finite Element Analysis and an Application to Error Assessment", *Computers & Structures*, Vol. 79, pp. 1275-1285, 2001.
26. Utku, M., "An Improved Transformation for Universal Serendipity Elements", *Computers & Structures*, Vol. 73, pp. 199-206, 1999.
27. Rathod, H.T. and Md. Shafiqul Islam, "Some Pre-computed Universal Numeric Arrays for Linear Convex Quadrilateral Finite Elements", *Finite Elements in Analysis and Design*, Vol. 38, pp. 113-136, 2001.
28. Baker, A. J., Pepper, D. W., *Finite Elements 1-2-3*, McGraw-Hill, New York, 1991.
29. Timoshenko, S.P. and J. N. Goodier, *Theory of Elasticity*, McGraw-Hill, New York, 1970.
30. *LUSAS Finite Element System, Version 13.4 LUSAS Theory Manual 2*, FEA Ltd. Surrey, England.
31. *LUSAS Finite Element System, Version 13.4 LUSAS Element Library*, FEA Ltd. Surrey, England.
32. Timoshenko, S. and S. Woinowsky-Krieger, *Theory of Plates and Shells*, McGraw-Hill, New York, 1959.

REFERENCES NOT CITED

- Chucheepsakul, S. and B. Chinnaboon, "An Alternative Domain/Boundary Element Technique for Analyzing Plates on Two-parameter Elastic Foundation", *Engineering Analysis Boundary Elements*, Vol. 26, pp. 547-555, 2001.
- Frostig, Y., O. T. Thomsen and F. Mortensen, "Analysis of Adhesive-Bonded Joints, Square-End, and Spew-Fillet – Higher Order Theory Approach", *Journal of Engineering Mechanics*, Vol. 125, No. 11, November, 1999.
- Generawicz, J. M., B. P. Hiett, S. J. Cox, M. Molinari, O. Beckett and K. S. Thamos, "Finite Element Modeling of Photonic Crystals", *Fourth International Conference on Computation in Electromagnetics*,
<http://www.ecs.soton.ac.uk/~bph99r/html/photronics.htm>
- Mohr, G. A., "The Finite Patch Method: A nodal Equation Method based on FEM", *Advances in Engineering Software*, Vol. 32, pp. 327-335, 2001.
- Roberts, S., "Combining Higher Order Finite Volume and Higher Order Finite Element Methods to Simulate Fluid Flow in Geologic Media", *The Tenth Biennial Computational Techniques and Applications Conference*, Australia, 16-18 July 2001,
<http://conference.maths.uq.edu.au/ctac2001/abstaracts/StephenRoberts1.html>
- Shah, M. S. and T. Kant, *Finite Element Analysis of Fibre Reinforced Polymer Shells Using Higher-order Shear Deformation Theories on Parallel Distributed Memory Machines*, <http://bitscap.bits-pilani.ac.in/param1apps/strucmech-reprint.htm>

