Development of FE-meshfree hybrid methods and their application to static and free vibration problems in 2-D solid methanics

Zhang, Bingru

2008

Zhang, B. R. (2008). Development of FE-MEeshfree hybrid methods and their application to static and free vibration problems in 2-D solid methanics. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/5598>

<https://doi.org/10.32657/10356/5598>

Nanyang Technological University

Downloaded on 26 Apr 2023 22:14:24 SGT

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**DEVELOPMENT OF FE-MESHFREE HYBRID
METHODS AND THEIR APPLICATION TO STATIC
AND FREE VIBRATION PROBLEMS IN 2-D SOLID
MECHANICS**

**ZHANG BINGRU
SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING
2008**

Development of FE-Meshfree Hybrid Methods and Their Application to Static and Free Vibration Problems in 2-D Solid Mechanics

Zhang Bingru

School of Mechanical and Aerospace Engineer

A thesis submitted to the Nanyang Technological University

in FULFILMENT of the requirement for the degree of Ph.D in Engineer

2008

ABSTRACT

Meshfree methods have been extensively investigated in recent years due to their flexibility in solving practical engineering problems. As for example, they do not require a mesh to discretize the problem domain, and the approximate solution is constructed entirely in terms of a set of scattered nodes. However, meshfree methods demand a high computational effort as compared to the well established finite element (FE) method. And establishing nodal connectivity in meshfree methods is relatively difficult. Furthermore, implementing the displacement boundary conditions is cumbersome in many meshfree methods due to the lack of Kronecker delta property of the meshfree shape functions. On the other hand, the finite element method has no such difficulty, and is well established and has been widely used in engineering. Nevertheless, the finite element method generally gives less accurate results compared to meshfree methods, more so under distorted meshes. The focus of this thesis is on the development of hybrid methods that aim at synergising the merits of FE and meshfree methods and its application to 2D solids.

To begin with, a FE-EFG method is proposed, which is based on the local Petrov-Galerkin approach. The method uses FE shape functions as test functions and Element Free Galerkin (EFG) shape functions as trial functions. Numerical testing reveals that the method has a good tolerance to mesh distortion but poses difficulty in applying the displacement boundary conditions as experienced in EFG method. To solve this problem, a new hybrid method called FE-PIM is developed. For the FE-PIM method, the FE shape functions and the Point Interpolation Method (PIM) shape functions are

used as trial and test functions, respectively. The FE-PIM method can handle the displacement boundary conditions easily. However, the choice of suitable polynomial basis for trial function is a tricky problem which is inherited from PIM. If an inappropriate polynomial basis is chosen, it leads to interpolation failure. Furthermore, the results given by the FE-PIM are not as accurate as expected.

In order to remove the above drawbacks of FE-PIM, a new hybrid method called FE-LSPIM is proposed. This method employs the concepts of partition of unity finite element methods (PU-FEM), wherein the shape functions of finite element are used for PU and that of least-square point interpolation method (LSPIM) for local approximation. Thereby the proposed method inherits the completeness properties of meshfree shape functions and compatibility properties of finite element shape functions. A QUAD4 element (called FE-LSPIM QUAD4) is developed based on this approach and its performance is studied for typical benchmark problems. The results show that this element performs better than the classical isoparametric QUAD4 element and comparable to meshfree method (EFG). In particular, the proposed element has good tolerance to mesh distortions.

Next, the FE-LSPIM QUAD4 is extended to free vibration problems. Numerical examples for free vibration analyses are presented and the results are compared with those obtained using other existing solution techniques. The studies indicate that the accuracy of FE-LSPIM QUAD4 is generally better than other methods. Also, the proposed element is more tolerant to mesh distortion than the classical QUAD4 element.

As a further extension, FE-LSPIM QUAD4 element is applied to geometrically nonlinear problems. The geometrical nonlinear formulation is derived based on the updated Lagrangian approach. The performance of the element is studied for several numerical examples. The element is seen to give better results than the QUAD4 element of ANSYS under distorted mesh.

Although the FE-LSPIM satisfies the objective of developing a method that synergises the FEM and meshfree methods, it does have demerits. Singularity of interpolation matrix, shear locking tendency and computational efficiency are some of them that are critically reviewed at the end of the thesis.

Keywords: Meshfree, hybrid, FE-EFG, FE-PIM, FE-LSPIM

ACKNOWLEDGEMENTS

The completion of this project would be impossible without the encouragement and help of my supervisor Assoc. Prof. Sellakkutti Rajendran. I would like to express my sincere appreciation and thanks to my supervisor, for his meticulous concerns and patient guidance, advice, information and inspiration throughout the progress of this project.

Thanks to all research staff and graduate students at CANES, for their assistance and enlightening discussions.

The scholarship offered by Nanyang Technological University is gratefully acknowledged.

I would also like to thank my dear friends. I really appreciate the time we spend together.

Finally, I would like to deeply appreciate my family for their love, support and concerns.

Zhang Bingru

Nov 2006

TABLE OF CONTENTS

ABSTRACT	1
TABLE OF CONTENTS	V
LIST OF FIGURES.....	XII
LIST OF TABLES.....	XXI
LIST OF SYMBOLS.....	XXVI
CHAPTER 1. INTRODUCTION.....	1
1.1. MOTIVATION	1
1.2. OBJECTIVE AND SCOPE	3
1.3. ORGANIZATION OF THESIS	4
CHAPTER 2. LITERATURE REVIEW.....	9
2.1. INTRODUCTION	9
2.2. REVIEW OF FINITE ELEMENT METHODS	10
2.2.1. REDUCED INTEGRATION AND STABILIZATION METHOD.....	11
2.2.2. INCOMPATIBLE MODES APPROACH.....	13
2.2.3. MULTI-FIELDS ELEMENTS	16
2.2.4. ENHANCED ASSUMED STRAIN APPROACH.....	20
2.2.5. ELEMENTS WITH DRILLING DEGREES OF FREEDOM	21

2.2.6. AREA COORDINATES ELEMENTS	23
2.2.7. UNSYMMETRIC ELEMENT	26
2.2.8. SUMMARY OF 4-NODE ELEMENTS	27
2.3. REVIEW MESHFREE METHODS	28
2.3.1. THE EARLY DEVELOPMENTS OF MESHFREE METHODS	29
2.3.2. ELEMENT FREE GALERKIN METHOD	30
2.3.3. THE NATURAL ELEMENT METHOD	32
2.3.4. REPRODUCING KERNEL PARTICLE METHOD	33
2.3.5. MESHFREE LOCAL PETROV-GALERKIN METHOD AND A LOCAL BOUNDARY INTEGRAL EQUATION METHOD	35
2.3.6. THE POINT INTERPOLATION METHODS [PIMs]	39
2.3.7. A POINT COLLOCATION METHOD	41
2.3.8. RECENT DEVELOPMENTS OF MESHFREE METHOD	42
2.4. COUPLING OF FINITE ELEMENTS WITH MESHFREE METHODS	44
2.5. MOTIVATION FOR THE PRESENT WORK	57
CHAPTER 3. TWO HYBRID FE-MESHFREE METHODS: FE-EFG AND FE- PIM	59
3.1. INTRODUCTION	59
3.2. FE-EFG METHOD	61
3.2.1. EFG SHAPE FUNCTIONS	62
3.2.1.1. EFG shape functions in one dimension	62
3.2.1.2. EFG shape functions in two dimensions	64
3.2.2. CHOICE OF WEIGHT FUNCTIONS	65

3.2.2.1. Weight function in one dimension	65
3.2.2.2. Weight function in two dimensions.....	65
3.2.3. PARAMETRIC SHAPE FUNCTIONS.....	66
3.2.3.1. FE Parametric shape functions in one dimensions.....	66
3.2.3.2. FE Parametric shape functions in two dimensions.....	67
3.2.4. STIFFNESS MATRIX AND LOAD VECTORS IN ONE DIMENSION ..	68
3.2.5. DISCRETE EQUATION IN TWO DIMENSIONS	70
3.2.6. ONE DIMENSIONAL NUMERICAL EXAMPLE	73
3.2.7. TWO DIMENSIONAL NUMERICAL EXAMPLES	79
3.2.7.1. A cantilever beam under pure bending.....	79
3.2.7.2. A cantilever beam under shear force.....	84
3.3. A FE-PIM METHOD	91
3.3.1. PIM SHAPE FUNCTIONS.....	91
3.3.2. DISCRETE EQUATION IN TWO DIMENSIONS	93
3.3.3. THE ALGORITHM OF PRESENT METHOD	96
3.3.4. TEST PROBLEMS	97
3.3.4.1. A cantilever beam under a pure bending moment.....	97
3.3.4.2. A cantilever beam under a shear force at the tip	101
3.4. CONCLUSION.....	105

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE

FUNCTIONS AND LEAST-SQUARE PIM SHAPE FUNCTIONS106

4.1. INTRODUCTION	107
4.2. FORMULATION OF FE-LSPIM QUAD4.....	111
4.2.1. SHAPE FUNCTIONS FOR THE FE-LSPIM QUAD4.....	111
4.2.2. DERIVATION OF NODAL SHAPE FUNCTION ARRAY, Φ_i	115

4.2.3. CONSTRUCTION OF SHAPE FUNCTION MATRIX, Φ_i , Φ AND Ψ	121
4.2.3.1. Construction of Φ_i	122
4.2.3.2. Assembly of shape functions Φ using Φ_i	123
4.2.3.3. Construction of shape functions Ψ	123
4.2.4. USEFUL PROPERTIES OF SHAPE FUNCTIONS	123
4.2.4.1. Kronecker-delta property of Φ_i	123
4.2.4.2. Kronecker-delta property of Ψ	124
4.2.4.3. Inter-element compatibility property of Ψ	125
4.2.4.4. Completeness (reproducibility) property of Ψ	126
4.2.5. FE-LSPIM QUAD4 SHAPE FUNCTION DERIVATIVES	129
4.2.6. FE-LSPIM QUAD4 STRAIN AND STRESS EXPRESSIONS	130
4.2.7. FE-LSPIM QUAD4 STIFFNESS MATRIX AND LOAD VECTORS	131
4.2.8. PSEUDO CODE FOR IMPLEMENTING THE FE-LSPIM QUAD4 ELEMENT	132
4.3. THE FORMULATION OF FE-LSPIM QUAD4U ELEMENT	133
4.3.1. TEST FUNCTIONS AND DERIVATIVE MATRIX OF FE-LSPIM QUAD4U	134
4.3.2. FE-LSPIM QUAD4U ELEMENT STIFFNESS MATRIX AND LOAD VECTORS	135
4.4. TEST PROBLEMS	136
4.4.1. PATCH TEST	136
4.4.1.1. Constant strain/stress patch test	138
4.4.1.2. The linear strain patch tests in the strong form	140
4.4.1.3. Quadratic strain patch tests	148
4.4.2. COOK'S SKEW BEAM PROBLEM	155
4.4.3. THICK CURVED BEAM	157

4.4.4. MACNEAL’S THIN CANTILEVER BEAM	158
4.4.5. A CANTILEVER BEAM SUBJECTED TO A PURE TIP MOMENT	160
4.4.6. A CANTILEVER BEAM SUBJECT TO PARABOLICALLY DISTRIBUTED TIP SHEAR FORCE.....	165
4.4.7. A BEAM SUBJECT TO MID-SPAN POINT LOAD	172
4.4.8. A PLATE WITH A CENTRAL HOLE	173
4.5. CONCLUDING REMARKS.....	182

CHAPTER 5. FE-LSPIM QUAD4 ELEMENT FOR FREE VIBRATION

ANALYSIS	184
5.1. INTRODUCTION	184
5.2. THE EIGENVALUE PROBLEM FOR FREE VIBRATION ANALYSIS	187
5.3. PSEUDO CODE FOR IMPLEMENTING FE-LSPIM QUAD4 FOR THE FREE VIBRATION ANALYSIS.....	187
5.4. NUMERICAL EXAMPLES.....	189
5.4.1. A CANTILEVER BEAM	189
5.4.2. A SHEAR WALL	197
5.4.3. DISTORTION SENSITIVITY TEST.....	199
5.4.4. RAYLEIGH-LAMB PROBLEM.....	200
5.5. CONCLUDING REMARKS.....	202

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR

ANALYSIS	203
6.1. INTRODUCTION	203

6.2. NOTATION.....	205
6.3. UPDATED LAGRANGIAN FORMULATION	207
6.3.1. PRINCIPLE OF VIRTUAL DISPLACEMENTS	207
6.3.2. CONSTITUTIVE EQUATIONS.....	209
6.3.3. UPDATED LAGRANGIAN FORMULATION	210
6.3.4. FINITE ELEMENT MODEL FOR TWO-DIMENSIONAL CONTINUA	213
6.3.5. FLOWCHART OF THE COMPUTER IMPLEMENTATION	217
6.4. NUMERICAL EXAMPLES.....	218
6.4.1. CANTILEVER BEAM UNDER SHEAR FORCE	218
6.4.2. CLAMPED BEAM UNDER CENTRAL POINT FORCE	223
6.4.3. A COLUMN UNDER AN ECCENTRIC AXIAL COMPRESSIVE LOAD	226
6.4.4. THRUST BEAM UNDER CENTRAL LOAD	231
6.4.5. COOK’S SKEW BEAM.....	234
6.4.6. DISTORTION SENSITIVITY TEST.....	236
6.5. CONCLUDING REMARKS.....	238
CHAPTER 7. CRITICAL ASSESSMENT OF FE-LSPIM QUAD4 ELEMENT.	239
7.1. MESH DISTORTION TOLERANCE.....	239
7.2. LINEAR INDENDENCE PROBLEM	240
7.2.1 EIGENVALUE ANALYSIS	241
7.3. SINGULARITY PROBLEM.....	243

7.4 LOCKING PROBLEM.....	246
7.5 THE INF-SUP TEST	248
7.5.1. TEST PROBLEM 1	250
7.5.2. TEST PROBLEM 2	253
7.5.3. TEST PROBLEM 3	255
7.5.4. TEST PROBLEM 4	256
7.6 COMPUTATIONAL EFFICIENCY	258
7.7. RELATIVE MERITS OF PRESENT METHOD (FE-LSPIM), FEM AND MESHFREE METHODS	258
7.8. CONCLUSIONS	259
CHAPTER 8. CONCLUDING REMARKS	261
8.1. SUMMARY	261
8.2. MAJOR CONCLUSIONS	261
8.3. FUTURE WORK.....	265
APPENDIX: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM	266
LIST OF PUBLICATIONS	284
REFERENCES	285

LIST OF FIGURES

Figure 1.1. Structure of the thesis.....	5
Figure 1.2. The methods and the elements developed in this thesis.....	6
Figure 2.1. A four node quadrilateral element by Long [63].	25
Figure 2.2. Plots of a typical MLS shape function and its derivatives in two dimensions [86].	31
Figure 2.3. Shape functions in a 2D regular node distribution: (a) MFEM; (b) MLSQ, SPH, RKPM; (c) NEM (Sergio [91]).	33
Figure 2.4. The support of the MLPG [100].....	37
Figure 2.5. Domain handled by EFG method and FE method [128].....	48
Figure 2.6. Node-based finite element data structures [143].....	56
Figure 3.1. The influence domain of test and trial functions.....	61
Figure 3.2. One dimensional example problem: a bar of unit length subjected to a linear body force and fixed at the point $x = 0$	74
Figure 3.3. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 2.	76
Figure 3.4. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 3.	77

Figure 3.5. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 4.	78
Figure 3.6. A cantilever beam under pure bending.	79
Figure 3.7. The contour plots of σ_x for the cantilever beam under pure bending (Regular mesh).	82
Figure 3.8. The contour plots of σ_x for the cantilever beam under pure bending (Distorted mesh).	83
Figure 3.9. A cantilever beam subjected to a shear force.	84
Figure 3.10. Variation of the percent error of displacement with different scaling parameter $d_{\max}$.	86
Figure 3.11. Comparison of normal stress and shear stress at the centre of the beam ($x = 24$).	87
Figure 3.12. The contour plots of σ_x for cantilever beam under tip shear force (Regular mesh).	89
Figure 3.13. The contour plots of σ_x for cantilever beam under tip shear force (Distorted mesh).	90
Figure 3.14. Domain of influence of FE-PIM method.	94
Figure 3.15. Domain of influence of PIM* method.	94
Figure 3.16. Domain of influence of PIM method.	94

Figure 3.17. The contour plots of σ_x for cantilever beam under pure bending (Regular mesh).	99
Figure 3.18. The contour plots of σ_x for cantilever beam under pure bending (Distorted mesh).	100
Figure 3.19. The contour plots of σ_x for cantilever beam under tip shear force (Regular mesh).	103
Figure 3.20. The contour plots of σ_x for cantilever beam under tip shear force (Distorted mesh).	104
Figure 4.1. Definition of nodal support of element 1-2-3-4.	113
Figure 4.2. Element support, $\Omega = \Omega_1 \oplus \Omega_2 \oplus \Omega_3 \oplus \Omega_4$	113
Figure.4.3. Definition of nodal support for boundary nodes 5 and 6 considering <i>first order nodal connectivity</i> , i.e., nodes of all elements connected to a given node.....	117
Figure.4.4. Definition of nodal supports for boundary nodes 5 and 6 considering. <i>second order nodal connectivity</i> , i.e., nodes of all elements connected to the nodes of all elements connected to a given node.....	118
Figure 4.5. The support of element 1.....	122
Figure 4.6. Type A patch test.	138
Figure 4.7. Type B patch test.....	138
Figure 4.8. Type C patch test.....	138
Figure 4.9. Constant strain patch test by Chen et al [66].....	139

Figure 4.10. Patch considered for patch test; (a) regular mesh; (b) distorted mesh.	141
Figure 4.11. The contour plot of linear strain type B patch test for regular mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.....	144
Figure 4.12. The contour plot of linear strain type B patch test for distorted mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.	145
Figure 4.13. The contour plot of linear strain type C patch test for regular mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.....	146
Figure 4.14. The contour plot of linear strain type C patch test for distorted mesh (a) u displacement; (b) v displacement; (c) σ_x stress.	147
Figure 4.15. The contour plot of quadratic strain Type B patch test for regular mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.....	151
Figure 4.16. The contour plot of quadratic strain Type B patch test for distorted mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.....	152
Figure 4.17. The contour plot of quadratic strain Type C patch test for regular mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.....	153
Figure 4.18. The contour plot of quadratic strain Type C patch test for distorted mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.....	154
Figure 4.19. Cook's skew beam problem.	155
Figure 4.20. Bending of thick curved beam.	158

Figure 4.21. MacNeal's beam subjected to (1) unit shear force (2) constant moment.	159
Figure 4.22. Contour plot of σ_x for pure tip moment problem under regular mesh.	163
Figure 4.23. Contour plot of σ_x for pure tip moment problem under distorted mesh.....	164
Figure 4.24. The contour plot of σ_x for tip shear problem under regular mesh. .	168
Figure 4.25. The contour plot of σ_x for tip shear problem under distorted mesh.	169
Figure 4.26. Convergence of the error of energy norm.	171
Figure 4.27. (a) A beam subject to mid-span point load; (b) The finite element mesh used for problem shown in (a).	172
Figure 4.28. Quarter model for the plate with a central hole.....	174
Figure 4.29. v -displacement distribution for the plate with a central hole subjected to a uni-axial tensile load along the $x = 0$ edge.	176
Figure 4.30. σ_x -stress distribution for the plate with a central hole subjected to a uni-axial tensile load along the $x = 0$ edge.	176
Figure 4.31. The contour plots of the u -displacement for the plate with a central hole.	177
Figure 4.32. The contour plots of the v -displacement for the plate with a central hole.	178

Figure 4.33. Contour plots of σ_x -stress for the plate with a central hole.	179
Figure 4.34. The contour plots of σ_y -stress for the plane with a central hole.	180
Figure 4.35. The contour plots of σ_{xy} -stress for the plane with a central hole.	181
Figure 5.1. Free vibration analysis of a cantilever beam. (a) The geometry of the cantilever beam; (b) Coarse mesh; (c) Fine mesh.	189
Figure 5.2. First ten vibration modes of the cantilever beam computed using present element.	193
Figure 5.3. The sequence of meshes used for the convergence study.	194
Figure 5.4. Convergence of the error in the computed frequency for the first two modes.	195
Figure 5.5. A shear wall with four openings.	198
Figure 5.6. The mesh used for the distortion sensitivity test.	199
Figure 5.7. The percent error in computed frequencies for distortion sensitivity test.	200
Figure. 5.8. The Rayleigh-Lamb problem.	201
Figure 6.1. Initial and two consecutive configurations of a body.	206
Figure 6.2. Cantilever beam under shear force (a) Geometry; (b) Regular mesh, (c) Distorted mesh.	219
Figure 6.3. Load versus deflection curve for point A the beam subjected to a shear force at the end as shown in Fig. 6.2 under regular mesh.	220

Figure 6.4. Load versus deflection curve at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 under distorted mesh.	221
Figure 6.5. Convergence of vertical deflection for the cantilever shear force problems (Fig. 6.2).	222
Figure 6.6. Clamped beam under central point force.	223
Figure 6.7. The load versus deflection curve at point A of the beam subjected to central point force as shown in Fig. 6.6 (For regular mesh).....	224
Figure 6.8. The load versus deflection curve at point A of the beam subjected to central point force as shown in Fig. 6.6 (For distorted mesh).	225
Figure 6.9. A column under an eccentric compressive load: (a) geometry; (b) regular mesh; (c) distorted mesh.	226
Figure 6.10. Load versus horizontal displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).	227
Figure 6.11. Load versus vertical displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).	228
Figure 6.12. Load versus horizontal displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).	229

Figure 6.13. Load versus vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).	230
Figure 6.14. Beam subject to central transverse load.	231
Figure 6.15. Load versus deflection curve for point A as shown in Fig. (6.14) under regular mesh.	233
Figure 6.16. Load versus deflection curve for point A as shown in Fig. (6.14) under distorted mesh.	233
Figure 6.17. Load versus deflection curve at point A of the Cook's skew beam as shown in Fig. 4.19.	235
Figure 6.18. A patch of elements with geometric distortions.	237
Figure 6.19. The present error of deflection of point A for distortion sensitivity test.	237
Figure 7.1. Patches for linear dependence check.	243
Figure 7.2. Nodal density.	245
Figure 7.3. Cantilever beam for the study of shear locking.	247
Figure. 7.4. Superelements used for test problem 1.	251
Figure.7.5. Meshes used for boundary condition 1.	251
Figure. 7.6. Meshes used for boundary condition 2.	252
Figure.7.7. Convergence of inf-sup value with mesh refinement for the boundary condition shown in Fig. 7.5.	252

Figure.7.8 Convergence of inf-sup value with mesh refinement for the boundary condition shown in Fig.7.6.	253
Figure. 7.9. Test problem 2 with boundary condition 1.	254
Figure. 7.10. Test problem 2 with boundary condition 2.	254
Figure. 7.11. Convergence of inf-sup value with mesh refinement for test problem 2.	255
Figure. 7.12. Convergence of inf-sup value with mesh refinement for test problem 3.	256
Figure 7.13. Bending of thick curved beam.	257
Figure. 7.14. Convergence of inf-sup value with mesh refinement for test problem 4.	257

LIST OF TABLES

Table 2.1. The elements for comparison.	27
Table 2.2. Performance comparison of typical 4-node elements.	28
Table 3.1. Vertical displacement (<i>mm</i>) at the tip of the beam under pure bending.	80
Table 3.2. Vertical displacement (<i>mm</i>) at the tip of the beam under a tip shear force.	85
Table 3.3. Vertical displacement (<i>mm</i>) at the tip of the beam under pending bending moment.	97
Table 3.4. Vertical displacement (<i>mm</i>) at the tip of the beam under a tip shear force	101
Table 4.1. Patch test results.	140
Table 4.2. Coordinates for patch (a).	141
Table 4.3. Normalized displacements at node 5 for linear strain type B Patch test (Regular mesh).	142
Table 4.4. Normalised displacements at node 5 for linear strain type B Patch test (Distorted mesh).	142
Table 4.5. Normalised displacements at node 5 for linear strain type C Patch test (Regular mesh).	142
Table 4.6. Normalised displacements at node 5 for linear strain type C Patch test (Distorted mesh).	143

Table 4.7. Normalised displacements at node 5 for Quadratic strain type B Patch test (Regular mesh).....	149
Table 4.8. Normalised displacements at node 5 for Quadratic strain type B Patch test (Distorted mesh).....	149
Table 4.9. Normalised displacements at node 5 for Quadratic strain type C Patch test (Regular mesh).....	149
Table 4.10. Normalised displacements at node 5 for Quadratic strain type C Patch test (Distorted mesh).....	150
Table 4.11. Computed ν_C for Cook's skew beam.....	156
Table 4.12. $(\sigma_A)_{\max}$ of Cook's skew beam.....	156
Table 4.13. $(\sigma_B)_{\min}$ of Cook's skew beam.....	157
Table 4.14. Computed vertical deflection of a thick curved beam (Fig. 4.20).....	157
Table 4.15. Normalized tip deflection for MacNeal's beam.	159
Table 4.16. Tip deflection for MacNeal's beam as $\nu \rightarrow 0.5$	160
Table 4.17. Vertical displacement* at the tip of the beam under pure tip moment.	161
Table 4.18. Vertical displacement* at the tip of the beam under a tip shear load.	165
Table 4.19. Computed displacements* for different orders of Gaussian integration of stiffness matrix.	166
Table 4.20. Error energy norm of the cantilever beam subjected to shear force...	171

Table 4.21. Maximum vertical displacement* of the beam at point A under mid-span point load.	173
Table 4.22. Comparison of error energy norm.	175
Table 5.1. Comparison of computed frequencies (Hz) for the cantilever beam with a coarse mesh (Mesh b).	191
Table 5.2. Comparison of computed frequencies (Hz) for the cantilever beam with a fine mesh (Mesh c).	192
Table 5.3. Convergence of computed natural frequencies (Hz) for the cantilever beam using the FE-LSPIM QUAD4 (present) element.	196
Table 5.4. Convergence of computed natural frequencies (Hz) for the cantilever beam using QUAD4.	196
Table 5.5. Convergence of computed natural frequencies (Hz) for the cantilever beam using QM6.	197
Table 5.6. Natural frequencies of the shear wall.	198
Table 5.7. Computed natural frequencies (Hz) for the distortion sensitivity test.	199
Table 5.8. Comparison of computed frequencies (Hz) and CPU time for the Rayleigh-Lamb problem.	201
Table 6.1. Deflection at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 (For regular mesh).	220
Table 6.2. Deflection (m) at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 (For distorted mesh).	221
Table 6.3. Deflection of the cantilever beam under shear force at the end.	222

Table 6.4. The computed results of vertical displacement (mm) at point A of the beam subjected to central point force as shown in Fig. 6.6 (For regular mesh).	224
Table 6.5. The computed results of vertical displacement (mm) at point A of the beam subjected to central point force as shown in Fig. 6.6 (For distorted mesh).	225
Table 6.6. The horizontal displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).	227
Table 6.7. The vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).	228
Table 6.8. The horizontal displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).	229
Table 6.9. The vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).	230
Table 6.10. The vertical displacement of point A as shown in Fig. (6.14) (For regular mesh).	232
Table 6.11. The vertical displacement of point A as shown in Fig. (6.14) (For distorted mesh).	232
Table 6.12. The vertical displacement at point A of the Cook's skew beam as shown in Fig. 4.19.	235
Table 6.13. The vertical displacement at point A of the elongated Cook's skew beam.....	236

Table 6.14. Comparison of deflection of point A for a cantilever beam with distorted mesh	238
Table 7.1. Eigenvalue analysis of global stiffness matrix.	242
Table 7.2. Normalized tip deflection for the beam subjected to constant moment.	247
Table 7.3. Normalized tip deflection for the beam subjected to shear force.	248
Table 7.4. Computational efficiency for the proposed element.	258
Table 7.5. FE-LSPIM versus FEM.	259
Table 7.6. FE-LSPIM versus Meshfree methods.	259

LIST OF SYMBOLS

CONVENTIONS

Vectors and vector fields, tensors-boldface;

OPERATORS

$\rightarrow$	A mapping
$\mathbf{A}^T$	Transpose of a matrix $\mathbf{A}$
	Inner product of a second order tensor with a vector (i.e., $(\mathbf{A} \cdot \mathbf{b})_i = A_{ij}b_j$)
	or a vector with a vector (i.e., $(\mathbf{b} \cdot \mathbf{b})_i = b_i b_i$)
	Inter product of two second order or higher (i.e., $(\mathbf{A} : \mathbf{B})_{ij} = A_{ijkl}B_{kl}$)
$\sum_{e=1}^n ()$	Finite element assembly process

MATERIAL CONSTANTS

ν	Poisson's ratio
μ	Lame's constant
κ	Bulk modulus
E	Young's modulus

Chapter 3

u, v	Displacement fields
$p(x)$	Basis function
J	Weighted L_2 norm
w	Weight function
N_i	Parametric shape function
v_i	Trial function
δv	Virtual trial function
Ω	Domain of the body
Γ_t	Natural boundary
Γ_u	Displacement boundary

Φ_I	EFG shape functions
ϵ	Strain field
σ	Stress field
$\mathbf{u}$	Displacement field
$\mathbf{b}$	Body force intensity
$\mathbf{f}$	Global vector of nodal forces
$\mathbf{L}$	Linear operator matrix
$\mathbf{N}$	Matrix of metric shape functions
$\mathbf{B}$	Strain-displacement matrix
$\mathbf{P}$	Polynomial basis matrix
$\mathbf{D}$	Constitutive matrix
$\mathbf{K}$	Stiffness matrix
$\mathbf{a}$	Constant vector
Φ	PIM shape functions

Chapter 4

$u_i(x, y)$	Nodal displacement values
Φ_i	LSPIM nodal shape function array associated with the support of node i .
Φ	LSPIM elemental shape functions matrix
$\mathbf{u}^e$	Displacement field in the support of element
Ψ	Shape function matrix of FE-LSPIM QUAD4 element
$\mathbf{M}$	Global shape functions matrix of FE-LSPIM QUAD4 element

Chapter 5

ω	Circular frequency
ρ	Mass density
$\mathbf{M}$	Mass matrix

Chapter 6

δW	Virtual work
C_i	Geometry configuration

F	Deformation gradient tensor
E	Green-Lagrange strain tensor
S	Second Piola-Kirchhoff stress tensor
e	Linear strain tensor
η	Nonlinear strain tensor
σ	Cauchy stress
$\mathbf{B}_L$	Linear strain-displacement matrix
$\mathbf{B}_{NL}$	Nonlinear strain-displacement matrix
J	The determinant of deformation gradient tensor

CHAPTER 1. INTRODUCTION

1.1. MOTIVATION

Over the last few decades, the finite element method (FEM) has been successfully applied to solve problems in various disciplines of engineering. In spite of its credibility and versatility accrued over the years, the FEM does have a few shortcomings, an important one being the requirement of a good quality mesh for a reliable solution. It is well known that many existing finite elements exhibit poor performance under distorted meshes. Distortion sensitivity of elements leads to considerable difficulties in large deformation analysis using updated Lagrangian formulation where the mesh quality progressively deteriorates with every updating of mesh geometry and thereby necessitates frequent remeshing which finally adds to the cost of analysis. Apparently, sensitivity of finite elements to mesh distortions is a problem that has not yet been completely resolved.

The recent meshfree methods, on the other hand, do not require a mesh and hence are free from problems related to mesh distortion. The interpolation functions in meshfree methods are particularly attractive in the presence of high stress gradients, concentrated forces, and large deformations. However, meshfree methods suffer from problems different from that of FEM. For example, implementing the displacement boundary conditions becomes cumbersome. They also suffer from an increased computational cost that reduces their effectiveness. On the other hand, from a practical point of view, finite element methods are less costly, implement essential boundary conditions in a

simple way (because there is no need for Lagrange multipliers), and, above all, they are widely used and trusted by practitioners. Nevertheless, the relative cost of mesh generation process can be high for some problems. In particular, the cost of remeshing in adaptive refinement problems is clearly not negligible. Moreover, the accuracy of finite element method is poor compared to that of meshfree methods. Thus, the FEM and meshfree methods have their own merits and demerits.

As a result, there have been new efforts in recent years to develop hybrid methods combining the FE and meshfree methods. These methods attempt to synergise the strengths of the two methods while avoiding their weaknesses. These hybrid methods can be divided into two main categories based on the concept on which these methods are based. In the first approach, the domain of the problem is divided into *two sub-domains*, modelled by the FE and meshfree methods individually. At the interface between the sub-domains, the continuity of displacements and the internal force balance are enforced. The second approach of hybrid methods, which is relatively of recent origin, is to construct better shape functions by combining the virtues of finite element approximations and meshfree approximations.

In the first approach of the hybrid FE-meshfree methods, it is often difficult to match the FE and meshfree methods in a natural way at the interface between the regions. This is because, in the interface region, the continuity of the primary variable and the balance of the secondary variables need to be maintained, and this becomes cumbersome as the nodal unknowns are real displacements in the FE method whereas they are *displacement parameters* for the meshfree methods. In the second approach, the standard finite

element spaces are augmented by adding special functions. The basic idea behind this approach is to use a meshfree approximation in combination with the usual finite element shape functions. In this approach, the problem domain need not be divided into *two sub-domains*, and hence the compatibility problem at the interface region, which is a serious problem in the first class of hybrid methods, does not arise. This makes the approach quite appealing. However, there can be many different ways of combining the meshfree and finite element shape functions, but only a few have been explored so far. There can be better ways to combine the shape functions and this forms the motivation for the present research.

1.2. OBJECTIVE AND SCOPE

The objectives of this research are as follows:

1. To develop a hybrid method combining the finite element and meshfree shape functions so that it has the following advantages:
 - i) It has accuracy comparable with meshfree method.
 - ii) Simplicity of implementing displacement boundary conditions and ease of coding as in the finite element method.
 - iii) It inherits the compatibility properties of finite element method and higher order completeness properties of meshfree shape functions so that it has good mesh-distortion tolerance, or in other words, less sensitivity to nodal arrangement as in meshfree methods.
2. To apply the hybrid method to static analysis, free vibration and geometrical nonlinear analysis, and assess its effectiveness using typical test problems.

3. To investigate and bring out the merits and demerits of the method.

The thesis is mainly concerned with the development of hybrid FE-meshfree method with respect to 4-node quadrilateral element and its scope limited to 2-D linear and geometrical nonlinear static analysis and free vibration analysis.

Two different means of combining the shape functions are explored this thesis. In the first, the finite element shape functions are used as test functions and the meshfree shape functions are used as trial functions. In the second, a product of finite element and meshfree shape functions is formed using the concept of partition of unity (PU) and the resulting shape functions are used as both test and trial functions.

The initial investigations focus on combining the finite element shape functions and element-free Galerkin (EFG) shape functions. Subsequent investigations combine the finite element shape functions and the shape functions of point interpolation method (PIM).

The computer codes for implementing the methods developed in this thesis have been written in MATLAB and the listing of some of them are included in Appendices I.

1.3. ORGANIZATION OF THESIS

The thesis is presented in seven chapters as listed in Fig. 1.1. The methods and elements developed in this thesis are shown schematically in Fig. 1.2.

Figure 1.1. Structure of the thesis.

Figure 1.2. The methods and the elements developed in this thesis.

In Chapter 2, the literature related to the present work is reviewed. The strength and limitations of previous works are discussed. This chapter organized in three parts. The first part gives a brief overview of several important approaches of finite element methods (FEM), the second covers some important developments in the meshfree methods and the third focuses on the development of hybrid methods that combine meshfree and finite element methods.

In Chapter 3, two hybrid methods, viz., the FE-EFG method and the FE-PIM method, are developed based on the local Petrov-Galerkin approach. The hybrid formulations combine the concepts of FEM and EFG/PIM methods. The FE-EFG method is characterized by use of two sets of shape functions, viz., FE shape functions used as test functions, the EFG shape functions used as trial functions. Similarly, for the FE-PIM method, the FE shape functions and PIM shape functions are used as test and trial functions, respectively. The performance of these two types of methods is studied using

several numerical examples. The hybrid methods developed in this chapter have certain drawbacks. Chapter 4 aims to remove these drawbacks.

In Chapter 4, two 4-node quadrilateral elements, viz., FE-LSPIM QUAD4 and FE-LSPIM QUAD4U, are developed. The FE-LSPIM QUAD4 employs novel shape functions which are obtained by a product of FEM and LSPIM shape functions. As a result, the element inherits the completeness properties of the meshfree shape functions and compatibility properties of the finite element shape functions. The FE-LSPIM QUAD4 is based on symmetric (Galerkin) formulation while FE-LSPIM QUAD4U is based on unsymmetric (Petrov-Galerkin) formulation.

In Chapter 5, the FE-LSPIM QUAD4 element is extended to free vibration analysis of two-dimensional solids. Typical benchmark problems are solved to test the efficacy of the element.

In Chapter 6, The FE-LSPIM QUAD4 element is extended to geometrically nonlinear problems. The geometrical nonlinear formulation is derived based on the updated Lagrangian formulation. The performance of the proposed element is studied under various numerical examples.

In Chapter 7, some critical issues of the FE-LSPIM QUAD4, such as singularity problem, linear independence problem and computing efficiency are studied.

Finally, in Chapter 8, the major conclusions from this research are summarized. Directions for future research works are outlined.

CHAPTER 2. LITERATURE REVIEW

2.1. INTRODUCTION

The present thesis is an effort to synergize the finite element method and meshfree methods so as to develop a hybrid method that enjoys the advantages of both classes of methods; the literature review is organized in three parts. In the first part of this review, the development of finite element methods (FEM) is briefly reviewed. The finite element method evolved in the late 1950s for structural analysis in aerospace industry. Since then extensive work has been done and the literature currently available on finite element method is vast. As it is hard to cover all of them in this chapter, the review is limited to latest developments focusing on improving the element performance, enhancing the mesh distortion tolerance and attempts to remove locking.

In the second part of this review, the development of the meshfree method is reviewed. Although the finite element methods have gained much popularity in the past years, it is not completely free from drawbacks. For example, the computed strain may exhibit discontinuity at element boundaries. Furthermore, if the mesh employed for the analysis has geometric distortions, then it often leads to serious degradation of accuracy of the computed displacements and stresses. In large deformation problems, even if the initial mesh chosen has no geometric distortions, the mesh distortions creep in as the geometry is updated during load increments. Such drawbacks of finite element method (FEM) are inherently avoided in the meshfree methods. The meshfree methods do not need a mesh to discretize the problem domain and hence are completely free of mesh and the related

problems. Furthermore, the meshfree approximations are usually smoother than the FE counterparts and hence the computed displacements and stresses are more accurate.

In the third part of this review, FE-meshfree hybrid methods are reviewed. Although, the meshfree methods are free from the drawbacks of FEM, they introduce their own. For example, the lack of *Kronecker delta* properties of meshfree shape functions makes it rather cumbersome to implement the displacement boundary conditions. Furthermore, the meshfree methods are computationally more expensive than the FE methods due to the use of complex shape functions. There has been much interest among researches in combining these two different numerical methods because such a coupling could exploit the potential benefits of the individual methods while avoiding their deficiencies. Many hybrid methods have appeared in recent years. Those related to the present method are reviewed in the third part of this review.

2.2. REVIEW OF FINITE ELEMENT METHODS

In area of computational mechanics, the FEM is one of the most effective tools available for the numerical solution of complex engineering problems. Usually, most practical problems encountered are too complicated to be solved satisfactorily by the classical analytical methods. The finite element method originated as a method for stress analysis. Today, the method is more widely used to analyze problems in heat transfer, fluid flow, lubrication, electric and magnetic fields. The method has achieved remarkable success in many linear and nonlinear problems. A salient feature of the finite element method is that it divides a continuum into many elements. This process of division is often called discretization. The individual elements are connected together by a topological map,

which is usually called a mesh. The finite element interpolation functions are then built over the mesh ensuring compatibility of the interpolation. A major goal of the element research has been to develop preferably low-order elements with as high an accuracy as possible to meet two main requirements, i.e. avoiding locking behavior in bending or incompressible problems, and enhancing the mesh distortion tolerance.

To construct elements that fulfill the above requirements, several approaches have been proposed over the years. Chronologically these can be classified as (1) Reduced integration and stabilization methods, (2) Incompatible modes approach (3) Multi-field elements, (4) Enhanced assumed strain approach, (5) Drilling degrees of freedom elements, (6) Area coordinates elements, (7) Unsymmetric elements. These approaches are briefly reviewed in the rest of this section.

2.2.1. REDUCED INTEGRATION AND STABILIZATION METHOD

Applications of FEM to industrial problems as well as non-linear and transient analysis motivated the finite element researchers to develop simple and efficient elements. The emergence of a series of elements that used lower-order polynomial expressions focused primarily on simplicity in mesh generation and robustness of performance in complicated non-linear problems with multiple contacting surfaces. These elements used the concept of reduced and selective reduced integration. A vast amount of literature exists on this topic. Some classical works are cited here. Hughes et al. [1] proposed a quadrilateral element with bilinear deflection and rotation based on Mindlin plate theory under the name U1. A single point quadrature was introduced in this paper.

However, the element U1 is rank deficient, which leads to singularity or near singularity of assembled stiffness matrix under some mesh geometries and boundary conditions. Later Hughes and Tezduyar [2] solved the rank deficiency by using 2×2 quadrature and refined the interpolation of the transverse shear by introduced a scheme motivated by the work of MacNeal [3]. However, the attractive potential of the bilinear element, namely, the use of one-point quadrature, is lost by use of this scheme. Another approach to this problem was introduced by Taylor [4], who explored the use of hourglass control scheme.

Locking in finite elements has been a major concern since its early developments. Locking appears because a poor numerical interpolation leads to an over-constrained system. The first element to solve this numerical problem is a clever implementation of 4-node quadrilateral [5]. Based on the selective integration concept, in which 2×2 and 1×1 Gaussian integration are used for computing bending and shear energy, this element presents a good behavior from moderately thick situations to very thin plates without locking. An analysis of the eigenvalues of stiffness matrix shows two spurious kinematical modes rendering element sensitive to boundary conditions [6]. The spurious modes may have an impact on the element behavior in thick plate situations.

The reduced integration was able to avoid shear locking, but introduced zero-energy modes into elements [7,8]. To overcome the setback of the appearance of spurious zero-energy modes caused by the usual reduced integration schemes, Kavanagh and Key [9] put forward the selective reduced integration (SRI) technique. Thereafter, many stabilization methods were also proposed for removing these undesirable modes. The

most widely known is the γ -method. Based on a vector γ orthogonal to some displacement fields, γ -method defined a stabilization matrix according to a technical artifice that is initiated by Flanagan and Belytschko [10]. The quadrilateral element of Belytschko and Tsay [11] and the triangular element of Belytschko et al. [12] are simple to formulate. They require only one integration point to compute the internal strain energy. Several other γ -methods are described in Reference [13]. With regard to the reduced integration technique, Prathap and Vishwanath [14] observed that the reduced integration does not ensure the absence of shear locking. The ‘consistent field’ approach proposed by them [15] employ an optimal strategy with (2×1) and (1×2) Gauss integration of the terms γ_{yz}^2 and γ_{xz}^2 terms respectively. Their QUAD4-CC element is free of locking and has a proper rank. Unfortunately, this advantage is limited to rectangular forms, the element locks when used as quadrilateral. Other stabilization techniques, more natural than the γ -methods and the selective-reduced integration methods have also appeared. Hughes and Tezduyar [16] are the first to show that shear locking could be avoided without losing important energy modes.

2.2.2. INCOMPATIBLE MODES APPROACH

Numerous applications of the FEM involve large strain deformations. Problems undergoing large elastic strains are often constrained by the incompressibility of the material. However, the presence of incompressibility constraint leads to locking phenomenon when standard four-node quadrilateral elements are used. The incompatible modes are introduced to solve this incompatibility locking phenomenon.

In this approach, the element displacement is considered with compatible part and incompatible part.

$$u = u_q + u_\lambda = N_q q + N_\lambda \lambda \quad (2.1)$$

where N_q is the compatible displacement interpolation function wherein the constant-strain mode and the rigid body modes should be included. N_λ is the additional incompatible modes, and q and λ are nodal displacement and internal displacement parameters, respectively.

Wilson et al. [17] introduced a set of incompatible displacements to improve the displacement element based bending behavior. Two quadratic incompatible modes are used to construct the shape functions of the element. The practical features of the method include higher-order accuracy for a coarse mesh, mesh distortion insensitivity and excellent performance in the analysis of nearly incompressible and non-linear materials. However, the deficiencies of the initial formulation of Wilson's elements when they assume the distorted configuration lead the researchers to ignore even the desirable features of the method. Their new element Q6 can solve in-plane bending problems for regular and parallelogram shaped elements but fail to pass the constant strain patch test.

Taylor et al. [18] modified the incompatible modes so that the distorted elements can pass the constant strain/stress patch test. Taylor developed the QM6 element using an ad hoc method in which the incompatible strains are computed using the Jacobian transformation evaluated at the local origin of the elements. The new modified element QM6 can pass the constant strain patch test but perform worse for bending modes under

distorted mesh. Thereafter, Taylor et al. [19] and Wu et al. [20] presented the revised incompatible shape functions which automatically pass the patch test. Wilson and Ibrahimbegovic [21] suggested a direct modification of the incompatible strain-displacement matrix which avoids seeking for shape functions satisfying the patch test. However, general formulations to revise incompatible shape functions or strain-displacement matrix are insufficient, especially for axisymmetric problems.

A QS6 element is proposed by Hueck and coworkers [22,23]. Hueck et al. [22] developed a bilinear displacement field rectangular four-node quadrilateral element, enhanced by incompatible modes. Corresponding to under-integration and stabilization of the element stiffness matrix, the gradient operators were divided into constant and linear parts. Further, Hueck and Wriggers [23] use a second-order Taylor series expansion of element basis functions in the physical co-ordinates to generate the expressions for gradient operators. By this way, the internal degrees of freedom of the incompatible modes were eliminated at the element level. Later, Wriggers and Hueck [24] developed a formulation of QS6 element for large elastic deformations. The Hu-Washizu principle is used to derive the equations for the element construction. The derived element performs well for standard test problems and does not lock in the incompressible limit. The QS6 element possesses good coarse mesh accuracy, and its robustness under geometric distortion is high.

Wu et al. [25] introduced the complete quadratic incompatible mode. In this paper, firstly, an energy consistency condition for axisymmetric problems is presented which can be used to develop axisymmetric incompatible elements. Thereafter, a revise-

stiffness (R-S) approach for incompatible elements is suggested. Their model can satisfy the patch test condition for axisymmetric problems automatically. But a special case which may lead to the formulation invalid is missed in this paper. Hence, Huang and Li [26] modified the model proposed by Wu et al. and developed a four node incompatible element with quadratic completeness (P2 element) in the physical space for plane and axisymmetric problems. Their P2 element has the same degrees of freedom as Q6 element but maintains quadratic completeness in physical space when irregular meshes are adopted. When coarse and distorted meshes are involved, the plane P2 element performs well.

2.2.3. MULTI-FIELDS ELEMENTS

As an alternative to the displacement models, multi-fields elements have also been an area of active research in finite element analysis. Since these elements perform very well in the incompressibility limit as well as bending situations, they have been applied to simulate geometrically and materially non-linear problems. The multi-fields elements are formulated from either Hellinger-Reissner (displacement and stress) or Wu-Washizu (displacements, stresses and strains) variational principles. Since the pioneering work of Pian [27] for hybrid stress elements, great efforts have been devoted to developing high-performance hybrid stress elements. The main issues have been coordinate frame invariance, satisfaction of equilibrium conditions and sensitivity to mesh distortion. In the early stage of development, hybrid stress elements were based on the principle of minimum complementary energy and assumed stresses. The performance of the resulting assumed stress elements are affected very much by mesh distortion and it is

not easy to satisfy coordinate invariance and equilibrium conditions simultaneously. To overcome these weaknesses, Pian and Sumihara [28] proposed a hybrid stress elements based on the Hellinger-Reissner principle and defined the assumed stresses in the natural coordinate system. The stresses are initially defined in the form of complete uncoupled polynomials, and then equilibrium conditions are enforced on them through integral-type equations with geometric perturbation, and finally, they are degenerated to optimal stress terms. As modified versions of this scheme, Pian and Tong [29] and Pian and Wu [30] proposed new constraint equations for assumed stresses which do not need geometrical perturbation to obtain the sufficient number of constraint equations and Yuan et al. [31] adopted oblique coordinate system in stress assumption. A unified discussion on the above approaches can be seen in the paper [32].

Chen and Cheung [33,34] developed a new approach for approximation of the assumed stresses. The key-point of this approach lies in the rational choice of the incompatible modes containing first-order terms, which are introduced to relax the sensitivity to mesh distortion. This is achieved by introducing a parameter which takes values between zero and unity in the assumed stresses. Other approaches for hybrid stress method can be seen in the works of Bouzeghoub and Gunn [35] who transformed the curvilinear stress components to the Cartesian stress components via equilibrium conditions. Wu and Cheung [36] introduced a penalty equilibrium term for enforcing equilibrium conditions in a variational sense. The equilibrium equation is enforced at the individual elements directly. By this way, neither the modification of the initial stress nor the additional variables are required for the improvement of many existing hybrid elements. And also the rational energy constraint to be used as the optimization condition for hybrid finite

element is presented. In this approach, the improvement of the element type is carried out by a pretreatment of the initially assumed stress field.

Size and Fan [37] proposed a highly economical 8-node brick element consolidated from the previous work on admissible matrix formulation. The element is modified from the Pian-Tong [29] element in a way that the latter's good bending response, applicability in nearly incompressible materials and insusceptibility to mesh distortion are retained. It is found that their element is 18% more economical than the 8-node displacement element. The transverse (thinnest) direction of their element can be aligned with any of the natural coordinate axes. Indeed, reliability of stress calculation in the case of distorted meshes is important in stress analysis problems, especially in the geometrical and material non-linear problems, where the solutions are strongly dependent on the accuracy of stresses. In this respect, the hybrid stress element would be more attractive than the displacement element. The main issue of hybrid stress model has been assuming a suitable stress that can ensure the accuracy of stresses even when meshes are distorted. Yeo and Lee [38] proposed a new technique for stress assumption which is based on the constraint equations of Pian and Tong [29] and a measure of mesh distortion. They suggest generalized incompatible modes for a four-node plane element and modified incompatible strains for an eight-node brick element. In relation to the constraint equations, the role of the general incompatible modes on the assumed stress field has been investigated.

Based on a combination of Hellinger-Reissner principle, Zhou and Nie [39] proposed a mixed/hybrid method of non-saddle point type. Four combined hybrid elements of

lower order CH(0), CH(P-S), CH(0-1), CH(1), are constructed by using same Wilson's [17] incompatible bubble functions with different choice of stress assumption. The paper shows that the energy compatibility condition is the ultimate key to the achievement of high performance. To improve on the defects of CH (0-1), Xie and Zhou [40] derived two optimal hybrid stress quadrilaterals elements called ECQ₄ and LQ₆. The ECQ₄ element is derived by using compatible isoparametric bilinear QUAD4 displacements and 5-parameter energy compatible stress assumption of the combined hybrid finite element CH (0-1). A convergence analysis is given and numerical experiments show that ECQ₄ and LQ₆ elements give high performance. They are accurate even with coarse meshes, insensitive to mesh distortions and free from locking.

Mijuca [41] proposed a hexahedral finite element HC8/27. This element is derived from primal-mixed variational formulation based on Hellinger-Reissner's principle. The novel properties of the HC8/27 are stress approximation by the continuous base functions, introduction of stress constraints as displacement boundary conditions, and initial displacement and stress/strain field capability. This element satisfies mathematical convergence requirements, like consistency and stability, even when it is rigorously slandered, distorted or used for the nearly incompressible materials.

Xie [42] proposed a hybrid macro-element (HQM/HQ4) by combining two triangles into a four node quadrilateral to improve the performance of the constant strain triangular element. On each triangle, compatible linear displacements are assumed and a 5-parameter incomplete linear stress mode is designed on the macro-element. The macro-element is equivalent to another new hybrid quadrilateral HQ4 with isoparametric

bilinear displacements. The HQM/HQ4 element is of good accuracy, such as insensitive to mesh distortions and free from Poisson locking. The deficiency of this element is that the meshes employed have to be quadrilateral based.

2.2.4. ENHANCED ASSUMED STRAIN APPROACH

In the 1990s, Simo and Rifai [43] introduced the Enhanced Assumed Strain method (EAS), related to the work of Wilson and co-authors (incompatible modes method [17]). The strain field can be enlarged with the aid of extra internal field variables, thus resulting in additional deformation modes. In their work, the total strain field is built up as a summation of the compatible symmetric gradient of the displacement field and the enhanced strain. This latter is not subjected to any inter-element continuity requirement, and can be related to an ‘incompatible mode’ field. Simo and Rifai presented a bilinear isoparametric plate element with enhanced transverse shear strain field. Stolarski and Chen [44] proposed a different set of assumed strain fields in their element. The core idea of their element depends on correct identification of various deformation modes in structural members. Korelc and Wriggers [45] proposed a 4-node quadrilateral element (QP6). This element uses the concept of the assumed strain method enhanced with the Taylor expansion of the shape function derivatives. Two additional strain fields are added in conjunction with the standard field in the enhanced assumed strain field.

Due to its versatility, the enhanced strain method has been successfully applied to 2D, 3D and shell formulations achieving very promising results [46, 47, 48, 49]. On the other hand, one of the major disadvantages of earlier EAS elements was their

computational inefficiency. In fact, the number of enhancing variables and the use of full quadrature rules dramatically increase the total CPU time. Several efforts have been made on the achievement of efficient and accurate solid elements designed specifically for thin-walled problems, either based on SRI or complete integration schemes, as illustrated in the works of Hauptmann and co-authors [50, 51] and many others. Although being accurate and free of locking, these formulations do have a limited number of integration points along thickness direction within a single layer model.

Kasperl and Taylor [52] developed a four-node quadrilateral element for the analysis of the axisymmetric-torsion problem. The deformation gradient is developed from two sets of element interpolants, of which one may be explicitly obtained resulting in only two undetermined enhanced parameters needed for the satisfaction of the near incompressible limit. The resulting element arrays yield proper rank with standard 2×2 quadrature.

2.2.5. ELEMENTS WITH DRILLING DEGREES OF FREEDOM

The need for membrane elements with drilling degrees of freedom arises in many practical engineering problems. When combined with a plate bending element, a versatile flat shell element which possess 6 degrees of freedom per node can be formulated. Introduction of the rotational degrees of freedom to hexahedral element is also viewed as particularly advantageous in structural analysis.

Earlier effort to develop different types of elements (thick shell and solid element) [53] or variable node elements (triangular and quadrilateral flat shell elements) [54] with drilling rotation fields in a unified framework have been reported. However, a unified approach to the formulation of the membrane and hexahedral elements with drilling degrees of freedom can be hardly found in the published literature.

One of the widely adopted approaches to take into account the ‘drilling degrees of freedom’ is to use the Allman-type shape functions [55, 56, 57]. For an in-plane bending problem the Allman-type element shows a good performance but fails to pass the one-element patch test as performed by Iura and Atluri [58] with a minimum number of constraints. The reason for the failure of the test is that they applied the nodal forces only but omitted the consistent nodal moments which are required in addition to the nodal forces to give a constant stress state in the element.

Choi et al. [59] shows the combination of basic non-conforming modes and additional higher order non-conforming modes can improve the performance of the distorted 8-node hexahedral elements. Their elements, however, did not have the rotational degrees of freedom at nodes. The direct modification method was successfully extended to modify nonconforming modes of membrane elements by Choi et al [60]. This modification scheme is utilized in the formulation of non-conforming modes to improve the general behavior of isoparametric-based elements. It is very effective in eliminating the undesirable locking in distorted meshes for both non-conforming quadrilateral membrane and hexahedral elements with drilling (or rotational) degrees of freedom.

2.2.6. AREA COORDINATES ELEMENTS

Versatile quadrilateral elements using isoparametric coordinates have been developed in the past years. The main advantage of isoparametric formulation is that the element boundary conditions can be easily described and satisfied in solving problems of C^0 continuity while such formulation approach still has several disadvantages:

1. ξ and η are irrational functions of x and y so cannot be expressed as terms of Cartesian coordinates x and y in finite terms.
2. The element stiffness matrix contains the determinant of Jacobian inverse, for which the value obtained by numerical integration is usually only an approximation.
3. The displacement fields corresponding to the constant strain fields are polynomials consisting of high order terms of ξ and η ; thus, it is more difficult to derive the element displacement fields.

The reasons for the isoparametric membrane elements to be very sensitive to mesh distortion are given by Lee and Bathe [61] as follows: For a 8-node serendipity element, for example, the displacement field is expressed as a second order polynomial in the isoparametric coordinate system (ξ, η) . For undistorted element shape (rectangular elements, for example), the transformation between (ξ, η) and (x, y) is linear, and the displacement field can be expressed as a polynomial of the same (second) order in the Cartesian coordinate system (x, y) . For a distorted element, the transformation between (ξ, η) and (x, y) is nonlinear, and the order of the complete polynomial in the Cartesian coordinate system will descend to 1, so the precision of the distorted element

is lower. This is the reason why isoparametric elements perform badly when they are distorted.

In order to overcome the shortcomings of the isoparametric coordinates mentioned above, the quadrilateral area coordinate method was proposed by Long and co-authors [62, 63]. The important advantage of the quadrilateral area coordinate system is: the transformation between the area coordinates and the Cartesian coordinates is always linear. Thus, the order of the displacement field expressed by the area coordinates will not be affected by the mesh distortion. It makes the element insensitive to mesh distortion.

Consider a thin quadrilateral plate element of four nodes, as shown in Fig. 2.1. Long et al. [63]. The position of an arbitrary point P in the element can be expressed in terms of the area coordinates L_1, L_2, L_3 and L_4 , which are defined as:

$$L_i = \frac{A_i}{A}, \quad i = 1, 2, 3, 4 \quad (2.2)$$

where A is the area of the quadrangle; and A_i is the area of the triangle formed by point P and the two nodes on the i th side, as shown in Fig. 2.1.

Figure 2.1. A four node quadrilateral element by Long [63].

Based on the area coordinates developed by Long et al. [62, 63], two 8-node membrane elements AQ8-I and AQ8-II, and a 4-node plate-bending element ACGCQ, were successfully developed by Soh and co-authors [64, 65], respectively. Compared with those traditional elements using isoparametric coordinates, these new elements are less sensitive to mesh distortion. The AQ8-I and AQ8-II elements are found have the same accuracy as the Q8 element when rectangular meshes are used. For arbitrary quadrilateral meshes, the displacement fields always possess second order completeness in Cartesian coordinates, while Q8 element does not. As a result, the AQ8-I and AQ8-II elements are insensitive to mesh distortion but the Q8 element is not. The two new elements can still work well when the element sides are curved.

Using the same approach, Chen et al. [66] proposed two 4-node quadrilaterals AGQ6-I and AGQ6-II. The displacement field of the two elements is expressed with the

polynomial in area coordinates instead of the isoparametric coordinates. Thus, the order of the complete polynomial in the Cartesian coordinates system will not change. In this work, various generalized conforming conditions such as integral form conforming conditions of the displacement along the perimeter of the element are used to determining displacement field. However, the proposed elements pass the constant strain patch test only in the weak form. The performance of these elements agrees with the theory proposed by Macneal [67], which states that the 4-node quadrilateral elements with two degree of freedom per node will either lock in-plane bending or fail the constant strain patch test when element geometries are in the form of trapezoids.

2.2.7. UNSYMMETRIC ELEMENT

Rajendran and Liew [68] proposed a novel 8-node quadrilateral finite element. This element is based on unsymmetric displacement formulation which is characterized by use of two sets of shape functions. The compatibility fulfilling shape functions are chosen to satisfy exactly the displacement continuity requirement while the completeness fulfilling shape functions are chosen to satisfy the completeness requirements.

$$\mathbf{K}^{(e)} = \int_{\Omega^{(e)}} \mathbf{B}_L^T \mathbf{D} \mathbf{B}_R d\Omega^{(e)} \quad (2.3)$$

The stiffness matrix, $\mathbf{K}$, is unsymmetric in view of the two different shape functions used in $\mathbf{B}_L$ and $\mathbf{B}_R$. The test functions are the usual parametric shape function. The trial functions are the so called metric shape functions. This element is capable of reproducing a Cartesian quadratic displacement under any admissible mesh distortions.

The idea of unsymmetric formulation of US-QUAD8 was extended to a new 20-node element for three dimensional problems by Ooi et al. [69]. The proposed element exhibits a very high tolerance to mesh distortions. In particular, for problems involving linear and quadratic displacement fields, the element is capable of reproducing exact solution under all admissible geometrical distortions of the mesh. For test problems involving higher-order displacement fields, the performance of the present element is in general better than that of the classical element.

2.2.8. SUMMARY OF 4-NODE ELEMENTS

Based on the review of literature in section 2.2.1 - 2.2.7, the comparison of several 4-node quadrilateral elements is shown in Table 2.2. The elements considered are listed in Table 2.1.

Table 2.1. The elements for comparison.

No.	Symbols	Element for comparison	Reference
1	Q4	4-node isoparametric element	-
2	Q6	4-node isoparametric element with internal parameters	17
3	QM6	4-node isoparametric element with internal parameters	18
4	QS6	4-node isoparametric element	22
5	P2	4-node incompatible elements	26
6	ECQ4	4-node isoparametric element with internal parameters	40
7	P-S	Stress hybrid element	28
8	HQM/HQ4	4-node isoparametric element with internal parameters	42
9	QP6	4-node isoparametric element with internal parameters	45
10	ACGCQ	4-node plate-bending element	65
11	AGQ6-I/ AGQ6-II	4-node quadrilaterals element	66

Table 2.2. Performance comparison of typical 4-node elements.

Element type	Distortion tolerance	Free of locking	Constant strain patch test (strong form)	Performance in bending beam	Performance in Cook's skew beam
Q4	×	×	-	Worse	Worse
QUAD4	×	×	-	-	-
Q6	√	-	Fail	Very good	Fair
QM6	×	-	Pass	Worse	Fair
P-S	-	-	-	Fair	Fair
QS6	√	√	-	Very good	Very good
P2	√	√	Pass	Very good	Fair
ECQ4	√	√	Fail	Very good	Very good
HQM/HQ4	√	√	-	Very good	Fair
QP6	×	-	-	Fair	Fair
ACGCQ	√		Pass	-	-
AGQ6-I/ AGQ6-II	√	√	Fail	Very good	Very good

2.3. REVIEW MESHFREE METHODS

To circumvent some of the well-known drawbacks of the finite element method, a new class of numerical methods, known as meshfree or meshfree methods has, in recent years, attracted considerable attention of researchers. These methods, unlike finite element method, do not require a mesh to discretize the problem, and employ an approximation of the unknown field based on a scattered set of nodes or particles. The approximate solution is constructed entirely in terms of a set of nodes, and no elements are needed to construct the discrete equations. It is thus possible to develop discrete equations from a set of nodes and description of the internal and external surfaces of the model. The primary attraction of meshfree methods is the possibility of simplifying problems with moving boundaries and discontinuities, such as phase changes or cracks. Furthermore,

meshfree methods can easily handle very large deformations. Another advantage of meshfree methods is that the accuracy can be controlled more easily, since in areas where more refinement is needed, nodes can be added quite easily (h -adaptively). Although meshfree methods have several advantages over the finite element method, they are not free of drawbacks. The primary drawback is high computational time and difficulty in applying boundary conditions. And also the continuity of meshfree approximations is a drawback in problems where the exact solutions have discontinues derivatives. The literature available on meshfree methods is vast although their origin is quite recent compared to finite element method. In the rest of this section, major meshfree methods are reviewed.

2.3.1. THE EARLY DEVELOPMENTS OF MESHFREE METHODS

The earliest development of meshfree methods can be traced to smooth particle hydrodynamics (SPH) proposed by Lucy [70], who used it for modeling astrophysical phenomena without boundaries such as exploding stars and dust clouds. Monaghan and coworkers [71,72] explained the methods as a kernel estimate, which is not directly relevant to the accuracy of the method in the solution of partial differential equations. The applications of SPH were explored in 1990's (Libersky et al. [73,74], Sellingwerf and Wingate [75], Attaway et al. [76]).

Another approach to the meshfree methods is based on the moving least-square (MLS) approximations, which started much later than the SPH. Nayroles et al. [77] developed a

diffuse element method (DEM) that used MLS interpolants in conjunction with Galerkin method to provide a meshfree computational formulation.

2.3.2. ELEMENT FREE GALERKIN METHOD

Belytschko and coworkers [78, 79, 80] refined and modified the method developed by Nayroles et al. [77] and proposed Element Free Galerkin method (EFG). In EFG method, the moving least-square (MLS) approximants is used to construct the shape function, the derivatives of interpolants that were omitted in DEM are included and a more accurate numerical integration method is employed to enhance solution accuracy. The displacement boundary condition is applied by use of Lagrange multipliers. Compared with DEM, the EFG method shows the high rates of convergence and also can treat the incompressible material without any modification. The method appears very effective for crack problems. However, the advantages of the method do not come without some costs. For example, the construction of background mesh for purpose of numerical quadrature can be awkward if there are large differences in node spacing with one mesh. Further more, the use of the Lagrange multipliers complicates the solution process. Some significant developments in EFG can be found in the works [81, 82, 83, 84, 85] etc.

A typical MLS shape function and its derivatives in two dimensions are shown in Fig. 2.2. It can be observed that MLS shape function attains the maximum value that is considerably less than 1 (around 0.6). It can also be seen from the plots that both the shape functions and its derivatives are continuous and smooth. For these plots the quadratic weight is used.

Figure 2.2. Plots of a typical MLS shape function and its derivatives in two dimensions [86].

Although, the EFG method has been used extensively for fracture problems and has yielded good results when adequate refinement is used near the crack tip, the stresses tend to be oscillatory near the crack tip unless substantial refinement is used. To reduce this oscillatory solution, Fleming et al. [87] proposed two ways to enrich the EFG formulation for the singular stress fields at the crack tip. One is to add the asymptotic fields to the trial function and the other is to augment the basis by the asymptotic fields. Both of them have been shown to provide similar accuracy and work well even with few

degrees of freedom. The first method can solve the multiple cracks problems with little additional expense but difficult to program. The second one is easy to program as it only requires a modification of the basis but is expensive for multiple cracks because the size of the moment matrix depends on the number of terms in the basis.

2.3.3. THE NATURAL ELEMENT METHOD

The Natural Element Method (NEM) proposed by Braun and Sambridge [88], Sukumar et al. [89], and Natural Neighbour Method (NNM) proposed by Sukumar et al. [90] are another class of meshfree methods. The interpolation scheme used in this class of methods is known as natural neighbour interpolation. The main advantage of this method is the use of *Voronoi* diagrams to define shape functions which yields a very stable partition. The other advantages include: its delta function property, linear property between adjacent nodes, and well-defined approximation without uncertain user-defined parameter.

Based on the natural element method, Sergio et al. [91] proposed a new meshfree finite element method (MFEM). The shape functions of the MFEM are also based on the *Voronoi* diagrams, but they are different. The NEM shape functions have C^∞ continuity and built using all the *Voronoi* diagram of all the natural neighbor nodes to each point. Hence, the obtained shape function is quite complicated and need several Gauss points to integrate. The MFEM use a particular definition of the non-sibsonian shape function for the limited set of nodes on the same *Voronoi* sphere. On a polyhedron surface, the shape functions depend only on the nodes of this surface. On

triangular surfaces (or in all the polygon boundaries in 2D), the shape functions are linear. The simple shape functions are used in a large part of domain. The shape functions have C^0 continuity between elements so it easy to apply the boundary conditions and easy to deal with the material discontinuities. The comparison of classical 2D shape functions with the MFEM can be seen in Fig.2.3.

Figure 2.3. Shape functions in a 2D regular node distribution: (a) MFEM; (b) MLSQ, SPH, RKPM; (c) NEM (Sergio [91]).

Cai and Zhu [92] proposed a new searching algorithm of NEM. This algorithm is a local search algorithm for natural neighbours instead of a global sweep algorithm of NEM, which quantifies the natural neighbour nodes of the given point based on the locally triangles. The proposed local algorithm is more expedient and convenient in the construction and computation of natural neighbour interpolations compared with the global sweep algorithm used in previous NEM.

2.3.4. REPRODUCING KERNEL PARTICLE METHOD

Liu and coworkers [93, 94] took a different approach to introduce a Reproducing Kernel Particle Method. They introduced a correction function in the kernel of integral transformation to impose reproducing conditions. Adding the correction function in the

kernel significantly enhances the solution accuracy when compared to SPH. The new approximation takes the form

$$u^h(\mathbf{x}) = \int u(\mathbf{y})C(\mathbf{x}, \mathbf{y})W(\mathbf{x} - \mathbf{y}, h)d\Omega_{\mathbf{y}} \quad (2.4)$$

where $C(\mathbf{x}, \mathbf{y})$ is the correction function. An example of the correction function in one dimension is

$$C(x, y) = c_1(x) + c_2(x)(y - x) \quad (2.5)$$

where $c_1(x)$ and $c_2(x)$ are coefficients. The coefficients are found by

$$c_1(x) = \frac{m_2(x)}{(m_0(x)m_2(x) - m_1^2(x))} \quad (2.6a)$$

$$c_2(x) = \frac{m_1(x)}{(m_0(x)m_2(x) - m_1^2(x))} \quad (2.6b)$$

where m_0, m_1 and m_2 are moments of W defined by

$$m_0(x) = \int W(x - y)dy \quad (2.7a)$$

$$m_1(x) = \int yW(x - y)dy \quad (2.7b)$$

$$m_2(x) = \int y^2W(x - y)dy \quad (2.7c)$$

If the integral in (2.4) is discretized, then a function $u(x)$ can be approximated by

$$u^h(\mathbf{x}) = \sum_I C(\mathbf{x}, \mathbf{x}_I)W(\mathbf{x} - \mathbf{x}_I)\Delta V_I \quad (2.8)$$

This method provides a general formulation for the construction of shape functions for meshfree computation. In computer implementation, RKPM is shown to be more efficient than DEM and EFGM. Moreover, if the window function is C^∞ , the solution and its derivatives are also C^∞ in the entire domain.

Liu et al. [95] combined the theory of wavelet analysis in RKPM by introducing the scaling function as the kernel function and successfully applied RKPM to multiple scale analysis. The convergence properties of RKPM and the moving least-squares reproducing kernel particle methods are discussed in [95] and [96], respectively. This is the first presentation of a rigorous mathematical formulation for meshfree methods.

Sukky et al. [97] extended the Reproducing Kernel Particle Method to the computational simulation of large deformation problems. In their work, the Lagrangian description referred to the reference configuration with explicit time integration are employed. For meshfree methods, it is well known that the calculation of shape function and the searching procedure are the most time-consuming processes during computation. It is shown that the formulation in this work is efficient since those procedures are needed only once.

2.3.5. MESHFREE LOCAL PETROV-GALERKIN METHOD AND A LOCAL BOUNDARY INTEGRAL EQUATION METHOD

The above-mentioned meshfree methods are “meshfree” only in terms of interpolation of the field variables as compared to the FEM and BEM. Most of meshfree methods have to use background cell for integration, this makes the method not entirely meshfree.

To overcome the above disadvantages of meshfree method, two truly meshfree methods were proposed, they are the local boundary integral equation (LBIE) method by Zhu and

coworkers [98, 99] and local Petrov-Galerkin method (MLPG) by Atluri and coworkers [100, 101, 102]. In the local boundary integral equation (LBIE) method, the local boundary integral equation is used to construct the values of the unknown function at the point of interest and a new discretization scheme is used to construct the system equation. In the formulation of this work, the requirements for the continuity of the trial function used in an approximation is greatly relaxed and no derivatives of the shape functions are needed in constructing the system stiffness matrix, at least for internal nodes. Compared with the other meshfree techniques based on the Galerkin formulation the LBIE method has several advantages such as the displacement boundary conditions can be enforced directly and no special integration scheme is needed to evaluate the volume and boundary integral. The local boundary integral equation method (LBIE) is extended for solving nonlinear boundary problems and solving problems of elasticity with nonhomogeneous material properties in Zhu et al. [103] and Sladek et al. [104] receptively.

Although the local boundary integral method (LBIE) requires no background cell either for interpolation purposes or for evaluation of the integrals to implement the formulation, and is very efficient in solving linear and nonlinear problems. However, singular integrals appear in the local boundary integral equation. To overcome this shortcoming, Atluri and Zhu [100, 101, 102] developed the Meshfree Local Petrov-Galerkin (MLPG) method. The method is based on a local symmetric weak form and the moving least-squares approximation. The displacement boundary conditions in the MLPG formulation are imposed by a penalty method. All integrals can be easily evaluated over regularly shaped domains and their boundaries with no background mesh

needed. Furthermore no post-smoothing technique is required for computing the derivatives of the unknown variable, as the original solution, using the moving least-squares approximation, is already smooth enough.

Figure 2.4. The support of the MLPG [100].

- (1) The domain of definition of the MLS approximation, Ω_x , for the trial function at any point x is the domain over which the MLS is defined, i.e., Ω_x covers all the nodes whose weight functions do not vanish at x .
- (2) The domain of influence for source point y is the union of all Ω_x ; $\forall x \in \Omega_s$ (taken to be a circular domain of radius r_0 in this paper).
- (3) The support of source point y_i is a sub domain (taken to be a circle of radius r_0 for convenience) in which the weight function w_i corresponding to this node is nonzero. Note that the “support” of v_i is distinct and different from the “domain” of influence of v_i .

Figure 2.4 is the support of the MLPG, where Γ_{su} is a part of the boundary $\partial\Omega_s$ of Ω_s , over which the displacement boundary conditions are specified. In general, $\partial\Omega_s = \Gamma_s \cup L_s$, with Γ_s being a part of the local boundary located on the global boundary and L_s being the other part of the local boundary over which no boundary condition is specified.

Barry [105] suggests a different approach to the integrations in the MLPG. In this work, the test function is taken to be a rational polynomial interpolant. By this way, the local domain is effectively redefined to an n -sided polygon in the 2D space. The sets of such polygons are found by partitioning the global convex domain through a general procedure based on *Voronoi* cells around all nodes. The integration over each polygonal sub-domain is then performed by using a Gaussian quadrature scheme while dividing the polygon into a collection of triangular sub-cells.

The method is called *Wachspress MLPG* (WMLPG) and is successfully applied to four numerical examples in elastostatics. The author points out that the Wachspress interpolants become increasingly more complex for more-sided polygons over which they are defined. The main drawback of the WMLPG appears to be the need for constructing *Voronoi* diagrams and hence loosing the truly meshfree character of the MLPG.

Radek [106] found the controlling factor of the integration accuracy is the accuracy of the MLS-approximation for integrands based on MLS, such as those arising in MLPG. The recommended approach for increasing the overall accuracy in this work is by

adding more MLS nodes and taking advantage of the computationally inexpensive cubature formulae. The author points out two ways to increase the accuracy of the MLS-approximation, by increasing the density of its nodal distribution or by increasing the order of its polynomial basis.

2.3.6. THE POINT INTERPOLATION METHODS [PIMs]

Although the MLPG method promises many advantages, it is difficult to implement displacement boundary conditions and it is computationally expensive due to use of MLS approximation. To overcome the shortcomings of the MLS, Liu and Gu [107,108] proposed the LPIM and PIM for two-dimensional solids, in which, a set of points is used to represent the problem domain. A technique to construct polynomial interpolation functions with delta function property is proposed. The PIM is based on global Galerkin weak form while the LPIM is based on a local weak form developed using the weighted residual method locally. As the shape functions of the LPIM and PIM method have delta function property, the displacement boundary conditions can be implemented as easily as in FEM. Moreover the computing cost is much lower because of the simple interpolation and the reduction in computation of the stiffness matrix. The LPIM is a truly meshfree method, which based on non-element interpolation and non-mesh integration. The LPIM has been extended to 2-D elasto-statics and 1-D fourth order thin beam analyses [109].

Later, based on the boundary node method [110], Gu and Liu [111] proposed a boundary point interpolation method (BPIM). The polynomial interpolants based on

scattered nodes arbitrarily distributed points on the boundary of the problem domain are used to construct the shape functions. The BPIM does not require any element connectivity in constructing system equation and possesses the usual dimensionality advantage. BPIM is computationally much cheaper than BNM due to simpler interpolation scheme and smaller set of system equations.

The point interpolation technique overcomes shortcomings of MLS, and it possesses some distinguished advantages. PIM shape functions possess *Kronecker delta* properties, so the displacement boundary conditions can be easily enforced in PIM, LPIM and BPIM. The integrations of terms involving shape functions and its derivatives can be accurately obtained, because shape functions so-formed are polynomials. The point interpolation technique is also more accurate and easier to implement than the MLS approximation, especially for regular nodal distributions. However, these advantages do not come without any costs. Like other methods that use polynomial as basis functions, it is tricky to choose the basis for interpolation in polynomial basis. If an inappropriate polynomial basis is chosen, it may result in a badly conditioned matrix, which could be even invertible. The radial interpolation method (RPIM) and the matrix triangularization algorithm (MTA) [112,113] are developed to circumvent this problem. The RPIM has been used to analyze buckling of Mindlin plates by Liew and Chen [114].

Liu et al. [115] discuss the properties of PIMs in detail. Some critical issues related to the compatibility of PIMs are studied. The problems of continuity and compatibility of PIMs are unveiled using the curve fittings and the patch tests both theoretically and

computationally. A technique of background cell-based nodal selections and a penalty method are proposed to overcome these problems. The so-called meshfree conforming (CPIM) and non-conforming PIMs (NPIM) are proposed. They point out CPIM can pass the standard patch test exactly. NPIM usually cannot pass the standard patch test because NPIM does not satisfy the compatibility requirement. The presence of polynomial terms in the radial PIM is the necessary condition for passing the patch test. Including linear polynomial, CRPIM can pass patch tests exactly. However, NRPIM usually cannot pass patch test. Although conforming PIMs have better accuracy and convergence than non-conforming PIMs, non-conforming PIMs are also convergent. Owing to the simplicity and efficiency of the NPIM, the NPIM is more efficient than the CPIM in the practical applications although the accuracy of the NPIM may be worse than that of the CPIM.

2.3.7. A POINT COLLOCATION METHOD

Oñate and coworkers [116,117] proposed a finite point method (FPM). In their work, the weighted least-squares interpolation using a fixed Gaussian weighting function and a simple point collocation technique are used for evaluating the approximation integrals. Compared with standard FEM, this method avoids the necessity of mesh generation, and compared with classical FDM, this method is easy to handle the boundary conditions and the non-structured distribution of points. Oñate et al. [118] extend the FPM to 2D and 3D analysis of simple elastic solids.

Aluru and Li [119] have reported satisfactory results for a large class of problems in engineering applications. However the robustness of collocation approach is an issue especially when scattered and random points are used to discretize the governing equations. Benito et al. [120] have reported that the solution of the generalized finite-difference method depends on the number of nodes in the cloud. Later, Jin et al. [121] studied the robustness of collocation method. They show that the robustness of collocation meshfree methods can be improved by ensuring that certain conditions, defined as the positivity conditions, are satisfied when constructing approximation functions and their derivatives. The significance of positivity conditions is pointed out by an error analysis of the finite cloud method, which is a collocation based meshfree method. The techniques are based on modification of weighting functions, to ensure satisfaction of positivity conditions on the approximation function and its derivatives when using a scattered set of points.

2.3.8. RECENT DEVELOPMENTS OF MESHFREE METHOD

A novel true meshfree Hermite-Cloud method is introduced by Li et al. [122]. This method uses the Hermit interpolation theorem to construct the interpolation functions and the point collocation technique to discretize the partial differential equations. It is based on the classical reproducing kernel particle method but uses a fixed reproducing kernel approximation. It is a very efficient numerical method for solving partial differential boundary value problems with high gradient fields.

Rabczuk et al. [123] developed a large deformation method. This method is based on the Krongauz-Belytschko corrected-gradient meshfree method [124] with Lagrangian kernels. The gradient is corrected by a linear transformation so that linear completeness is satisfied. Shepard functions are used as test functions. Quadrature of the Galerkin weak form is performed over the particles and a set of stress points. It is shown that the method is able to deal with unstable phenomena such as shear bands and fracture.

Lam et al. [125] developed a novel local Kriging method which is based on the local weak form of the partial differential governing equations and employs the Kriging interpolation to construct the meshfree shape functions. The local weak form of the partial differential governing equations is derived by the weighted residual method within the simple local quadrature domain. The spline function with continuity is used as weight functions. The major advantage of the Kriging interpolation is that it has vigorous theoretical basis to ensure interpolation accuracy. Although the final shape functions of this method coincide with RPIM [114], the theories and algorithms of RPIM and Kriging interpolation are completely different.

To solve the potential problems in two dimensions, Young et al. [126] proposed a non-singular and boundary-type meshfree method. The solution is represented by a distribution of the kernel functions of double layer potentials. By using the desingularization technique to regularize the singularity and hypersingularity of the kernel functions, the source points can be located on the real boundary and therefore the diagonal terms of influence matrices are determined. The main difficulty of the

coincidence of the source and collocation points then disappears. By employing the two-point function, the off-diagonal coefficients of influence matrices are easily obtained.

Based on the partition of unity method and the finite cover approximation theory Li et al. [127] proposed the meshfree manifold method. Two cover systems, the mathematical cover system and the physical cover system are used in this method. The former one provides the nodes for forming finite covers of the solution domain and the partition of unity functions. The later one describes geometry of the domain and the discontinuous surfaces in the domain. The shape functions are derived by the partition of unity and the finite covers approximation theory. The crack propagation are solved with the method, each node in the affected domain is separated into two or more nodes. All nodes that are not affected by the crack remain unchanged. As a result, arbitrary crack growth can be treated easily.

2.4. COUPLING OF FINITE ELEMENTS WITH MESHFREE METHODS

It is seen from the above literature, the success of the traditional finite element method is primarily due to the fact that computations can be performed flexibly on complex geometries using unstructured computational grids. However, the FEM is not without shortcomings. One of them is that distortion in the mesh that occurs if the body undergoes a significant deformation; this mesh distortion can either terminate the calculation or result in dramatic deterioration of accuracy. Another is the mesh generation process often becomes a serious bottleneck in large-scale parallel computing, especially for problems that require frequent mesh refinement, such as in moving

boundary problems, compressible flows containing shocks, crack propagation problems and large deformation problems. And also there are still some other problems such as discontinuous derivatives of the primary variable at element boundaries.

A variety of meshfree methods have been proposed in recent decades to overcome the shortcomings of FEM. Meshfree methods do not use elements and therefore they do not have difficulties associated with elements. They are specially suited for certain problems, for instance adaptive refinement computations, discontinuous field problems, shape design sensitivity analysis and optimization, large deformation, penetration, etc. Moreover, the interpolation functions in meshfree methods are particularly attractive in the presence of high gradients, concentrated forces, and large deformations. Meshfree methods use shape functions that can be smoother than FEM shape functions which can increase the accuracy and rate of convergence and avoid volumetric locking in near compressible problems.

Meshfree methods, besides many benefits, have shortcomings when compared to the FEM. Computation of meshfree shape functions and their derivatives is usually more complicated and time-consuming than the FEM shape functions, because identification of the nodes affecting a sample point and computation of shape functions are performed in running time. Also calculation of meshfree shape functions is a complicated task in both theory and application. Another problem of meshfree methods is that the background integration grids should be created separately, after distributing the nodes. Usually, creating the integration grids is a sophisticated task and requires human involvement. Therefore almost for every problem one should think about the best

possible integration grids. The same problem exists for choosing basis functions, weight functions and nodal influence domains associated with the moving least-square (MLS) approximation. While in the FEM, shape functions and Gaussian integration points are automatically mapped from the parent element to the physical elements. Meshfree methods need node connectivity to define the interpolations. The accuracy of a meshfree method depends, to a great extent, on the node connectivity. The correct choice of the node connectivity is not an easy task. Most of the shape functions of the meshfree methods lack of the *Kronecker delta* properties, so it is difficult to apply the boundary conditions. Also the meshfree methods are weak at representing the three-dimensional complex shapes required in practical engineering. While the finite element methods are less costly, implement displacement boundary conditions in a simple way. So the meshfree methods, particularly in the current state of development, are still not as fast as finite element methods.

There is a great interest in combining two different numerical methods in order to exploit the potential of each method while avoiding their deficiencies. In a generic approach to couple finite element and meshfree method, the analysis domain is divided into two regions and each method is used in a region where it is most efficient. At the interface between the regions, the continuity of the primary variables and the balance of the secondary variables need to be maintained. Some of these techniques are discussed to follow.

Several techniques have been proposed to couple different numerical methods, Belytschko et al. [128] developed a procedure for coupling the element free Galerkin

method with finite element methods. The coupling is developed so that continuity and consistency are preserved on the interface elements. They do a mixed interpolation in the transition region i.e., in the area where both finite elements and particles have an influence. This mixed interpolation requires the substitution of finite element nodes by particles and the definition of ramp functions. Thus, the transition is of the size of one finite element and the interpolation is linear. Results are presented for both elastostatic and elastodynamic problems, including a problem with crack growth.

Consider the domain with nodal points depicted in Fig. 2.5. The EFG method for Ω_1 will be equivalent with a regular FE method. For all the nodal points in Ω_1 , the weight functions are chosen such that their supports equal the union of the surrounding rectangles. For all the nodal points in Ω_2 , the weight functions are zero on Ω_1 . For Ω_2 , a general EFG approach can be used. Of course, all the weight functions should be chosen such that MLSA is well defined for Ω_2 .

Figure 2.5. Domain handled by EFG method and FE method [128].

A combination of EFG and FE as described above does not work very well, due to the fact that the weight functions for the EFG part have to be zero on the FE part, some weight functions will have relatively small supports with respect to the mesh size of the nodal distribution. As already described, this will lead to shape functions are not convenient for numerical integration.

Hegen [129] couples the finite element domain and the meshfree region with Lagrange multipliers. He achieved consistent coupling by setting the DOI (domain of influence) in the meshfree region adjacent to the FE region such that it coincides with the virtual elements. Here, a new formulation is proposed. His approach follows the ideas of Belytschko et al. [128], generalizes them for any order of interpolation, suppresses the ramp functions, and does not require the substitution of nodes by particles. That is, as many particles as needed can be added where they are needed independently of the adjacent finite element mesh. This is done preserving the continuity of the solution and enforcing uniform consistency for the mixed interpolation.

A mixed hierarchical approximation based on finite elements and meshfree methods is presented by Huerta et al. [130]. Two cases are considered in their paper. The first one couple regions where finite elements or meshfree methods are used to interpolate, by which, continuity and consistency are preserved. The second one enriches a finite element mesh with particles. Thus, there is no need to remesh in adaptive refinement processes. The first case allows implementing displacement boundary conditions in a

standard finite element as reference [128]. A simple generalization avoids the use of any ramp function and the need of substituting nodes per particles. The particles are added arbitrarily in the region of the computational domain where the finite element interpolation is not complete. This ensures continuity of the solution (so no coupling via Lagrange multipliers is imposed) and also enforces a uniform order of consistency (and thus of convergence) everywhere in the computational domain. The second case, enrichment of finite elements with EFG, helps to improve the accuracy of finite elements where needed in an adaptive process without any remeshing. The formulation discussed in the paper is developed for the EFG method. However, its generalization to other particle methods is straightforward.

A collocation method is introduced to couple the FE with the EFG method by Xiao and Dhanasekar [131]. The basic idea is to evaluate the real value of nodes at the interface between the FE and the EFG regions using the MLS interpolant for EFG and then assign this value to the FE nodal set. This method is conveniently implemented by converting the FE element matrices corresponding to its nodal values to matrices relevant to the EFG virtual nodal values.

Liu and Gu [132] developed the coupled MLPG/FE and MLPG/BE methods. Like the previous couple EFG/FE method, in these proposed methods, the problem domain is divided into two parts. The MLPG is used in one part and the FE or BE is used in other parts. The interface elements are defined with shape functions composed of FE and MLPG shape functions along the combined boundary. The shape functions are constructed so that the linear consistency is met exactly.

A formulation that couples the FE and MLPG methods is presented for two dimensional potential problems based on a single variational form by Chen and Raju [133]. In this method, the analysis domain is divided into two regions (FEM region and MLPG region). The trial and test functions are independently chosen in the FE and MLPG regions. The trial function blending from each of these regions is achieved using a “Coons patch” methodology which is widely used in the geometric design.

Michael and Suvranu [134] proposed five different techniques of coupling the method of finite spheres with the finite element method. These include techniques using Lagrange multipliers, penalty formulation and different schemes of replacing the Lagrange multipliers with their physical significance. From an examination of the accuracy of the solutions of some example problems as well as some rough estimates of computational time, the authors recommend the use of Lagrange multipliers as an efficient means of achieving coupling.

Another generic approach to the couple FE and Meshfree method is to construct the shape function by combining the virtues of finite element approximation and meshfree approximation. Some of these methods are discussed in the following:

One class of these methods is partition-of-unity based finite element methods. Melenk and Babuska [135] have proposed the following approximation which they call partition of unity finite element method (PUFEM)

$$u^h(\mathbf{x}) = \sum_I \Phi_I^0(\mathbf{x}) \sum_j \beta_{jI} p_j(\mathbf{x}) \quad (2.9)$$

where β_{j_I} are the unknowns (several per node) and p_j is the basis which will typically include monomial terms up to a certain degree and possibly some enhancement functions.

To improve the conditioning of the discrete system, another form was also proposed such that

$$u^h(\mathbf{x}) = \sum_I \sum_{J: \mathbf{x}_I \in \Omega_J} \Phi_J^0(\mathbf{x}) L_{JI}(\mathbf{x}) u_I = \sum_I \phi_I(\mathbf{x}) u_I \quad (2.10)$$

where

$$\phi_I(\mathbf{x}) = \sum_{J: \mathbf{x}_I \in \Omega_J} \Phi_J^0(\mathbf{x}) L_{JI}(\mathbf{x}) \quad (2.11)$$

and L_{JI} is the Lagrange interpolant that has value 1 at node I and vanishes at all the other interpolation points in patch Ω_J . Thus $L_{JI}(\mathbf{x}_K) = \delta_{IK}$. The ϕ_I satisfy the *Kronecker delta* condition:

$$\phi_I(\mathbf{x}_K) = \delta_{IK} \quad (2.12)$$

This can be shown by

$$\phi_I(\mathbf{x}_K) = \sum_{J: \mathbf{x}_I \in \Omega_J} \Phi_J^0(\mathbf{x}_K) L_{JI}(\mathbf{x}_K) = \sum_{J: \mathbf{x}_I \in \Omega_J} \Phi_J^0(\mathbf{x}_K) \delta_{IK} = \delta_{IK} \quad (2.13)$$

since Φ_J^0 represents a partition of unity, i.e.

$$\sum_J \Phi_J^0 = 1 \quad (2.14)$$

An advantage of this method, besides the *Kronecker delta* properties, is the ease with which arbitrary continuity interpolants can be constructed, since the continuity is determined by that of Φ^0 . A disadvantage, however, is the difficulty of constructing Lagrange polynomials in multi-dimensions.

The essence of the partition of unity method is to take a partition of unity and multiply it with any independent basis to form a new and better basis. This flexibility provides leverage in computation practice. Sometimes the choices of the independent basis can be based on users' prior knowledge and experience about the problem that they are solving.

Some of the important works in this class include partition of unity method (PUM) [136], partition of unity finite-element method (PUFEM) [135], and generalized finite element method (GFEM) proposed by Strouboulis and coworkers [137, 138] and Tian and coworkers [139,140], the *hp*-meshfree cloud method by Liszka et al. [141] and Oden et al. [142]. In these methods, the standard finite element spaces are augmented by adding special functions. These special functions are selected to take advantage of the boundary value problems and are pasted together using the linear finite element hat functions as a partition of unity. The partition of unity based finite element approximation can construct high order global finite element approximation by increasing the order of the locally defined functions without adding new nodes. However the linear dependence problem of the PU based method has been a bottleneck to further progress. This problem has been seen to occur when both the PU functions and the local functions are taken as explicit polynomials. In order to eliminate the linear dependence problem, Oden et al. [142] suggested excluding the space spanned by the terms of finite element shape functions from that of the local approximation functions. The mapped quadrilateral shape functions were used in the work of Strouboulis et al. [137, 138], but they could not eliminate the linear dependencies completely. Tian and Yagawa [139] proposed a 4-node quadratic tetrahedral element and 3-node quadratic

triangular element. They point out the triangular meshes help to avoid the linear dependence problem.

Yagawa [143] proposed another method called node-by-node finite element method (NBN-FEM). This method employs a local mesh generation technique and a node-by-node algorithm for finite element calculations. In this method, the local finite elements are generated around each node, with local mesh data structures and a system of equations created based on these nodes. The NBN-FEM overcomes difficulties arising from the distortion of elements by simply adding or deleting nodes as particle methods. In the meanwhile, it maintains the superior theoretical foundations and the ability to handle complicated boundary shapes in finite element method. This method is particularly advantageous when a parallel computer is employed, as only the distribution of the nodes needs to be considered to perform the parallel remeshing computations when the mesh generation is generally difficult.

Hao and coworkers [144, 145] introduced a new method called the moving particle finite element method (MPFEM) by which the displacement boundary conditions can be handled exactly as in FEM. The fundamental idea of this method is to use a kernel approximation in conjunction with general finite element shape functions. As the general shape functions already satisfy certain reproducing conditions, the kernel only needs to be a partition unity. The shape functions at each point are constructed by using neighboring particles that form element-like configurations that satisfy certain desired reproducing conditions. However, unlike FEM shape functions, the MPFEM shape functions are not used to interpolate values within the domain of the problem, and are

instead constructed at discrete points in the domain. The method was tested in a benchmark beam bending problem, and was shown to give superior performance than the standard FEM element for both the compressible and incompressible cases. For the incompressible case, the MPFEM was shown to alleviate locking, and maintained full rates of convergence in displacement as the mesh was refined.

However, for coarse mesh, this method shows numerical oscillations. Based on the same idea, Liu and coworkers [146, 147, 148, 149] developed a reproducing kernel element method (RKEM). In this new method, the continuous global partition polynomial which is constructed from the each element is patched over the whole domain; and the compactly reproducing kernel function is used to localize the global partition polynomial such that the reproducing condition and owns the interpolation property. By doing so, the smoothness of the reproducing kernel element shape function is determined by the kernel function, which can be chosen as an arbitrary smooth function.

The reproducing kernel element shape function retains the *Kronecker delta* properties under a restriction on the support size of kernel function i.e., the support size is to be chosen small so that the influence of node only covers the elements which include this node. Under this restriction, the numerical solution of RKEM generally has the same convergence rate as that of FEM. The continuity of approximation in the RKEM turns out to be a drawback when used to solve the problem involving material interface. Several techniques are developed to enrich the meshfree approximation by adding

special function that contains discontinuities in the derivatives or by adding additional constraints at the variational level.

Figure 2.6 illustrates the basic concept of NBN-FEM. First, a single node P_i among the distributed nodes is designated a *central node*. A local mesh is then generated around the central node P_i using a local mesh generation technique. Elements formed by the local mesh are called *satellite elements* and are associated with the central node P_i . Nodes of satellite elements, other than the central node, are referred to as *satellite nodes*. Local meshes around each node must be generated under the constraints that the edges of satellite elements do not cross each other, and that the analysis domain is covered completely by local meshes. After local mesh generation, the components of the global coefficient matrix associated with the central node P_i are computed by integrating all the satellite elements around P_i .

Tian and Yagawa [139] developed a robust and high accuracy finite element with only corner nodes using a partition-of-unity-based finite-element approximation.

The standard finite element approximation expressed as

$$u^h = \sum_i N_i u_i \quad (2.15)$$

where u_i is the nodal unknowns of the node i , which are nodal displacement in the finite element method.

Figure 2.6. Node-based finite element data structures [143].

The nodal displacement is further approximated by

$$u_i = u_i(x) = \mathbf{F}_i \mathbf{D}_i \quad (2.16)$$

where $\mathbf{F}_i$ is the matrix of the local functions associated with the node i , and $\mathbf{D}_i$ is the vector of the new nodal unknowns. Then the PU-based generalized finite element approximation is constructed by Substituting (2.16) into (2.15).

$$u^h = \sum_i N_i u_i = \sum_i N_i u_i(x) = \sum_i N_i \mathbf{F}_i \mathbf{D}_i \quad (2.17)$$

The nodal unknown introduced by this specially defined local function has clear and definite physical meaning. The physically defined local function proposed was encapsulated into the term of the *generalized node*. The usage of the generalized node, as its name was chosen to suggest, entails the simple replacement of the node of the low-order element with a *super node*. A 4-node quadratic tetrahedron, GNTet4, and a 3-node quadratic triangle, GNT3, based on the idea of the PU based approximation were developed. Numerical calculations demonstrate that the new elements are robust,

insensitive to mesh distortion, and offer quadratic accuracy, while also keeping mesh generation extremely simple. The GNT3, with only corner nodes, is comparable with T6 and Q8, and perform better than Q8 with regard to sensitivity to mesh distortion; GNTet4, with four corner nodes offers even better accuracy and performance than TET10, and HEX20 (especially for distorted meshes). One demerit of their new elements is the extra nodal unknowns. However, this disadvantage is compensated for by high accuracy, robustness and their remarkably straightforward mesh generation.

2.5. MOTIVATION FOR THE PRESENT WORK

The literature review reveals that both FEM and meshfree methods have their own merits and demerits. The recent efforts to develop coupled methods combining FEM and meshfree are attractive as they attempt to synergize the merits of the two methods while trying to avoid the demerits. The coupled methods can be grouped into two categories. The first involves dividing the problem domain into two parts, one modeled by FEM and the other by meshfree method. The second involves methods that combine the shape functions of FEM and meshfree methods in some way. In the former, modeling the interface conditions between the two domains is recognized as a difficult problem. The second is free from this problem and is therefore attractive. This forms the motivation for the present work. Different methods of combining the FEM and meshfree shape functions are explored in this thesis:

One method of combining is to use FEM and meshfree shape functions as test and test trial functions. Such an approach falls under the broad category of Petrov-Galerkin

formulation and has been successfully used in developing *unsymmetric elements* [68, 69]. This method is explored in Chapter 3.

Another method is to use *composite shape functions* obtained by multiplying the shape functions of FEM and meshfree method appropriately, and is explored in Chapter 4 for static analysis and in Chapter 5 for vibration analysis and in Chapter 6 for geometrically nonlinear analysis. Chapter 7 gives a critical review of this method.

CHAPTER 3. TWO HYBRID FE-MESHFREE METHODS: FE-EFG AND FE-PIM

In this chapter, two hybrid FE-meshfree methods, viz., FE-EFG and FE-PIM, are developed for two dimensional linear elastic problems. The former combines the concepts of finite element method and element free Galerkin (EFG) method and the latter combines that of finite element method and point interpolation method (PIM). In the FE-EFG method, FEM shape functions are used as test functions and EFG shape functions as trial functions. Similarly, in the FE-PIM method, the FEM and PIM shape functions are used as the test and trial functions respectively. Several numerical examples are presented to assess the efficacy of the proposed methods.

3.1. INTRODUCTION

One of the important observations from the literature review (Chapter 2) is that inter-element compatibility properties and higher order completeness properties of shapes functions play an important role in developing a better performing element. The completeness requirements that the shape functions must satisfy so as to reproduce a given polynomial displacement field under an arbitrary but permissible geometrical shape of the element may be expressed in the form:

$$\sum_{i=1}^n M_i x_i^p y_i^q = x^p y^q, \quad p, q = 0, 1, 2, \dots \text{ where } n \text{ is the number of monomial terms in the}$$

displacement field (and therefore equal to the number of nodes in the element), and

$x^p y^q$ is a typical monomial term. As for the continuity requirements, the shape functions must be such that the displacement interpolation satisfies the appropriate intra- and inter-element continuity requirements of the problem.

Ideally, a single set of shape functions satisfying all the completeness requirements as well as the inter-element displacement continuity all along the element boundary is desirable. However, such shape functions are hard to obtain. In order to circumvent this problem, Rajendran and Liew [68] developed an 8-node quadrilateral plane finite element, which is based on a novel formulation called “unsymmetric formulation”. This formulation comes under the broad class of Petrov-Galerkin methods, and is characterized by the use of two different sets of shape functions for test and trial functions. Their element stiffness integral is of the form $\mathbf{K} = \int_{V^{(e)}} \mathbf{B}_L^T \mathbf{D} \mathbf{B}_R dV^{(e)}$ where $\mathbf{B}_L$ and $\mathbf{B}_R$ are two different strain-displacement matrices and $\mathbf{D}$ is the usual constitutive matrix. Classical FE parametric shape functions are used for constructing $\mathbf{B}_L$ whereas a different set of shape functions called the metric shape functions are used for constructing $\mathbf{B}_R$ matrix. The former satisfy the continuity requirements while the latter are chosen to satisfy all the completeness requirements necessary for reproducing exactly a quadratic displacement field even under distorted meshes. The numerical experiments confirm that the elements (based on this idea of combining two shape functions) are distortion-tolerant even under severe mesh distortions.

In the light of the unsymmetric formulation of Rajendran and Liew [68], it is motivating to investigate whether meshfree shape functions can be used to construct $\mathbf{B}_R$ matrix

while still using FE parametric shape functions to construct $\mathbf{B}_L$ matrix as in the unsymmetric formulation. This is the focus of investigation in this chapter.

The investigations are reported in two sections. In section 3.2, FE-EFG method is developed wherein FE parametric shape functions and EFG shape functions are used to construct $\mathbf{B}_L$ and $\mathbf{B}_R$ respectively. In section 3.3, FE-PIM method is developed wherein FE parametric shape functions and PIM shape functions are used to construct $\mathbf{B}_L$ and $\mathbf{B}_R$, respectively.

Figure 3.1. The influence domain of test and trial functions.

Ω^{trial} is the influence domain of trial function and Ω^{test} is influence domain of test function

3.2. FE-EFG METHOD

Unlike in the conventional Galerkin method where the trial and the test functions are chosen from the same space, the Petrov-Galerkin method uses the trial and test functions from different spaces. In the FE-EFG method, the trial functions are based on

EFG shape functions, while the test functions are based on FEM shape functions. The influence domain of the test and trial functions are chosen as in Fig. 3.1.

3.2.1. EFG SHAPE FUNCTIONS

The shape functions in EFG method are derived by means of a weighted moving least-square (MLS) approximation which is described in the following section.

3.2.1.1. EFG shape functions in one dimension

The MLS approximation described in this section is as given in reference [150].

The MLS interpolant $u^h(x)$ is defined by

$$u^h(x) = \sum_{j=0}^m p_j(x) a_j(x) = \mathbf{p}^T(x) \mathbf{a}(x) \quad (3.1)$$

where $p_j(x)$, $j = 1, 2, \dots, m$ are the basis functions in the space coordinate x and m is the order of the polynomial $p_j(x)$. As an illustration, for a linear basis ($m = 2$)

$$\mathbf{p}^T(x) = [1 \quad x] \quad (3.2)$$

The coefficients $a_j(x)$ are functions of x and determined by minimizing a weighted L_2 norm respect to $\mathbf{a}(x)$,

$$\begin{aligned} J &= \sum_{I=1}^n w(x - x_I) [u^h(x) - u_I]^2 \\ &= \sum_{I=1}^n w(x - x_I) [\mathbf{p}^T(x_I) \mathbf{a}(x) - u_I]^2 \end{aligned} \quad (3.3)$$

where n is the number of nodes in the neighborhood of x . The minimum of J with respect to $\mathbf{a}(x)$ leads to the following equations:

$$\mathbf{A}(x)\mathbf{a}(x) = \mathbf{B}(x)\mathbf{u} \quad (3.4)$$

$$\mathbf{a}(x) = \mathbf{A}^{-1}(x)\mathbf{B}(x)\mathbf{u} \quad (3.5)$$

where

$$\begin{aligned} \mathbf{A} &= \sum_{I=1}^n w(x-x_I) \mathbf{p}(x_I) \mathbf{p}^T(x_I) \\ &= w(x-x_1) \begin{bmatrix} 1 & x_1 \\ x_1 & x_1^2 \end{bmatrix} + w(x-x_2) \begin{bmatrix} 1 & x_2 \\ x_2 & x_2^2 \end{bmatrix} + \cdots w(x-x_n) \begin{bmatrix} 1 & x_n \\ x_n & x_n^2 \end{bmatrix} \end{aligned} \quad (3.6)$$

and

$$\begin{aligned} \mathbf{B}(\bar{x}) &= [w(x-x_1)\mathbf{p}(x_1), w(x-x_2)\mathbf{p}(x_2), \cdots, w(x-x_n)\mathbf{p}(x_n)] \\ &= \begin{bmatrix} w(x-x_1) \begin{bmatrix} 1 \\ x_1 \end{bmatrix} & w(x-x_2) \begin{bmatrix} 1 \\ x_2 \end{bmatrix} & \cdots & w(x-x_n) \begin{bmatrix} 1 \\ x_n \end{bmatrix} \end{bmatrix} \end{aligned} \quad (3.7)$$

$$\mathbf{u}^T = [u_1, u_2, \cdots, u_n] \quad (3.8)$$

The MLS approximants can be obtained by substituting Equation (3.5) into Equation (3.1)

$$u^h(x) = \sum_{I=1}^n \Phi_I(x) u_I = \mathbf{\Phi}(x) \mathbf{u} \quad (3.9)$$

where

$$\Phi_I(x) = \sum_{j=0}^m p_j(x) (\mathbf{A}^{-1}(x) \mathbf{B}(x))_{jI} = \mathbf{p}^T \mathbf{A}^{-1} \mathbf{B}_I \quad (3.10)$$

It should be noted that EFG shape functions, $\Phi_I(x)$, defined in Equation (3.10) do not satisfy the *Kronecker Delta* criterion, so the nodal parameters u_I are not the nodal values $u^h(x_I)$.

3.2.1.2. EFG shape functions in two dimensions

In two dimensions, x and y components of displacement need to be considered, but the same shape functions can be used for discretization.

$$u^h(\mathbf{x}) = \mathbf{p}^T(\mathbf{x})\mathbf{a}(\mathbf{x}) \quad (3.11)$$

where the scalar variable x simply becomes a vector, $\mathbf{x}^T = [x \ y]$, and the $\mathbf{p}$ vector for a linear basis is given by

$$\mathbf{p}^T(\mathbf{x}) = [1 \ x \ y] \quad (3.12)$$

Similar to Equation (3.6), the $\mathbf{A}$ matrix for a linear basis in two dimensions is given by

$$\begin{aligned} \mathbf{A}(\mathbf{x}) &= \sum_{I=1}^n w(\mathbf{x} - \mathbf{x}_I) \mathbf{p}(\mathbf{x}_I) \mathbf{p}^T(\mathbf{x}_I) \\ &= w(\mathbf{x} - \mathbf{x}_1) \begin{bmatrix} 1 & x_1 & y_1 \\ x_1 & x_1^2 & x_1 y_1 \\ y_1 & x_1 y_1 & y_1^2 \end{bmatrix} + w(\mathbf{x} - \mathbf{x}_2) \begin{bmatrix} 1 & x_2 & y_2 \\ x_2 & x_2^2 & x_2 y_2 \\ y_2 & x_2 y_2 & y_2^2 \end{bmatrix} \\ &\quad + \cdots + w(\mathbf{x} - \mathbf{x}_n) \begin{bmatrix} 1 & x_n & y_n \\ x_n & x_n^2 & x_n y_n \\ y_n & x_n y_n & y_n^2 \end{bmatrix} \end{aligned} \quad (3.13)$$

$$\text{And } \mathbf{B}(\mathbf{x}) = [w(\mathbf{x} - \mathbf{x}_1)\mathbf{p}(\mathbf{x}_1), w(\mathbf{x} - \mathbf{x}_2)\mathbf{p}(\mathbf{x}_2), \cdots w(\mathbf{x} - \mathbf{x}_n)\mathbf{p}(\mathbf{x}_n)]$$

(3.14)

where $\mathbf{x} = [x \ y]$ and $\mathbf{x}_I = [x_I \ y_I]$. The two dimensional shape functions are given by

$$\Phi_I(\mathbf{x}) = \sum_{j=0}^m p_j(\mathbf{x}) (\mathbf{A}^{-1}(\mathbf{x}) \mathbf{B}(\mathbf{x}))_{jI} = \mathbf{p}^T \mathbf{A}^{-1} \mathbf{B}_I \quad (3.15)$$

3.2.2. CHOICE OF WEIGHT FUNCTIONS

3.2.2.1. Weight function in one dimension

In this thesis, for 1D problem, the cubic spline weight function is used. The weight function for node I is defined as

$$w(x - x_I) \equiv w(r) = \begin{cases} \frac{2}{3} - 4r^2 + 4r^3 & \text{for } r \leq \frac{1}{2} \\ \frac{4}{3} - 4r + 4r^2 - \frac{4}{3}r^3 & \text{for } \frac{1}{2} < r \leq 1 \\ 0 & \text{for } r > 1 \end{cases} \quad (3.16)$$

where $r = \frac{d_I}{d_{mI}}$, $d_I = \|x - x_I\|$, x is the point at which the weight is to be determined and x_I is the nodal coordinate. The parameter d_{mI} is used to define the size of the domain of influence, in which the weight functions and shape functions are nonzero.

$$d_{mI} = d_{\max} c_I \quad (3.17)$$

where $d_{\max}$ is a scaling parameter, the recommended value of which is typically 2.0 - 4.0 for static analysis so as to include enough neighbor nodes for $\mathbf{A}$ to be regular. The parameter c_I is the distance between the nearest neighboring nodes.

3.2.2.2. Weight function in two dimensions

Tensor product of cubic spline weight function is used in 2D problems.

$$w(\mathbf{x} - \mathbf{x}_I) = w(r_x) \cdot w(r_y) = w_x \cdot w_y \quad (3.18)$$

where $w(r_x)$ or $w(r_y)$ is given by Equation (3.16) with r replaced by r_x or r_y and r_x and r_y are given by

$$r_x = \frac{\|x - x_I\|}{d_{mx}} \quad (3.19a)$$

$$r_y = \frac{\|y - y_I\|}{d_{my}} \quad (3.19b)$$

where

$$d_{mx} = d_{\max} \cdot c_{xI} \quad (3.20a)$$

$$d_{my} = d_{\max} \cdot c_{yI} \quad (3.20b)$$

c_{xI} and c_{yI} are determined at a particular node by including enough neighbor nodes so that the $\mathbf{A}$ matrix is non-singular everywhere in the domain, and thus invertible.

3.2.3. PARAMETRIC SHAPE FUNCTIONS

3.2.3.1. FE Parametric shape functions in one dimensions

For the present FE-EFG method, the well-known **parametric** shape functions are used to construct test shape function, and are reproduced below for convenience:

$$N_1 = \frac{1-\xi}{2}, \quad N_2 = \frac{1+\xi}{2} \quad (3.21)$$

The test function written in the form

$$v(\xi) = [N_1, N_2] \begin{Bmatrix} v_1 \\ v_2 \end{Bmatrix} \equiv [\mathbf{N}] \{ \mathbf{v}^e \}$$

(3.22)

where v_1 and v_2 are the weights associated with the nodes.

The variation of derivative of test functions is written as

$$\begin{aligned}\delta v_{,x} &= \delta \frac{d\mathbf{v}}{dx} = \delta \frac{d}{dx} \left([\mathbf{N}] \{ \mathbf{v}^e \} \right) = \frac{d}{dx} \left([N_1, N_2] \begin{Bmatrix} \delta v_1 \\ \delta v_2 \end{Bmatrix} \right) \\ &= \left[-\frac{1}{L}, \frac{1}{L} \right] \begin{Bmatrix} v_1 \\ v_2 \end{Bmatrix} \equiv [\mathbf{B}_L] \{ \delta \mathbf{v}^e \}\end{aligned}\quad (3.23)$$

where the symbol δ is the variational operator and

$$[\mathbf{B}_L] = \left[-\frac{1}{L}, \frac{1}{L} \right] \quad (3.24)$$

where $\mathbf{N}$ is a matrix of parametric shape functions, $\mathbf{v}^e$ is a vector of arbitrary nodal weight values.

3.2.3.2. FE Parametric shape functions in two dimensions

The usual parametric shape functions of the bilinear quadrilateral finite element are used for the purpose:

$$N_i = \frac{1}{4} (1 + \xi_i \xi) (1 + \eta_i \eta) \quad (3.25)$$

where i refers to a node number, and ξ_i and η_i represent the coordinates of the nodes in ξ and η natural coordinates. The test functions and the corresponding strain and stress fields are written as

$$\mathbf{v} = \begin{Bmatrix} \delta v_u(\xi, \eta) \\ \delta v_v(\xi, \eta) \end{Bmatrix} = \mathbf{N} \mathbf{v}^{(e)} \quad (3.26)$$

where $\mathbf{v}^{(e)}$ is a vector arbitrary nodal weight values.

$$\mathbf{v}^{(e)} = [v_{u1} \quad v_{v1} \quad v_{u2} \quad v_{v2} \quad v_{u3} \quad v_{v3} \quad v_{u4} \quad v_{v4}]^T \quad (3.27)$$

$\mathbf{N}$ is a matrix of parametric shape functions.

$$\mathbf{N} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \quad (3.28)$$

The variation of derivative of test functions is written as

$$\delta \mathbf{v}_{,x} = \mathbf{B}_L \delta \mathbf{v}^{(e)} \quad (3.29)$$

where

$$\mathbf{B}_L = \begin{bmatrix} N_{1,x} & 0 & N_{2,x} & 0 & N_{3,x} & 0 & N_{4,x} & 0 \\ 0 & N_{1,y} & 0 & N_{2,y} & 0 & N_{3,y} & 0 & N_{4,y} \\ N_{1,y} & N_{1,x} & N_{2,y} & N_{2,x} & N_{3,y} & N_{3,x} & N_{4,y} & N_{4,x} \end{bmatrix} \quad (3.30)$$

where $N_{i,x}$ and $N_{i,y}$ ($i=1,2,3,4$) are the Cartesian derivatives of parametric shape functions.

3.2.4. STIFFNESS MATRIX AND LOAD VECTORS IN ONE DIMENSION

Consider the following governing differential equation in one dimensional domain,

$$0 \leq x \leq L:$$

$$Eu_{,xx} + b = 0 \quad \text{in } \Omega = (0, L) \quad (3.31)$$

with the boundary conditions:

$$Eu_{,x} n = \bar{t} \quad \text{on } \Gamma_t$$

(3.32)

$$u = \bar{u} \quad \text{on } \Gamma_u$$

(3.33)

where $u \equiv u(x)$ is the displacement, E is Young's modulus, b is the body force per unit volume, Γ_t is the natural boundary, Γ_u is the displacement boundary, and $\bar{t}$ and $\bar{u}$ are the prescribed traction and displacement.

The virtual work principle for the above problem is given by

$$\int_0^L \delta v_{,x}^T E u_{,x} dx - \int_0^L \delta v^T b dx - \delta v^T \bar{t} \Big|_{\Gamma_l} - \delta \lambda^T (u - \bar{u}) \Big|_{\Gamma_u} - \delta v^T \lambda \Big|_{\Gamma_u} = 0 \quad (3.34)$$

For the present work, two node elements are used for discretizing the problem domain. The parametric shape functions shown in Equation (3.21) are used to construct the test function. EFG shape functions shown in Equation (3.10) are used to construct the trial functions. The final discrete equations can be obtained by substituting the trial functions and test functions into the weak form (3.34):

$$\begin{bmatrix} \mathbf{K} & \mathbf{G} \\ \mathbf{G}^T & 0 \end{bmatrix} \begin{pmatrix} \mathbf{u} \\ \lambda \end{pmatrix} = \begin{pmatrix} \mathbf{f} \\ \mathbf{q} \end{pmatrix} \quad (3.35)$$

$$\mathbf{K}_{IJ} = \int_0^L N_{I,x}^T E \Phi_{J,x} dx \quad (3.36)$$

$$\mathbf{G}_{Ik} = -N_K \Big|_{\Gamma_{ul}} \quad (3.37)$$

$$\mathbf{f}_I = N_I t_x \Big|_{\Gamma_l} + \int_0^L N_I b dx \quad (3.38)$$

$$q_K = -\bar{u}_K \quad (3.39)$$

At each quadrature point, x_Q , all the nodes whose domain of influence contains the point, are first determined. These are referred to as “support nodes” for the quadrature point. The equation corresponding to a given node I is obtained by assembling the contributions due to all the nodes, J , in its support. The numerical example problems are discussed in section 3.2.6.

3.2.5. DISCRETE EQUATION IN TWO DIMENSIONS

The FE-EFG formulation for 2D problems is developed in this section. The boundary value problem for a linear elastic body on the domain Ω defined by the boundary Γ in equilibrium under the action of body forces is written as

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{b} = 0 \text{ in } \Omega \quad (3.40)$$

where $\boldsymbol{\sigma}$ is the stress tensor, which corresponds to the displacement field $\mathbf{u}$ and $\mathbf{b}$ is body force vector. The boundary conditions are as follows:

$$\boldsymbol{\sigma} \cdot \mathbf{n} = \bar{\mathbf{t}} \text{ on } \Gamma_t \quad (3.41)$$

$$\mathbf{u} = \bar{\mathbf{u}} \text{ on } \Gamma_u \quad (3.42)$$

where Equation(3.41) and Equation(3.42) are the prescribed traction and prescribed displacement, respectively, and $\mathbf{n}$ is the unit normal to the boundary Γ_t .

The variational (or weak) form of the equilibrium equation is written as

$$\begin{aligned} \int_{\Omega} \delta(\nabla_s \mathbf{v}^T) : \boldsymbol{\sigma} d\Omega - \int_{\Omega} \delta \mathbf{v}^T \cdot \mathbf{b} d\Omega - \int_{\Gamma_t} \delta \mathbf{v}^T \cdot \bar{\mathbf{t}} d\Gamma - \\ \int_{\Gamma_u} \delta^T \lambda (\mathbf{u} - \bar{\mathbf{u}}) d\Gamma - \int_{\Gamma_u} \delta \mathbf{v}^T \cdot \lambda d\Gamma = 0, \forall \delta \mathbf{v} \in H^1, \delta \lambda \in H^0 \end{aligned} \quad (3.43)$$

where

$$\lambda(\mathbf{x}) = N_I(s) \lambda_I, \mathbf{x} \in \Gamma_u \quad (3.44a)$$

$$\delta \lambda(\mathbf{x}) = N_I(s) \delta \lambda_I, \mathbf{x} \in \Gamma_u \quad (3.44b)$$

and H^1 and H^0 are Hilbert spaces of degree one and zero, respectively. As the trial functions (i.e., EFG shape functions defined by Equation (3.15)) do not possess the *Kronecker delta* properties, the displacement boundary conditions cannot be applied directly. For the present work, they are imposed using Lagrange multipliers. $N_I(s)$ is a

Lagrange interpolation function and s is the arc length along the boundary. The repeated indices in Equations (3.44a) and (3.44b) stand for summation.

Substituting the test and trial functions into the weak form Equation (3.43),

$$\begin{aligned} \delta\{\mathbf{v}\}^T \int_{\Omega} \mathbf{B}_L^T \mathbf{D} \mathbf{B}_R \mathbf{u} d\Omega - \int_{\Omega} \mathbf{N}^T \mathbf{b} d\Omega - \int_{\Gamma_t} \mathbf{N}^T \bar{\mathbf{t}} d\Gamma - \\ \int_{\Gamma_u} \delta\lambda_l^T (\mathbf{u} - \bar{\mathbf{u}}) d\Gamma - \int_{\Gamma_u} \mathbf{N}^T \mathbf{N} \lambda d\Gamma = 0, \forall \delta\mathbf{v} \in H^1, \delta\lambda \in H^0 \end{aligned} \quad (3.45)$$

which can be written in matrix form as

$$\begin{bmatrix} \mathbf{K} & \mathbf{G} \\ \mathbf{H} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \lambda \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \mathbf{q} \end{Bmatrix} \quad (3.46)$$

where

$$\mathbf{K}_{IJ} = \int_{\Omega} \mathbf{B}_{LI}^T \mathbf{D} \mathbf{B}_{RJ} d\Omega \quad (3.47)$$

$$\mathbf{G}_{IK} = - \int_{\Gamma_u} \mathbf{N}_I \mathbf{N}_K d\Gamma \quad (3.48)$$

$$\mathbf{H}_{IK} = - \int_{\Gamma_u} \Phi_I \mathbf{N}_K d\Gamma \quad (3.49)$$

$$\mathbf{f}_I = \int_{\Gamma_t} \mathbf{N}_I \bar{\mathbf{t}} d\Gamma + \int_{\Omega} \mathbf{N}_I \mathbf{b} d\Omega \quad (3.50)$$

$$\mathbf{q}_k = - \int_{\Gamma_u} \mathbf{N}_K \bar{\mathbf{u}} d\Gamma \quad (3.51)$$

$$\mathbf{B}_{LI} = \begin{bmatrix} N_{I,x} & 0 \\ 0 & N_{I,y} \\ N_{I,y} & N_{I,x} \end{bmatrix} \quad (3.52)$$

$$\mathbf{B}_{RJ} = \begin{bmatrix} \Phi_{J,x} & 0 \\ 0 & \Phi_{J,y} \\ \Phi_{J,y} & \Phi_{J,x} \end{bmatrix} \quad (3.53)$$

$$\mathbf{N}_K = \begin{bmatrix} N_K & 0 \\ 0 & N_K \end{bmatrix} \quad (3.54)$$

$$\mathbf{\Phi}_I = \begin{bmatrix} \Phi_I & 0 \\ 0 & \Phi_I \end{bmatrix} \quad (3.55)$$

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad \text{for plane stress} \quad (3.56)$$

where E and ν are the Young's modulus and Poisson's ratio, respectively.

Because different sets of shape functions are used as test and trial functions, the stiffness matrix, $\mathbf{K}$, is unsymmetric, and hence an unsymmetric solver is needed for the solution of resulting equations. For numerical integration, the isparametric mapping of geometry is carried out as in FEM using the parametric shape functions, and the numerical integration of stiffness and load integrals performed using Gaussian quadrature.

MATLAB programs have been developed for solving 1D and 2D problems using the proposed FE-EFG method. The program listing is too long and hence is not included here. The flowchart of 2D FE-EFG program is as follows:

Flowchart of 2D FE-EFG program

Define the physical dimensions and material properties.

Define the plane stress D matrix.

Set up the nodal coordinates.

Identify the domain of influence for each node for a uniform mesh.

Set up the quadrature cells in the domain.

Set up Gauss points, weight, and Jacobian for each cell.

Loop over Gauss points:

Identify the nodes in the neighborhood of the Gauss point.

Determine weights, EFG shape functions, FEM shape functions corresponding to nodes 1 to n using Equations (3.15) and (3.25).

Assemble $\mathbf{B}_{LI}$ and $\mathbf{B}_{RJ}$ matrix, and compute $\mathbf{K}_{IJ}$ using Equation (3.47).

Add $\mathbf{K}_{IJ}$ contributions to $\mathbf{K}$ matrix.

Identify the nodes on the traction boundary and displacement boundary.

Set up Gauss points along traction boundary and displacement boundary.

Integrate forces along traction boundary to obtain $\mathbf{f}$ vector using Equation (3.50).

Integrate along displacement boundary to form the $\mathbf{G}$ using Equation (3.48) matrix and the $\mathbf{q}$ vector using Equation (3.51).

Solve for nodal parameters u_I using Equation (3.46).

3.2.6. ONE DIMENSIONAL NUMERICAL EXAMPLE

In order to validate the proposed FE-EFG method, a simple 1-D numerical example is considered first. A one dimensional bar of unit length subjected to a linear body force of magnitude x shown in Fig. 3.2. The bar is fixed at the left end, and the right end is traction free. The bar has unit cross sectional area, and a modulus of elasticity of E units.

Figure 3.2. One dimensional example problem: a bar of unit length subjected to a linear body force and fixed at the point $x = 0$.

The equilibrium equation and boundary conditions for this problem are given by

$$Eu_{,xx} + x = 0 \quad 0 < x < 1 \quad (3.57a)$$

$$u(0) = 0 \quad (3.57b)$$

$$u_{,x}(1) = 0 \quad (3.57c)$$

The exact solution to the above problem is given by

$$u(x) = \frac{1}{E} \left[\frac{1}{2}x - \frac{x^3}{6} \right] \quad (3.58)$$

)

Figures 3.3(a) and (b) shows a comparison of the axial displacement and strain obtained using EFG method and present method as well as exact solution. The results shown are for eleven nodes along the length of the bar. One point integration is used for evaluating the stiffness matrix and force vector. To begin with, the scaling parameter $d_{\max}$ is set as 2 for the computations. It can be seen that the computed displacements with the EFG and the present method are very close to the exact ones. The computed strains are also

very close except at the natural boundary. (As is well known, this difference can be reduced by refining the mesh near the natural boundary.)

Figures 3.4(a) and (b) give the computed results when the scaling parameter $d_{\max}$ is increased to 3. It can be seen that the displacements computed with the present method are still continue to be close to the exact values and are more accurate than those computed with EFG method. The strain field computed with the present method is also more accurate, particularly at the right and left boundaries.

Further increasing the value of scaling parameter $d_{\max}$ to 4, the results given by present method are much better than those given by EFG method, as shown in Figs. 3.5(a) and (b). In this case, the displacement field computed with present method is still close to the exact solution while that of EFG is very bad. The strain field computed with the present method also exhibits a better accuracy, particularly near the boundaries, whereas the strain field computed by EFG involves large errors.

(a)

(b)

Figure 3.3. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 2.

CHAPTER 3. TWO HYBRIDFE-MESHFREE METHODS: FE-EFG AND FE-PIM

Figure 3.4. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 3.

(a)

(b)

Figure 3.5. Comparison of the computed axial displacements and strains with exact ones for one dimensional bar problem. The parameter $d_{\max}$ is set to 4.

3.2.7. TWO DIMENSIONAL NUMERICAL EXAMPLES

3.2.7.1. A cantilever beam under pure bending

Figure 3.6. A cantilever beam under pure bending.

The purpose of the test problem is to check whether the FE-EFG method can reproduce quadratic displacement field. A beam of length L subjected to a pure bending moment at the free end as shown in Fig. 3.6 is considered. The moment is applied as a linearly varying distributed load of load intensity $\sigma_x = \frac{M}{I} y$. The beam is of height D and unit thickness, and is assumed to be in a state of plane stress. The material, geometric and loading parameters are taken as $E = 3.0 \times 10^7$, $\nu = 0.3$, $D = 12$, $L = 48$, and $M = -24000$. The parameter $d_{\max}$ is set to 3.5 for the present computation. A 2×2

Gauss integration is used for all computations reported in this chapter. The exact solution for this problem is given by [151]

$$u = \frac{M}{EI}xy \quad (3.59)$$

$$v = -\frac{\mu M}{2EI}y^2 - \frac{M}{2EI}x^2 \quad (3.60)$$

$$\sigma_x = \frac{M}{I}y \quad (3.61)$$

$$\sigma_y = 0, \sigma_{xy} = 0 \quad (3.62)$$

Table 3.1 lists the computed vertical displacement at point A of the beam along with the exact vertical displacement. It is seen that the results obtained using EFG method and FE-EFG method are better than those obtained from QUAD4, while the QM6 [18] can reproduce the exact results. In this chapter, the results of QUAD4 and EFG have been computed by the author's computer program whereas the QM6 have been obtained using ANSYS.

Table 3.1. Vertical displacement (*mm*) at the tip of the beam under pure bending.

Mesh	QUAD4	QM6 [18]	EFG	FE-EFG
2×2	-2.497	-6.4	-6.406	-6.706
2×8	-5.709	-6.4	-6.419	-6.381
4×16	-6.194	-6.4	-6.407	-6.394
6×24	-6.301	-6.4	-6.403	-6.398
8×32	-6.342	-6.4	-6.402	-6.399
Exact solution = -6.4				

Figure 3.7 shows the contour plots of σ_x stress for this problem. A 8×2 mesh is used for plotting the stresses. As this problem is a constant bending moment problem, the

normal stress must have a linear distribution across the depth of the beam conforming to Equation (3.61). If a given solution method were to faithfully reproduce the exact solution, the stress contours must be straight and horizontal, and also continuous across the element boundaries. As seen from Fig. 3.7(a), FE-EFG and EFG give continuous stress contours (Figs. 3.7 (c,d)) although the contour lines show a trace of waviness. The QUAD4 gives straight and continuous stress contours.

Figure 3.8 shows the contour plots of σ_x stress obtained using a distorted mesh. As seen from Fig. 3.8, the EFG as well as FE-EFG continue to give continuous contour plots for the distorted mesh too, although the contour lines exhibit some waviness. However, QUAD4 gives a highly discontinuous contour plot. It turns out that QUAD4 is highly sensitive to mesh distortion and EFG and FE-EFG are less sensitive to mesh distortion.

CHAPTER 3. TWO HYBRIDFE-MESHFREE METHODS: FE-EFG AND FE-PIM

Figure 3.7. The contour plots of σ_x for the cantilever beam under pure bending (Regular mesh).

Figure 3.8. The contour plots of σ_x for the cantilever beam under pure bending (Distorted mesh).

3.2.7.2. A cantilever beam under shear force

Figure 3.9. A cantilever beam subjected to a shear force.

The purpose of this test problem is to assess the reproducibility of the cubic displacement field. The cantilever beam shown in Fig. 3.9 is considered here for the purpose and a shear force P is applied at the free end as a parabolic traction (conforming to Equation (3.68)). The material, geometry and load parameters are taken as $E = 3.0 \times 10^7$, $\nu = 0.3$, $D = 12$, $L = 48$ and $P = 1000$. The exact solution to this problem is given by Timoshenko and Goodier [151]:

$$u = -\frac{Py}{6EI}[(6L - 3x)x + (2 + \nu)(y^2 - \frac{D^2}{4})] \quad (3.63)$$

$$v = -\frac{P}{6EI}[3\nu y^2(L - x) + (4 + 5\nu)\frac{D^2 x}{4} + (3L - x)x^2] \quad (3.64)$$

where the second moment of area I of the beam is given by

$$I = \frac{D^3}{12} \quad (3.65)$$

The stresses corresponding to the displacements (3.63) and (3.64) are

$$\sigma_x(x, y) = -\frac{P(L - x)y}{I} \quad (3.66)$$

$$\sigma_y = 0 \quad (3.67)$$

$$\sigma_{xy}(x, y) = -\frac{Py}{2I} \left[\frac{D^2}{4} - y^2 \right] \quad (3.68)$$

Table 3.2 lists the computed vertical displacement at point A of the beam. A good agreement is seen between the exact solution and that of FE-EFG. The results obtained by the FE-EFG are more efficient than that QUAD4 and comparable to that of QM6 [18] and EFG. The results of QM6 reported here have been obtained using ANSYS (PLANE42 element with bubble function option).

Table 3.2. Vertical displacement (*mm*) at the tip of the beam under a tip shear force.

Mesh	QUAD4	QM6	EFG	FE-EFG (Present)
2×2	-3.475	-8.129	-7.785	-7.931
2×8	-7.964	-8.788	-8.875	-8.823
4×16	-8.612	-8.863	-8.91	-8.889
6×24	-8.759	-8.883	-8.905	-8.896
8×32	-8.816	-8.892	-8.903	-8.898
Exact solution = -8.9				

Figure 3.10 shows the variation of the error of the vertical displacement at A with different values of scaling parameter $d_{\max}$. It is seen that if inappropriate support size is adapted, both the EFG and FE-EFG method will give bad results.

Figures 3.11(a) and (b) give a comparison of the stresses computed at the center of the beam ($x = 24$) with the exact stresses. A 10×6 mesh is used. An excellent agreement is seen between the result of the present method, EFG and the analytical solution.

Figure 3.10. Variation of the percent error of displacement with different scaling parameter d_{max} .

(a) Normal stress

(b) Shear stress

Figure 3.11. Comparison of normal stress and shear stress at the centre of the beam ($x = 24$).

Figure 3.12 shows the contour plots of σ_x stress obtained with a 8×2 regular mesh. As the normal stress distribution for this problem is a bilinear function (Equation (3.66)), its magnitude should vary along the length and depth of the beam as shown in Fig. 3.12 (a). For an element that can faithfully reproduce the exact solution, the stress bands must be smooth and continuous across the element boundaries. Figure 3.12 shows that the stress bands given by the EFG and FE-EFG are smooth and continuous across the inter-element boundaries whereas that of QUAD4 are highly discontinuous.

Figure 3.13 shows the contour plot of σ_x obtained with an 8×2 distorted mesh. It is seen that, although the mesh is severely distorted, the stress contours given by the FE-EFG and EFG are still continuous and smooth whereas that of QUAD4 are highly discontinuous. This suggests that the performance of FE-EFG and EFG are less sensitive to mesh distortion for the present problem.

Figure 3.12. The contour plots of σ_x for cantilever beam under tip shear force (Regular mesh).

Figure 3.13. The contour plots of σ_x for cantilever beam under tip shear force (Distorted mesh).

3.3. A FE-PIM METHOD

In the previous section, the FE-EFG method has been developed. The FEM shape functions were used as test functions and the EFG shape functions were used as trial functions. The MLS approximation is used in EFG methods to construct shape functions. However, there are some disadvantages in using MLS, i.e. the shape functions using the MLS approximation lack Kronecker properties. Hence, it is difficult to apply displacement boundary conditions. Liu and Gu [107] used a point interpolation method (PIM) wherein the shape functions satisfy Kronecker delta properties. (The point interpolation technique uses the polynomial basis and scattered nodes.) Therefore, the displacement boundary conditions can be easily enforced. The PIM is also more accurate and easier to implement than the EFG method. In view of their relative merits, the PIM shape functions appear more attractive to serve as trial functions than EFG shape functions. This is explored in this section.

3.3.1. PIM SHAPE FUNCTIONS

The derivation of the PIM shape functions in this section follows the work of Liu and Gu [107]. Consider a function $u(x, y)$ defined in domain Ω discretized by a set of field nodes. The PIM interpolation, $u(x, y)$ is written as

$$u(x, y) = \sum_{i=1}^n p_i(x, y) a_i = \mathbf{p}(x, y) \mathbf{a} \quad (3.69)$$

)

where

$$\mathbf{p}^T(x, y) = [1, x, y, xy, x^2, y^2, x^2y, xy^2, x^3, y^3, \dots] \quad (3.70)$$

CHAPTER 3. TWO HYBRIDFE-MESHFREE METHODS: FE-EFG AND FE-PIM

is a vector of monomials in the co-ordinates space, n is the number of nodes in the neighborhood of node x , and

$$\mathbf{a}^T = [a_1, a_2, a_3, \dots, a_n] \quad (3.71)$$

is a vector of arbitrary constants.

By enforcing the satisfaction of Equation (3.69) at the n nodes surrounding a given point x_Q , the resulting system of n equations can be written in the following matrix form as

$$\mathbf{u}^e = \mathbf{P}_Q \mathbf{a} \quad (3.72)$$

where

$$\mathbf{u}^e = [u_1, u_2, u_3, \dots, u_n]^T \quad (3.73)$$

$$\mathbf{P}_Q^T = [\mathbf{p}(x_1, y_1), \mathbf{p}(x_2, y_2), \mathbf{p}(x_3, y_3), \dots, \mathbf{p}(x_n, y_n)] \quad (3.74)$$

The set of nodes n mentioned above are nothing but the nodes in the support of a given node when x_Q refers to the given nodal point. If x_Q refers to the Gauss point, the set of nodes refers to the nodes in the domain of influence of Gauss point x_Q . The domain of influence includes all the nodes for which x_Q in its support.

From Equation (3.72),

$$\mathbf{a} = \mathbf{P}_Q^{-1} \mathbf{u}^e \quad (3.75)$$

Hence, the PIM interpolate $u(x, y)$ can be expressed as

$$u(x, y) = \Phi(x, y) \mathbf{u}^e \quad (3.76)$$

where the interpolation function $\Phi(x)$ is defined by

$$\Phi(x, y) = \mathbf{p}^T(x, y) \mathbf{P}_Q^{-1} = [\phi_1(x, y), \phi_2(x, y), \phi_3(x, y), \dots, \phi_n(x, y)] \quad (3.77)$$

The interpolation function $\phi_i(x, y)$ obtained through the above procedure satisfies the conditions:

$$\phi_i(x = x_i, y = y_i) = 1, \quad i = 1 - n \quad (3.78a)$$

$$\phi_j(x = x_i, y = y_i) = 0, \quad j \neq i \quad (3.78b)$$

$$\sum_{i=1}^n \phi_i(x, y) = 1 \quad (3.78c)$$

Therefore, the interpolation functions have the much desired *Kronecker delta* properties, and hence the displacement boundary conditions can be easily imposed in PIM. Equation (3.78c) ensures that the interpolation functions can represent the rigid body motion.

It is possible that $\mathbf{P}_Q^{-1}$ in Equation (3.75) does not exist for nodal arrangements. Whenever singularity is encountered, nodal perturbation, i.e., the method of moving some or all of the nodes by a small distance is used to avoid the singularity of $\mathbf{P}_Q$ [107].

3.3.2. DISCRETE EQUATION IN TWO DIMENSIONS

In the FE-PIM method, the PIM shape functions are used as trial functions, while the FEM shape functions are continued to be used as test functions as in FE-EFG. The influence domains of the trial and test functions are defined as in Fig. 3.14. Unlike the PIM method, where different nodes have different influence supports (Fig. 3.16), in the present method (Fig. 3.14), the four nodes of an element have the same supports. During computation, this leads to a reduction in computational time as compared to PIM

because it avoids the time consuming step of searching for nodes in the influence domain of each Gauss point separately.

Figure 3.14. Domain of influence of FE-PIM method.

Ω^{test} is the domain influence for the test functions, and Ω^{trial} is the domain of influence for the trial functions at any Gauss point

Figure 3.15. Domain of influence of PIM* method

Figure 3.16. Domain of influence of PIM method

Ω_a is the domain of influence of Gauss point a, and Ω_b is the domain of influence of Gauss point b

Using the PIM shape functions as trial functions and following a similar discretization approach as in FE-EFG method (except that the boundary conditions are applied directly as in FEM instead of using Lagrange multipliers), the following discrete equations are obtained:

$$\mathbf{K} \mathbf{U} = \mathbf{F} \quad (3.79)$$

$$\mathbf{K}_{IJ} = \int_{\Omega} (\mathbf{B}_{LI})^T \mathbf{D} (\mathbf{B}_{RJ}) \quad (3.80)$$

$$\mathbf{f}_I = \int_{\Gamma} \mathbf{N}_I^T \bar{\mathbf{t}} d\Gamma + \int_{\Omega} \mathbf{N}_I \mathbf{b} d\Omega \quad (3.81)$$

where

$$\mathbf{N}_I = \begin{bmatrix} N_I & 0 \\ 0 & N_I \end{bmatrix} \quad (3.82)$$

$$\mathbf{B}_{LI} = \begin{bmatrix} N_{I,x} & 0 \\ 0 & N_{I,y} \\ N_{I,y} & N_{I,x} \end{bmatrix} \quad (3.83)$$

$$\mathbf{B}_{RJ} = \begin{bmatrix} \Phi_{J,x} & 0 \\ 0 & \Phi_{J,y} \\ \Phi_{J,y} & \Phi_{J,x} \end{bmatrix} \quad (3.84)$$

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \quad \text{for plane stress} \quad (3.85)$$

The shape functions N_i are already defined in Equation (3.25). Here again, as in FE-EFG, the stiffness matrix, $\mathbf{K}$, is unsymmetric because different sets of shape functions are used as test and trial functions, and hence an unsymmetric solver is needed for the solution of resulting equations. For numerical integration again, the isparametric mapping of geometry is carried out as in FEM using the parametric shape functions, and

the numerical integration of stiffness and load integrals performed using Gaussian quadrature.

In PIM, the test and trial functions are chosen from the same space. For numerical testing, PIM results will also be presented for comparison purposes. However, for implementing PIM, a different definition of an influence domain is used here from the ones found in the literature. All the Gauss points in an element have the same influence domain instead of different ones. Hence, the symbol PIM* is used here to denote the fact that it is modified PIM. The definition of the the influence domain of PIM* and PIM can be seen in Figs. 3.15 and 3.16.

3.3.3. THE ALGORITHM OF PRESENT METHOD

The present method is implemented in MATLAB programming language. The program listing is too long and hence is not included here. The pseudo code of the hybrid FE-PIM method is as follows:

Loop over cells of domain

Loop over quadrature points x_Q in cell i :

According to the definition of domain of influence of x_Q (see Fig.

3.14) and find the nodes included in the domain;

Compute B_{Li} and B_{Rj} at the quadrature points according to Equations (3.83) and (3.84);

Evaluate the contribution to the stiffness matrix and load vectors.

End quadrature point loop

End cell loop.

3.3.4. TEST PROBLEMS

3.3.4.1. A cantilever beam under a pure bending moment

To test the reproducibility of quadratic displacement by the FE-PIM method, the beam in Fig. 3.6 is considered again using the same geometric and load parameters. Table 3.3 compares the results obtained using QUAD4, QM6, EFG, PIM^{*} and FE-PIM with the exact vertical displacement given by Equation (3.60). It is observed that QM6, FE-PIM and PIM^{*} are able to reproduce the exact solution while QUAD4 is not.

Table 3.3. Vertical displacement (*mm*) at the tip of the beam under pending bending moment.

Mesh	QUAD4	QM6	PIM [*]	FE-PIM
2×2	-2.497	-6.4	-6.4	-6.4
2×8	-5.709	-6.4	-6.4	-6.4
4×16	-6.194	-6.4	-6.4	-6.4
6×24	-6.301	-6.4	-6.4	-6.4
Exact = -6.4				

Figure 3.17 shows the contour plots of σ_x stress under pure bending. As this problem is a constant bending moment problem, the normal stress distribution is linear across the beam cross section as can be seen from Equation (3.61). If a solution can faithfully reproduce the exact solution, the stress contours must appear straight and horizontal, and also continuous across the element boundaries. Although PIM^{*} and FE-PIM give equally results, PIM^{*} performs slightly inferior when distorted mesh used. This is demonstrated in the results to be presented next.

Figure 3.18 gives the contour plots of σ_x stress for a distorted mesh. The plots show the FE-PIM gives straight contour lines similar to exact solution despite mesh distortion. QUAD4 gives highly discontinuous contours while PIM* gives somewhat better contours.

The high distortion tolerance of FE-PIM is result of appropriate choice of test and trial functions. As long as the test functions (FE shape functions in this case) are compatible and the trial functions (PIM shape functions in this case) have higher order completeness, the formulations have in general high tolerance to mesh distortion. This has been extensively studied in references [68, 69]. The results obtained here seem to confirm the same. The FE-EFG has a distortion tolerance but not as much as FE-PIM (Fig. 3.18). This partially because the trial functions (EFG shape functions) do not have higher order completeness with respect to monomial terms x^2 , xy , y^2 . In addition, and there is difficulty in applying displacement boundary conditions due to the lack of Kronecker delta properties.

CHAPTER 3. TWO HYBRIDFE-MESHFREE METHODS: FE-EFG AND FE-PIM

Figure 3.17. The contour plots of σ_x for cantilever beam under pure bending (Regular mesh).

Figure 3.18. The contour plots of σ_x for cantilever beam under pure bending (Distorted mesh).

3.3.4.2. A cantilever beam under a shear force at the tip

To test the reproducibility of cubic displacement field as well as the mesh distortion sensitivity of the FE-PIM method, the tip shear force problem shown in Fig. 3.9 is considered here.

Table 3.4 compares the vertical tip displacement (at point A) given by QUAD4, QM6, PIM* and FE-PIM results with the exact vertical displacement given by Equation (3.64). It is observed that the converged displacement given by QM6, FE-PIM or PIM* is slightly different from that of the exact solution. Although the difference is small, it is a defect of FE-PIM. The method developed in Chapter 4 circumvents this problem.

Table 3.4. Vertical displacement (mm) at the tip of the beam under a tip shear force.

Mesh	QUAD4	QM6	PIM*	FE-PIM
2×2	-6.033	-8.129	-7.676	-8.032
2×8	-7.992	-8.788	-8.922	-8.93
4×16	-8.718	-8.863	-8.985	-8.988
6×24	-8.875	-8.883	-8.996	-9.0
Exact: -8.9				

Figure 3.19 shows the contour plots of σ_x stress obtained with 8×2 mesh. As the σ_x stress distribution is a bilinear function. Hence σ_x should vary along the length and depth of the beam as in Fig. 3.19(a). PIM* and FE-PIM are seen to yield fairly accurate stress contours. However, QUAD4 does not give continuous contour lines.

Figure 3.20 shows the contour plots for a distorted mesh. QUAD4 gives a highly discontinuous contour plot. Although the contour plots of FE-PIM and PIM^* are not as smooth as in Fig. 3.19, the limited smoothness of contour plots does reveal a better reliability of FE-PIM and PIM^* compared to QUAD4 when the mesh is distorted.

Figure 3.19. The contour plots of σ_x for cantilever beam under tip shear force (Regular mesh).

Figure 3.20. The contour plots of σ_x for cantilever beam under tip shear force (Distorted mesh).

3.4. CONCLUSION

In this chapter, firstly, the FE-EFG method has been developed based on the concept unsymmetric formulation. The FEM shape functions have been used as test functions and EFG shape functions as trial functions. The FE-EFG method, developed based on this method, has demonstrated good tolerance to mesh distortions.

However, the lack of *Kronecker delta* properties of the EFG shape functions has made it difficult to apply displacement boundary conditions. To solve this problem, a FE-PIM method has been proposed, in which the PIM shape functions were used as trial function instead of EFG shape functions. Compared to the EFG shape functions, PIM shape functions possess the much desired *Kronecker delta* properties and thus, the displacement boundary conditions could be easily enforced. Furthermore, the PIM is more accurate and easier to implement than the MLS approximation. For regular meshes, the FE-PIM performs similar to the PIM method. For distorted meshes, however, the FE-PIM performs superior to PIM as well as QUAD4 element.

However, the FE-PIM has also some drawbacks. For example, the basis functions to construct PIM shape functions need to be chosen carefully. If an inappropriate polynomial basis is chosen, it may result in a badly conditioned or singular interpolation ($\mathbf{P}$) matrix. This happens particularly when the mesh is regular (with no distortions). Another drawback is that the result of the FE-PIM method is not accurate enough for the tip shear problem under distorted mesh (Fig. 3.20). These problems motivated further investigation into developing better hybrid method. This method will be discussed in Chapter 4.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Two 4-node quadrilateral elements, labelled FE-LSPIM QUAD4 and FE-LSPIM QUAD4U, are developed by combining the shape functions of finite element method (FEM) and that of least-square point interpolation method (LSPIM). The objective is to synergise the attractive features of finite element and meshfree methods using the concepts of partition of unity finite element methods (PUFEM). The quadrilateral finite element shape functions are used for the PU and the least-square point interpolation shape functions for the local approximation. Although the concept of PUFEM is already known, the present choice of shape functions for the local approximation is new. The present choice of shape functions enables the proposed element inherit the compatibility property of FEM shape functions and the quadratic polynomial completeness property of LSPIM shape functions. The FE-LSPIM QUAD4U, which is an unsymmetric version of FE-LSPIM QUAD4, is formulated rather differently; the shape functions of classical QUAD4 element of FEM (denoted as QUAD4 hereinafter) are used as test functions and the shape functions of the FE-LSPIM QUAD4 are used as trial functions. Numerical results on the bench mark problems show that the proposed elements have good tolerance to distorted meshes.

4.1. INTRODUCTION

In the previous chapter (Chapter 3), two hybrid FE-meshfree methods, labeled as FE-EFG and FE-PIM, were proposed. These methods employ two different sets of shape functions to serve as test and trial functions, and their formulation is based on virtual work principle. These two hybrid FE-meshfree methods showed good tolerance to mesh distortions. However, they are not completely free from drawbacks. For example, the unsymmetry of the stiffness matrix is a serious disadvantage for vibration analysis as this can lead to complex eigenvalues. Furthermore, enforcing the displacement boundary conditions in FE-EFG method is difficult because the EFG shape functions, which are used as trial functions in FE-EFG method, lack of *Kronecker delta* properties. Although, the FE-PIM method is free from this difficulty, the choice of suitable polynomial basis for trial function is a tricky issue. If an inappropriate polynomial basis is chosen, it results in a badly conditioned $\mathbf{P}$ matrix and this leads to interpolation failure. Furthermore, the results given by the FE-PIM are not accurate enough.

To circumvent the above disadvantages of the FE-EFG and FE-PIM while still maintaining their advantages, a method called FE-LSPIM is developed in this chapter. Two new quadrilateral elements, labelled FE-LSPIM QUAD4 and FE-LSPIM QUAD4U, are proposed based on this method.

The FE-LSPIM is based on the concept of partition of unity (PU) finite element method, which comes under the category of partition of unity method (PUM). The PUM was originally proposed by Babuska and Melenk [135]. A PU is a concept whereby a

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

domain is covered with overlapping patches (sub-domains, Ω_c), of which, each is associated with a function $N_i(\mathbf{x})$, which is non-zero in Ω_c and has the property.

$\sum_{i=1}^N N_i(\mathbf{x}) = 1$. The displacement field, u^h , is written as $u^h(\mathbf{x}) = \sum_{i=1}^N N_i(\mathbf{x})L_i(\mathbf{x})$, where

the L_i are the local approximations of the exact displacement field, $u(\mathbf{x})$, centered about the nodes, $\mathbf{x}_i$. The later equation is the starting point of the PUM.

In the proposed FE-LSPIM, FEM shape functions will be used for PU and the least-square point interpolation shape functions will be used for local approximation. The motivation for this choice is that the former enables the inter-element compatibility property to be satisfied as in FEM while the latter enables higher order completeness (reproducibility) of the interpolation.

The formulation of FE-LSPIM QUAD4 is presented in section 4.2. This element can be looked upon as a new QUAD4 element with *composite shape functions* instead of the classical isoparametric shape functions. The composite shape functions are obtained by a product of FEM and least-square PIM shape functions, and hence the proposed element is labeled as the FE-LSPIM QUAD4.

The element is characterized by the following three features:

1. The discretization of the domain is similar to that of the FEM. The problem domain is discretised with 4 node quadrilateral shaped elements for displacement interpolation as well as numerical integration.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

2. The nodal displacement at node i ($i = 1, 2, 3, 4$) of the element is obtained as a least-square interpolation of the displacements of all the nodes in the support (neighbourhood) of node i (Fig. 4.1). This aspect rather resembles that of the meshfree method called the *point interpolation method* (PIM) [107].
3. The displacement within any element is interpolated in terms of the nodal displacement interpolations obtained as in step 2 above. The classical isoparametric finite element shape functions of QUAD4 are used for the purpose. This aspect resembles that of the FEM. As a result, the effective shape functions of the proposed method turn out to be a product of FEM shape functions and PIM shape functions.

The following are the merits of the proposed FE-LSPIM QUAD4 element:

1. The global stiffness equations are obtained by assembling the element stiffness matrices as in FEM. Hence, the assembly and equation-solution routines of a FE code need only a slight modification in implementing the proposed method.
2. The composite shape functions possess the much desired *Kronecker-delta* property.
3. In addition to the Kronecker-delta property, the composite functions also possess the *compatibility property* (i.e., the shape function of a node takes a value of unity at that node and zero at all edges not containing this node).

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

4. The numerical results for benchmark problems show that the present element not only gives accurate results for regular meshes but also exhibits high tolerance to mesh distortions.
5. The accuracy of meshfree methods depends to a great extent on the node connectivity, and correct choice of nodal connectivity is crucial to get accurate results. In the present method, the nodal connectivity is easily established using element connectivity data.
6. The proposed method inherits the advantages of PU-FEM [135] wherein high order global approximation can be obtained by simply increasing the order of the local approximation without necessarily adding new nodes to the underlying mesh.

The FE-LSPIM QUAD4U element is developed in the section 4.3. The shape functions of the FE-LSPIM QUAD4 are used as trial functions and the QUAD4 shape functions are used as test functions.

Numerical examples to study the efficiency of the proposed elements are considered in section 4.4. Important observations from this chapter are summarized in section 4.5.

4.2. FORMULATION OF FE-LSPIM QUAD4

4.2.1. SHAPE FUNCTIONS FOR THE FE-LSPIM QUAD4

The shape functions for the element are constructed using the classical isoparametric shape functions of QUAD4 and the shape functions of the least-square point interpolation method.

Similar to the formulation of isoparametric elements, the displacement interpolation at the interior of an element is first written in the form

$$u(x, y) = \mathbf{N} \mathbf{u}^e \quad (4.1)$$

where

$$\mathbf{N} = [N_1 \quad N_2 \quad N_3 \quad N_4] \quad (4.2)$$

is the shape function matrix with shape functions, which are the well known parametric shape functions of QUAD4 element, defined as

$$N_1 = \frac{1}{4}(1 - \xi)(1 - \eta), \quad N_2 = \frac{1}{4}(1 + \xi)(1 - \eta) \quad (4.3, 4.4)$$

$$N_3 = \frac{1}{4}(1 + \xi)(1 + \eta), \quad N_4 = \frac{1}{4}(1 - \xi)(1 + \eta) \quad (4.5, 4.6)$$

and

$$\mathbf{u}^e = \{u_1(x, y) \quad u_2(x, y) \quad u_3(x, y) \quad u_4(x, y)\}^T \quad (4.7)$$

wherein $u_i(x, y)$, $i = 1, 2, 3, 4$, are *nodal displacement functions* which simply reduce to nodal displacement values at nodal locations, i.e., $u_i(x_i, y_i) \equiv u_i$, $i = 1, 2, 3, 4$.

Thus, although Equation (4.1) resembles that of a classical finite element formulation, there is an important difference. In the classical finite element formulation, $\mathbf{u}^e$ is simply

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

a vector of nodal displacements values, but here it is a vector of nodal displacement functions.

The major challenge in the derivation of the FE-LSPIM QUAD4 is to obtain appropriate nodal displacement functions, $u_i(x_i, y_i)$, satisfying the condition, $u_i(x_i, y_i) \equiv u_i$, $i = 1, 2, 3, 4$. For the present element, this is achieved using the least-square point interpolation method (LSPIM) to be described in section 4.2.2. Using this method, the nodal displacement function $u_i(x, y)$ of a typical node i is interpolated in terms of the nodal displacements of all nodes in the *support of node i* (Fig. 4.1) as

$$u_i(x, y) = \Phi_i \mathbf{u}_i, \quad i = 1, 2, 3, 4 \quad (4.8)$$

$$\Phi_i = [\Phi_1^i \quad \Phi_2^i \quad \Phi_3^i \quad \cdots \quad \Phi_N^i] \quad (4.9)$$

$$\mathbf{u}_i = [u_1 \quad u_2 \quad u_3 \quad \cdots \quad u_N]^T \quad (4.10)$$

where Φ_i is the LSPIM nodal shape function array associated with the support of node i , $\mathbf{u}_i$ is a vector of nodal displacement of all the nodes in the support of a node i (i.e., in domain Ω_i , Fig. 4.1), and N is the total number of nodes in the support of node i . The index I is used to refer to a typical node in the support domain while the index i is reserved to refer to a typical node of the quadrilateral element ($1 \leq i \leq 4, 1 \leq I \leq N$). Note that N , the number of nodes in the support of every node i , may not be constant. N may vary with i depending on the support size specified for node i .

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.1. Definition of nodal support of element 1-2-3-4.

Figure 4.2. Element support, $\Omega = \Omega_1 \oplus \Omega_2 \oplus \Omega_3 \oplus \Omega_4$.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Now, the concept of *element support* is introduced, Ω , which is the union of the supports of all nodes of an element, as shown in Fig. 4.2. Let M be the total number of nodes in the *element support* and the index J refer to a typical node in the *element support* ($1 \leq J \leq M$). The four equations of Equation (4.8) are combined in terms of the element support as follows:

$$\mathbf{u}^e = \begin{Bmatrix} u_1(x, y) \\ u_2(x, y) \\ u_3(x, y) \\ u_4(x, y) \end{Bmatrix} = \begin{Bmatrix} \Phi_1 \mathbf{u}_1 \\ \Phi_2 \mathbf{u}_2 \\ \Phi_3 \mathbf{u}_3 \\ \Phi_4 \mathbf{u}_4 \end{Bmatrix} = \begin{bmatrix} \Phi_1^1 & \Phi_1^2 & \Phi_1^3 & \Phi_1^4 & \Phi_1^5 & \dots & \Phi_1^M \\ \Phi_2^1 & \Phi_2^2 & \Phi_2^3 & \Phi_2^4 & \Phi_2^5 & \dots & \Phi_2^M \\ \Phi_3^1 & \Phi_3^2 & \Phi_3^3 & \Phi_3^4 & \Phi_3^5 & \dots & \Phi_3^M \\ \Phi_4^1 & \Phi_4^2 & \Phi_4^3 & \Phi_4^4 & \Phi_4^5 & \dots & \Phi_4^M \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ \vdots \\ u_M \end{Bmatrix} \equiv \Phi \mathbf{u} \quad (4.11)$$

$4 \times M \quad M \times 1$

where Φ is the *LSPIM elemental shape function matrix* associated with the *element support* and $\mathbf{u}$ is the vector of nodal x -displacements of all nodes in the element support. Note that some of the entries Φ_j^i in Equation (4.11) may be zero if the given node, J , is not in the support of node i . The procedure to construct Φ from Φ_i is described in detail in section 4.2.3 for a typical illustrative mesh. It is assumed that the first four columns of the Φ matrix in Equation (4.11) correspond to the four nodes of the element under consideration and the other columns correspond to the rest of the nodes in the element support. Consistent with this definition, the first four entries of vector $\mathbf{u}$ are the displacements of four nodes of the element under consideration and the rest correspond to the other nodes in the *element support*.

Using Equation (4.11) in Equation (4.1),

$$u(x, y) = \mathbf{N}(\Phi \mathbf{u}) = (\mathbf{N}\Phi) \mathbf{u} \equiv \Psi \mathbf{u} \quad (4.12)$$

Similarly, $v(x, y)$ can be expressed as

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$v(x, y) = (\mathbf{N}\Phi)\mathbf{v} \equiv \Psi\mathbf{v} \quad (4.13)$$

where $\mathbf{v}$ is a vector of nodal y -displacement, defined as

$$\mathbf{v} = [v_1 \quad v_2 \quad v_3 \quad v_4 \quad | \quad v_5 \quad \cdots \quad v_M]^T \quad (4.14)$$

From Equation (4.12) and (4.13), the shape function matrix for the FE-LSPIM QUAD4 element is identified as

$$\Psi_{1 \times M} \equiv [\Psi_1, \Psi_2, \Psi_3, \Psi_4, \dots, \Psi_M] = \mathbf{N}_{1 \times 4} \Phi_{4 \times M} \quad (4.15)$$

which is simply the product of the shape function matrix, $\mathbf{N}$, the isoparametric QUAD4 element given by Equation (4.2) and the LSPIM elemental shape function matrix, Φ , defined in Equation (4.11).

4.2.2. DERIVATION OF NODAL SHAPE FUNCTION ARRAY, Φ_i

As illustrated in section 4.2.3, the entries of matrix Φ are to be filled out using the entries of Φ_i ($i = 1, 2, 3, 4$). This section is devoted to the derivation of the nodal shape function array Φ_i defined in Equation (4.8).

It is well known that the PIM shape functions possess the *Kronecker delta* properties and hence facilitate easy imposition of displacement boundary conditions similar to FEM. Thus, on the face of it, the PIM shape functions appear to be a suitable candidate for constructing Φ_i . However, the PIM [107] shape functions cannot be straight away used to construct Φ_i , because the interpolation matrix $\mathbf{P}$ may become singular for certain nodal arrangements and this will lead to interpolation failure in deriving PIM shape functions. The number of basis terms and the number of nodes in the support have to be carefully adjusted so as to avoid the possible singularity of interpolation matrix. In order

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

to circumvent this trouble, a least-square point interpolation method (LSPIM) is proposed here to construct Φ_i . Although this approach turns out to be successful in avoiding the singularity of the interpolation matrix, the process of least-square minimization spoils the *Kronecker delta* properties. Nevertheless, a special technique is adopted here to restore the *Kronecker delta* properties and, at the same time, avoid the singularity of the interpolation matrix. The LSPIM nodal shape functions Φ_i are derived as follows:

The nodal displacement function for the i^{th} node of an element is first expressed as

$$u_i(x, y) = \mathbf{p}(x, y) \mathbf{a} \quad (4.16)$$

where

$$\mathbf{p}(x, y) = [1 \quad x \quad y \quad x^2 \quad xy \quad y^2 \quad x^2y \quad xy^2] \quad (4.17)$$

and

$$\mathbf{a} = [a_1 \quad a_2 \quad a_3 \quad a_4 \quad \cdots \quad a_8]^T \quad (4.18)$$

is a vector of arbitrary constants.

Recall that N refers to the number of nodes in the support of a given node i of an element. Node i is also included in this count. For the sake of convenience, the first node in the list is treated as node i . In this chapter, we generally consider a support size with $N > 8$ because eight monomial terms are used in the basis (Equation (4.17)). Under certain conditions, the support may contain less number of nodes. For example, the boundary nodes 5 and 6 in Fig. 4.3 have only 3 and 6 nodes, respectively, in their support. As an eight term basis is used in this thesis, the resulting $\mathbf{P}$ matrix becomes non-square and hence finding the inverse becomes difficult. To solve this problem, a

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

second order nodal connectivity is used to define the support for all the nodes on the boundary. The definition of the second order nodal connectivity is shown in Fig. 4.4.

For some problems, the number of nodes in the support may be less than 8 (see e.g., Fig. 4.9) even after considering second order nodal connectivity. In such cases, an 8-term basis cannot be used as it will lead to a singular $\mathbf{P}$ matrix (therefore singular $\bar{\mathbf{P}}$ matrix) during the least-square fit procedure to be discussed in the next paragraph. Hence, a six term basis (i.e., $1, x, y, xy, x^2, y^2$) will be used for solving such problems.

$$\mathbf{u}_i = \mathbf{P}\mathbf{a} \quad (4.19)$$

Figure.4.3. Definition of nodal support for boundary nodes 5 and 6 considering *first order nodal connectivity*, i.e., nodes of all elements connected to a given node.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure.4.4. Definition of nodal supports for boundary nodes 5 and 6 considering *second order nodal connectivity*, i.e., nodes of all elements connected to the nodes of all elements connected to a given node

where

$$\mathbf{P} = \begin{bmatrix} 1 & x_i & y_i & x_i^2 & x_i y_i & x_i^2 & x_i^2 y_i & x_i y_i^2 \\ 1 & x_2 & y_2 & x_2^2 & x_2 y_2 & x_2^2 & x_2^2 y_2 & x_2 y_2^2 \\ 1 & x_3 & y_3 & x_3^2 & x_3 y_3 & x_3^2 & x_3^2 y_3 & x_3 y_3^2 \\ 1 & x_4 & y_4 & x_4^2 & x_4 y_4 & x_4^2 & x_4^2 y_4 & x_4 y_4^2 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & x_N & y_N & x_N^2 & x_N y_N & x_N^2 & x_N^2 y_N & x_N y_N^2 \end{bmatrix} \quad (4.20)$$

and

$$\mathbf{u}_i = [u_i \quad u_2 \quad u_3 \quad u_4 \quad \dots \quad u_N]^T \quad (4.21)$$

is a vector of nodal displacements. The classical least-square solution, viz., $\mathbf{a} = (\mathbf{P}^T \mathbf{P})^{-1} \mathbf{P}^T \mathbf{u}_i$, of the above over-constrained system may not guarantee exact satisfaction of any of the equations of the system (4.19). Such a least-square solution is useless for the present work because nonsatisfaction of the first equation would then mean $u_i \neq \mathbf{p}(x_i, y_i) \mathbf{a}$, which in turn, means that $u_i(x_i, y_i) \neq u_i$, i.e., the least-square

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

interpolant at node i is not equal to the nodal displacement. Thus *Kronecker delta* properties of interpolation are lost. However, in order to restore the *Kronecker delta* properties so that the imposition of displacement boundary conditions will be easy as in the FEM, a different approach to least-square solution is adopted: Out of the N equations of system (4.19), the first equation (corresponding to node i) is assumed to be satisfied exactly and the rest in the least-square sense. This is done by using the first equation to eliminate a_1 from the rest of the equations:

$$a_1 = u_i - (a_2 x_i + a_3 y_i + a_4 x_i^2 + a_5 x_i y_i + a_6 y_i^2 + a_7 x_i^2 y_i + a_8 x_i y_i^2) \quad (4.22)$$

Substituting for a_1 in Equation (4.19), the reduced system of equations is obtained:

$$\bar{\mathbf{u}}_i = \bar{\mathbf{P}} \bar{\mathbf{a}} \quad (4.23)$$

where

$$\bar{\mathbf{P}} = \begin{bmatrix} x_2 - x_i & y_2 - y_i & x_2^2 - x_i^2 & x_2 y_2 - x_i y_i & y_2^2 - y_i^2 & x_2^2 y_2 - x_i^2 y_i & x_2 y_2^2 - x_i y_i^2 \\ x_3 - x_i & y_3 - y_i & x_3^2 - x_i^2 & x_3 y_3 - x_i y_i & y_3^2 - y_i^2 & x_3^2 y_3 - x_i^2 y_i & x_3 y_3^2 - x_i y_i^2 \\ x_4 - x_i & y_4 - y_i & x_4^2 - x_i^2 & x_4 y_4 - x_i y_i & y_4^2 - y_i^2 & x_4^2 y_4 - x_i^2 y_i & x_4 y_4^2 - x_i y_i^2 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ x_N - x_i & y_N - y_i & x_N^2 - x_i^2 & x_N y_N - x_i y_i & y_N^2 - y_i^2 & x_N^2 y_N - x_i^2 y_i & x_N y_N^2 - x_i y_i^2 \end{bmatrix} \quad (4.24)$$

$$\bar{\mathbf{a}} = [a_2 \quad a_3 \quad a_4 \quad \cdots \quad a_8]^T \quad (4.25)$$

and

$$\bar{\mathbf{u}}_i = [u_2 - u_i \quad u_3 - u_i \quad u_4 - u_i \quad \cdots \quad u_N - u_i] \quad (4.26)$$

Least-square solution of Equation (4.23) is given by

$$\bar{\mathbf{a}} = \bar{\mathbf{Q}} \bar{\mathbf{u}}_i \quad (4.27)$$

where $\bar{\mathbf{Q}} \equiv (\bar{\mathbf{P}}^T \bar{\mathbf{P}})^{-1} \bar{\mathbf{P}}^T$.

Equation (4.22) can be written as

$$a_1 = u_i - \bar{\mathbf{p}}(x_i, y_i) \bar{\mathbf{a}} \quad (4.28)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

where

$$\bar{\mathbf{p}}(x_i, y_i) = [x_i \quad y_i \quad x_i^2 \quad x_i y_i \quad y_i^2 \quad x_i^2 y_i \quad x_i y_i^2] \quad (4.29)$$

Similarly, Equation (4.16) can be written as

$$u_i(x, y) = a_i + \bar{\mathbf{p}}(x, y) \bar{\mathbf{a}} \quad (4.30)$$

$$\bar{\mathbf{p}}(x, y) = [x \quad y \quad x^2 \quad xy \quad y^2 \quad x^2 y \quad xy^2] \quad (4.31)$$

Using Equations (4.27) and (4.28) in Equation (4.30),

$$u_i(x, y) = u_i + \bar{\mathbf{q}} \bar{\mathbf{Q}} \bar{\mathbf{u}}_i \quad (4.32)$$

where

$$\bar{\mathbf{q}}(x, y) \equiv \bar{\mathbf{p}}(x, y) - \bar{\mathbf{p}}(x_i, y_i) \quad (4.33)$$

From Equation (4.33), observe that $\bar{\mathbf{q}}(x_i, y_i) = 0$, which means the vector $\bar{\mathbf{q}}$ reduces to a null vector when evaluated at node i . Using this result in Equation (4.32), we can show that $u_i(x_i, y_i) = u_i$ the *Kronecker delta* property of the interpolation is restored, which means that the least-square interpolant, $u_i(x, y)$, evaluated at node i simply reduces to the displacement of node i .

Equation (4.32) may be written concisely as

$$u_i(x, y) = \Phi_i \mathbf{u}_i \quad (4.34)$$

where the LSPIM nodal shape function array, Φ_i , is defined by

$$\Phi_i = [1 - \bar{\mathbf{q}} \bar{\mathbf{Q}} \mathbf{1} \quad | \quad \bar{\mathbf{q}} \bar{\mathbf{Q}}] \quad (4.35)$$

$1 \times N$

In Equation (4.35), $\mathbf{1}$ is defined to be a vector of dimension $(N-1)$ with all entries being equal to unity. Note that the first entry of nodal displacement vector $\mathbf{u}_i$ (Equation (4.21)) is the actual nodal displacement whereas the character of the rest of the entries is dual, i.e., the rest of the entries may be actual nodal displacements or nodal

displacement parameters. The dual character is due to the least-square nature of the solution of system (4.19). When the displacement field of the problem being solved involves only the assumed basis functions stated in Equation (4.17), the rest of the entries turn out to be the actual nodal displacements. (The proof is given in section 4.2.4.2.) If the displacement field of the problem being solved involves basis functions not appearing in Equation (4.17), the rest of the entries are just nodal displacement parameters.

4.2.3. CONSTRUCTION OF SHAPE FUNCTION MATRIX, Φ_i , Φ AND Ψ

The nodal shape function arrays, Φ_i , ($i = 1, 2, 3, 4$) given by Equation (4.35) are assembled as rows to construct the element shape function matrix, Φ , as illustrated in the following. The construction of shape functions is illustrated for element 1 in Fig. 4.5.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.5. The support of element 1.

4.2.3.1. Construction of Φ_i

From Equation (4.1),

$$u(x, y) = \mathbf{N}\mathbf{u}^e = N_1u_1(x, y) + N_2u_2(x, y) + N_3u_3(x, y) + N_4u_4(x, y) \quad (4.36)$$

For node 1, a support as shown by the dotted line in Fig. 4.5 is considered, which includes nodes $\{1, 2, 3, 4, 5, 6, 7, 8, 9\}$. Hence, from Equation (4.8), the LSPIM approximation is written in the form

$$u_1(x, y) = \Phi_1^1u_1 + \Phi_2^1u_2 + \Phi_3^1u_3 + \Phi_4^1u_4 + \Phi_5^1u_5 + \Phi_6^1u_6 + \Phi_7^1u_7 + \Phi_8^1u_8 + \Phi_9^1u_9 \quad (4.37)$$

For node 2, a support can similarly be defined to include nodes $\{1, 2, 3, 4, 8, 9, 10, 11, 12\}$.

Equation (4.8) gives the LSPIM approximation as

$$u_2(x, y) = \Phi_1^2u_1 + \Phi_2^2u_2 + \Phi_3^2u_3 + \Phi_4^2u_4 + \Phi_8^2u_8 + \Phi_9^2u_9 + \Phi_{10}^2u_{10} + \Phi_{11}^2u_{11} + \Phi_{12}^2u_{12} \quad (4.38)$$

Similarly, for nodes 3 and 4, the following equations can be written

$$u_3(x, y) = \Phi_1^3u_1 + \Phi_2^3u_2 + \Phi_3^3u_3 + \Phi_4^3u_4 + \Phi_{11}^3u_{11} + \Phi_{12}^3u_{12} + \Phi_{13}^3u_{13} + \Phi_{14}^3u_{14} + \Phi_{15}^3u_{15} \quad (4.39)$$

$$u_4(x, y) = \Phi_1^4u_1 + \Phi_2^4u_2 + \Phi_3^4u_3 + \Phi_4^4u_4 + \Phi_5^4u_5 + \Phi_6^4u_6 + \Phi_{14}^4u_{14} + \Phi_{15}^4u_{15} + \Phi_{16}^4u_{16} \quad (4.40)$$

Hence, Φ_i -shape function matrices (Equation (4.9)) can be written as

$$\Phi_1 = [\Phi_1^1, \Phi_2^1, \Phi_3^1, \Phi_4^1, \Phi_5^1, \Phi_6^1, \Phi_7^1, \Phi_8^1, \Phi_9^1] \quad (4.41)$$

$$\Phi_2 = [\Phi_1^2, \Phi_2^2, \Phi_3^2, \Phi_4^2, \Phi_8^2, \Phi_9^2, \Phi_{10}^2, \Phi_{11}^2, \Phi_{12}^2] \quad (4.42)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$\Phi_3 = [\Phi_1^3, \Phi_2^3, \Phi_3^3, \Phi_4^3, \Phi_{11}^3, \Phi_{12}^3, \Phi_{13}^3, \Phi_{14}^3, \Phi_{15}^3] \quad (4.43)$$

$$\Phi_4 = [\Phi_1^4, \Phi_2^4, \Phi_3^4, \Phi_4^4, \Phi_5^4, \Phi_6^4, \Phi_{14}^4, \Phi_{15}^4, \Phi_{16}^4] \quad (4.44)$$

4.2.3.2. Assembly of shape functions Φ using Φ_i

The entries of shape function matrices Φ_1, Φ_2, Φ_3 and Φ_4 defined in Equations (4.41)-(4.44) are assembled appropriately to obtain Φ -matrix as shown below:

$$\Phi = \begin{bmatrix} \Phi_1^1 & \Phi_2^1 & \Phi_3^1 & \Phi_4^1 & \Phi_5^1 & \Phi_6^1 & \Phi_7^1 & \Phi_8^1 & \Phi_9^1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ \Phi_1^2 & \Phi_2^2 & \Phi_3^2 & \Phi_4^2 & 0 & 0 & 0 & \Phi_8^2 & \Phi_9^2 & \Phi_{10}^2 & \Phi_{11}^2 & \Phi_{12}^2 & 0 & 0 & 0 & 0 \\ \Phi_1^3 & \Phi_2^3 & \Phi_3^3 & \Phi_4^3 & 0 & 0 & 0 & 0 & 0 & 0 & \Phi_{11}^3 & \Phi_{12}^3 & \Phi_{13}^3 & \Phi_{14}^3 & \Phi_{15}^3 & 0 \\ \Phi_1^4 & \Phi_2^4 & \Phi_3^4 & \Phi_4^4 & \Phi_5^4 & \Phi_6^4 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \Phi_{14}^4 & \Phi_{15}^4 & \Phi_{16}^4 \end{bmatrix} \quad (4.45)$$

4.2.3.3. Construction of shape functions Ψ

Ψ -matrix is then computed as a product of N and Φ matrices defined in Equations (4.2) and (4.45), respectively, as shown in Equation (4.15).

4.2.4. USEFUL PROPERTIES OF SHAPE FUNCTIONS

4.2.4.1. Kronecker-delta property of Φ_i

Recall from Equation (4.33) that $\bar{\mathbf{q}}(x_i, y_i) = 0$. Using this result in Equation (4.35), the

Kronecker-delta property of LSPIM shape functions is established:

$$\Phi_i = [1 \ 0 \ 0 \ 0 \ \dots \ 0]_{1 \times N} \quad \text{at } (x, y) = (x_i, y_i) \quad (4.46)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Equation (4.46) means that at node i , the shape function for node i takes a value of unity and all other shape functions take zero values.

4.2.4.2. Kronecker-delta property of Ψ

Using the definition, $\Psi = \mathbf{N}\Phi$, the Ψ can be written:

$$\Psi = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{bmatrix} \Phi_1^1 & \Phi_2^1 & \Phi_3^1 & \Phi_4^1 & \Phi_5^1 & \dots & \Phi_M^1 \\ \Phi_1^2 & \Phi_2^2 & \Phi_3^2 & \Phi_4^2 & \Phi_5^2 & \dots & \Phi_M^2 \\ \Phi_1^3 & \Phi_2^3 & \Phi_3^3 & \Phi_4^3 & \Phi_5^3 & \dots & \Phi_M^3 \\ \Phi_1^4 & \Phi_2^4 & \Phi_3^4 & \Phi_4^4 & \Phi_5^4 & \dots & \Phi_M^4 \end{bmatrix} \quad (4.47)$$

Let us now evaluate Equation (4.47) at $(x, y) = (x_1, y_1)$. Because of the *Kronecker delta* property of Φ_i stated in Equation (4.35) and the well known *Kronecker delta* property of the FEM shape functions N_1, N_2, N_3 and N_4 , zeros and ones can be obtained as shown below:

$$\Psi = \begin{bmatrix} 1 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ \Phi_1^2 & \Phi_2^2 & \Phi_3^2 & \Phi_4^2 \\ \Phi_1^3 & \Phi_2^3 & \Phi_3^3 & \Phi_4^3 \\ \Phi_1^4 & \Phi_2^4 & \Phi_3^4 & \Phi_4^4 \end{bmatrix} \begin{bmatrix} 0 & \dots & 0 \\ \Phi_5^2 & \dots & \Phi_M^2 \\ \Phi_5^3 & \dots & \Phi_M^3 \\ \Phi_5^4 & \dots & \Phi_M^4 \end{bmatrix} \text{ at } (x, y) = (x_1, y_1) \quad (4.48)$$

Equation (4.48) simplifies to

$$\Psi = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & \dots & 0 \end{bmatrix} \text{ at } (x, y) = (x_1, y_1) \quad (4.49)$$

Equation (4.49) means that at node 1, the shape function for node 1 takes a value of unity and all other shape functions take zero values. Similarly

$$\Psi = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & \dots & 0 \end{bmatrix} \text{ at } (x, y) = (x_2, y_2) \quad (4.50)$$

$$\Psi = \begin{bmatrix} 0 & 0 & 1 & 0 & 0 & \dots & 0 \end{bmatrix} \text{ at } (x, y) = (x_3, y_3) \quad (4.51)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$\Psi = [0 \quad 0 \quad 0 \quad 1 \mid 0 \quad \cdots \quad 0] \quad \text{at } (x, y) = (x_4, y_4) \quad (4.52)$$

Thus, the shape functions in Ψ possess the desirable *Kronecker-delta* property and hence the displacement boundary conditions can be implemented in the present method as easily as in FEM.

4.2.4.3. Inter-element compatibility property of Ψ

Equation (4.1) is expanded as

$$u(x, y) = N_1 u_1(x, y) + N_2 u_2(x, y) + N_3 u_3(x, y) + N_4 u_4(x, y) \quad (4.53)$$

It is known that the inter-element compatibility property for isoparametric elements will be satisfied by a given element formulation if the displacement interpolation at any point on a given edge of the element can be written purely in terms of the nodal displacements of the connecting nodes and the respective shape functions. It is trivial to verify that this is satisfied by the FE-LSPIM QUAD4. For example, let us consider the edge connecting nodes 1-2. On this edge, the shape functions N_3 and N_4 take zero values and hence Equation (4.53) becomes

$$u(x, y) = N_1 u_1(x, y) + N_2 u_2(x, y) \quad (4.54)$$

Note that this is always true although the nodal displacement functions $u_3(x, y)$ and $u_4(x, y)$ may not be zero along edge 1-2. Thus, the displacement interpolation at any point on edge 1-2 is expressible purely in terms of the displacement functions of the connecting nodes and their shape functions. Hence, the inter-element compatibility property across edge 1-2 is assured. Similarly, the inter-element compatibility across other edges can be verified.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

The present element inherits its compatibility property from the compatibility property of the shape functions of the classical QUAD4 element, viz., N_1, N_2, N_3 and N_4 . Furthermore, the order of continuity across the element boundaries remains that of finite element. Thus, despite the use of composite shape functions in the interpolation, the order of continuity is no better than that of FEM

It is important to realise from Equation (4.54) that the displacement at any point along edge 1-2 depends not only on the nodal displacements of node 1 and 2, but also on the nodal displacements of all the support nodes. This is because $u_1(x, y)$ and $u_2(x, y)$ are nodal displacement functions which involve the nodal displacements of all the nodes in the supports of nodes 1 and 2. As a result, enforcement of prescribed nodal displacement boundary conditions at nodes 1 and 2 does not automatically ensure fulfillment of exact essential boundary condition along the entire length of the edge. Hence, in order to enforce the essential boundary condition along the entire length, penalty method or Lagrange multiplier method may have to be used for the purpose. This aspect of the present method is in contrast to the classical finite element method and rather similar to the meshfree methods.

4.2.4.4. Completeness (reproducibility) property of Ψ

The FE-LSPIM QUAD4 is capable of reproducing exactly any of the monomial terms, $\{1, x, y, x^2, xy, y^2, x^2y, xy^2\}$, used in the definition of $\mathbf{p}(x, y)$ (Equation (4.17)). In other words, the element can reproduce exactly any arbitrary displacement field of the form $\hat{u}(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 + b_7x^2y + b_8xy^2$. The proof is given below through the following two propositions:

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Proposition 1. If all the nodes in the support of node i are imposed with displacements conforming to the field, $\hat{u}(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 + b_7x^2y + b_8xy^2$, then the field captured by the nodal displacement function of node i ($i = 1, 2, 3, 4$) is exactly same as the imposed field, i.e., $u_i(x, y) \equiv \hat{u}(x, y)$.

Proof. The imposed field is expressed as

$$\hat{u}(x, y) = \mathbf{p}(x, y) \mathbf{b} \quad (4.55)$$

where

$$\mathbf{b} = [b_1 \quad b_2 \quad b_3 \quad b_4 \quad b_5 \quad b_6 \quad b_7 \quad b_8]^T \quad (4.56)$$

The imposed nodal displacements, $\hat{\mathbf{u}}$, can be expressed as

$$\hat{\mathbf{u}} = \mathbf{P} \mathbf{b} \quad (4.57)$$

where $\mathbf{P}$ is given by Equation (4.20) and

$$\hat{\mathbf{u}} = [\hat{u}(x_1, y_1) \quad \hat{u}(x_2, y_2) \quad \hat{u}(x_3, y_3) \quad \dots \quad \hat{u}(x_N, y_N)]^T \quad (4.58)$$

Now, impose these nodal displacements on the nodes in the support of node i .

Substituting $\mathbf{u}_i$ appearing on the right side of Equation (4.34) by $\hat{\mathbf{u}}$,

$$u_i(x, y) = \Phi_i \mathbf{P} \mathbf{b} \quad (4.59)$$

Using Equation (4.35) in Equation (4.59),

$$u_i(x, y) = [1 - \bar{\mathbf{q}} \bar{\mathbf{Q}} \mathbf{1} \quad | \quad \bar{\mathbf{q}} \bar{\mathbf{Q}}] \mathbf{P} \mathbf{b} \quad (4.60)$$

Partitioning the matrix $\mathbf{P}$ as

$$\mathbf{P} = \begin{bmatrix} 1 & x_i & y_i & x_i^2 & x_i y_i & x_i^2 & x_i^2 y_i & y_i^2 x_i \\ 1 & x_2 & y_2 & x_2^2 & x_2 y_2 & x_2^2 & x_2^2 y_2 & y_2^2 x_2 \\ 1 & x_3 & y_3 & x_3^2 & x_3 y_3 & x_3^2 & x_3^2 y_3 & y_3^2 x_3 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & x_I & y_I & x_I^2 & x_I y_I & x_I^2 & x_I^2 y_I & y_I^2 x_I \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & x_N & y_N & x_N^2 & x_N y_N & x_N^2 & x_N^2 y_N & y_N^2 x_N \end{bmatrix} \equiv \left[\begin{array}{c|c} \mathbf{p}(x_i, y_i) \\ \mathbf{1} & \mathbf{R} \end{array} \right] \quad (4.61)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Equation (4.61) is written as

$$u_i(x, y) = (1 - \bar{\mathbf{q}}\bar{\mathbf{Q}}.\mathbf{1})(\mathbf{p}(x_i, y_i)\mathbf{b}) + \bar{\mathbf{q}}\bar{\mathbf{Q}}[\mathbf{1} \mid \mathbf{R}]\mathbf{b} \quad (4.62)$$

Partitioning vector $\mathbf{b}$ as

$$\mathbf{b} = [b_1 \mid b_2 \ b_3 \ b_4 \ b_5 \ b_6 \ b_7 \ b_8]^T \equiv [b_1 \mid \bar{\mathbf{b}}]^T, \quad (4.63)$$

Equation (4.62) is written as

$$u_i(x, y) = \mathbf{p}(x_i, y_i)\mathbf{b} (1 - \bar{\mathbf{q}}\bar{\mathbf{Q}}.\mathbf{1}) + \bar{\mathbf{q}}\bar{\mathbf{Q}}[\mathbf{1}.b_1 + \mathbf{R}\bar{\mathbf{b}}] \quad (4.64)$$

From the first equation of the system (4.57),

$$b_1 = \hat{u}(x_i, y_i) - (b_2x_i + b_3y_i + b_4x_i^2 + b_5x_iy_i + b_6y_i^2 + b_7x_i^2y_i + b_8x_iy_i^2) \quad (4.65)$$

Substituting this expression for b_1 in Equation (4.64),

$$u_i(x, y) = \mathbf{p}(x_i, y_i)\mathbf{b} (1 - \bar{\mathbf{q}}\bar{\mathbf{Q}}.\mathbf{1}) + \bar{\mathbf{q}}\bar{\mathbf{Q}}[\mathbf{1}.\hat{u}(x_i, y_i) + \bar{\mathbf{P}}\bar{\mathbf{b}}] \quad (4.66)$$

$$= \mathbf{p}(x_i, y_i)\mathbf{b} - \mathbf{p}(x_i, y_i)\mathbf{b} \bar{\mathbf{q}}\bar{\mathbf{Q}}.\mathbf{1} + \bar{\mathbf{q}}\bar{\mathbf{Q}}\mathbf{1}.\hat{u}(x_i, y_i) + \bar{\mathbf{q}}\bar{\mathbf{Q}}\bar{\mathbf{P}}\bar{\mathbf{b}} \quad (4.67)$$

$$= \mathbf{p}(x_i, y_i)\mathbf{b} + (\hat{u}(x_i, y_i) - \mathbf{p}(x_i, y_i)\mathbf{b} \bar{\mathbf{q}}\bar{\mathbf{Q}}.\mathbf{1}) + \bar{\mathbf{q}}\bar{\mathbf{Q}}\bar{\mathbf{P}}\bar{\mathbf{b}} \quad (4.68)$$

By the first equation of the system (4.57), $\hat{u}(x_i, y_i) - \mathbf{p}(x_i, y_i)\mathbf{b} = 0$, and hence Equation (4.68) becomes

$$u_i(x, y) = \mathbf{p}(x_i, y_i)\mathbf{b} + \bar{\mathbf{q}}\bar{\mathbf{Q}}\bar{\mathbf{P}}\bar{\mathbf{b}} \quad (4.69)$$

By Equation (4.27), $\bar{\mathbf{Q}} \equiv (\bar{\mathbf{P}}^T \bar{\mathbf{P}})^{-1} \bar{\mathbf{P}}^T$, which gives $\bar{\mathbf{Q}}\bar{\mathbf{P}} = \mathbf{I}$, the identity matrix. Hence,

Equation (4.69) becomes

$$u_i(x, y) = \mathbf{p}(x_i, y_i)\mathbf{b} + \bar{\mathbf{q}}\bar{\mathbf{b}} \quad (4.70)$$

Using Equations (4.33) and (4.63) in Equation (4.70),

$$u_i(x, y) = \mathbf{p}(x, y)\mathbf{b} \equiv \hat{u}(x, y) \quad (4.71)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

and hence Proposition 1.

Proposition2. If the nodes in the element support are imposed with displacements conforming to the field, $\hat{u}(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 + b_7x^2y + b_8xy^2$, then the displacement at any point of the element conforms to the imposed field.

Proof. Recall that the support of an element is defined as the union of the supports of the four nodes of the element, *i.e.*, $\Omega = \Omega_1 \oplus \Omega_2 \oplus \Omega_3 \oplus \Omega_4$. This would mean that, if the element support is imposed with displacements conforming to the field, $\hat{u}(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 + b_7x^2y + b_8xy^2$, then every nodal support is also imposed with the same field. Then, by Proposition 1, every nodal displacement function captures the imposed field. Hence,

$$u_i(x, y) \equiv \hat{u}_i(x, y) \quad i = 1, 2, 3, 4 \quad (4.72)$$

Using this result in Equation (4.1),

$$\begin{aligned} u(x, y) &= N_1u_1(x, y) + N_2u_2(x, y) + N_3u_3(x, y) + N_4u_4(x, y) \\ &= (N_1 + N_2 + N_3 + N_4)\hat{u}(x, y) \\ &= \hat{u}(x, y) \quad (\because N_1 + N_2 + N_3 + N_4 = 1) \end{aligned} \quad (4.73)$$

and hence Proposition 2.

4.2.5. FE-LSPIM QUAD4 SHAPE FUNCTION DERIVATIVES

The Cartesian derivatives of Ψ are obtained as

$$\Psi_{,x} = \mathbf{N}_{,x} \Phi + \mathbf{N} \Phi_{,x} \quad (4.74)$$

$$\Psi_{,y} = \mathbf{N}_{,y} \Phi + \mathbf{N} \Phi_{,y} \quad (4.75)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

The derivatives, $\Phi_{,x}$ and $\Phi_{,y}$, which are of size $4 \times M$ are obtained by assembling the row matrices:

$$\Phi_{i,x} = \begin{bmatrix} -\bar{q}_{,x} \bar{Q}.1 & | & \bar{q}_{,x} \bar{Q} \end{bmatrix} \quad (4.76)$$

$1 \times N$

and

$$\Phi_{i,y} = \begin{bmatrix} -\bar{q}_{,y} \bar{Q}.1 & | & \bar{q}_{,y} \bar{Q} \end{bmatrix} \quad (4.77)$$

$1 \times N$

which have been obtained by differentiating Equation (4.35) with respect to x and y , respectively. The assembly procedure is the same as that used for assembling Φ_i to get Φ (see Equation (4.11)).

Φ (see Equation (4.11)).

The derivative matrices, $N_{,x}$ and $N_{,y}$, are obtained from Equation (4.2) as in FEM:

$$N_{,x} = N_{,\xi} \xi_{,x} + N_{,\eta} \eta_{,x} \quad (4.78)$$

and

$$N_{,y} = N_{,\xi} \xi_{,y} + N_{,\eta} \eta_{,y} \quad (4.79)$$

respectively, where the coordinate transformation between the natural (ξ, η) system to the physical (x, y) is achieved as in FEM by using the shape functions, N_1, N_2, N_3 and N_4 , defined in Equations (4.3)-(4.6).

4.2.6. FE-LSPIM QUAD4 STRAIN AND STRESS EXPRESSIONS

Considering linear elastic material behavior, the strain and stress fields are expressed as

$$\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{q} \quad (4.80)$$

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon} \equiv \mathbf{D}\mathbf{B}\mathbf{q} \quad (4.81)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$\mathbf{q} = [u_1, v_1 \mid u_2, v_2 \mid u_3, v_3 \mid \cdots \mid u_m, v_m]^T \quad (4.82)$$

where $\mathbf{D}$ is the constitutive matrix and $\mathbf{q}$ is the vector of x- and y-nodal unknowns of all nodes in the support of element. Hence,

$$\mathbf{B} = \left[\begin{array}{cc|cc|cc| \cdots | cc} \Psi_{1,x} & 0 & \Psi_{2,x} & 0 & \Psi_{3,x} & 0 & \cdots & \Psi_{M,x} & 0 \\ 0 & \Psi_{1,y} & 0 & \Psi_{2,y} & 0 & \Psi_{3,y} & \cdots & 0 & \Psi_{M,y} \\ \hline \Psi_{1,y} & \Psi_{1,x} & \Psi_{2,y} & \Psi_{2,x} & \Psi_{3,y} & \Psi_{3,x} & \cdots & \Psi_{M,y} & \Psi_{M,x} \end{array} \right] \quad (4.83)$$

4.2.7. FE-LSPIM QUAD4 STIFFNESS MATRIX AND LOAD VECTORS

Choosing both the test and trial functions from FE-LSPIM QUAD4 shape functions, the element stiffness matrix and load vectors are obtained as in the FEM:

$$\mathbf{K}^{(e)}_{2M \times 2M} = \int_{\Omega^{(e)}} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega^{(e)} \quad (4.84)$$

$$\mathbf{f}^{(e)} = \mathbf{f}_b^{(e)} + \mathbf{f}_t^{(e)} \quad (4.85)$$

$$\mathbf{f}_b^{(e)} = \int_{\Omega^{(e)}} \mathbf{M}^T \mathbf{b}^{(e)} d\Omega^{(e)} \quad (4.86)$$

$$\mathbf{f}_t^{(e)} = \int_{\Gamma^{(e)}} \mathbf{M}^T \mathbf{t}^{(e)} d\Omega^{(e)} \quad (4.87)$$

$$\mathbf{M} = \left[\begin{array}{cc|cc|cc| \cdots | cc} \Psi_1 & 0 & \Psi_2 & 0 & \Psi_3 & 0 & \cdots & \Psi_M & 0 \\ 0 & \Psi_1 & 0 & \Psi_2 & 0 & \Psi_3 & \cdots & 0 & \Psi_M \end{array} \right] \quad (4.88)$$

where the element loads $\mathbf{f}_b^{(e)}$ and $\mathbf{f}_t^{(e)}$ are due to body forces, $\mathbf{b}$, and surface traction forces, $\mathbf{t}$, acting on the element, respectively. Note that the element stiffness matrix $\mathbf{K}^{(e)}$ is of size $2M \times 2M$ and the element load vectors $\mathbf{f}_b^{(e)}$ and $\mathbf{f}_t^{(e)}$ are of size $2M \times 1$. The displacement boundary conditions and concentrated forces, if any, are applied at the global level as in FEM. The numerical evaluation of the stiffness and load integrals is similar to FEM.

4.2.8. PSEUDO CODE FOR IMPLEMENTING THE FE-LSPIM QUAD4 ELEMENT

The flowchart of the present method is as follows:

Loop over the nodes in the mesh:

For each nodes, determine the number of nodes in its support, N , and save this information.

End loop over nodes

Loop over the elements in the mesh:

Loop over quadrature points in each element:

Determine the number of nodes, M , in the support of the element.

Loop over the four nodes of the element:

a) Compute Φ_i and assemble it as described in section 4.2.3 to form Φ .

b) Compute $\mathbf{N}$ using Equations (4.3)-(4.6) and then compute Ψ using Equation (4.15).

c) Compute $\mathbf{N}_{,x}, \mathbf{N}_{,y}, \Psi_{,x}$ and $\Psi_{,y}$ as outlined in section 4.2.5.

End loop over nodes of the elements.

Construct $\mathbf{B}$ matrix as outlined in section 4.2.6.

Compute the product $\mathbf{B}^T \mathbf{D} \mathbf{B}$.

Compute $\mathbf{M}^T \mathbf{b}^{(e)}$ and $\mathbf{M}^T \mathbf{t}^{(e)}$.

Add quadrature point contribution to element stiffness matrix $\mathbf{K}^{(e)}$ as in FEM.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Add quadrature point contribution to element force vectors $f_b^{(e)}$ and $f_t^{(e)}$ as
in FEM.

End quadrature point loop.

Assemble element stiffness matrix into the global stiffness matrix.

Assemble element force vector into the global force vector.

End element loop.

Add point loads to the global force vectors, if any.

Solve the stiffness equations for displacements.

4.3. THE FORMULATION OF FE-LSPIM QUAD4U ELEMENT

The FE-LSPIM QUAD4U element to be developed in this section is an unsymmetric version of FE-LSPIM QUAD4 element already discussed in section 4.2.

In the last chapter (Chapter 3), two FE-meshfree method, viz., FE-EFG and FE-PIM, were proposed. They were based on ‘unsymmetric formulation’ wherein the element stiffness is of the form $\mathbf{K} = \int_{\Omega^e} \mathbf{B}_L^T \mathbf{D} \mathbf{B}_R d\Omega^e$, where $\mathbf{B}_L$ and $\mathbf{B}_R$ are two different strain-displacement matrices and $\mathbf{D}$ is the constitutive matrix. FEM shape functions were used for constructing $\mathbf{B}_L$, whereas meshfree shape functions, EFG and PIM shape functions were used for constructing $\mathbf{B}_R$. The two unsymmetric elements showed good tolerance to mesh distortions. This forms the motivation to try FE-LSPIM QUAD4U element.

4.3.1. TEST FUNCTIONS AND DERIVATIVE MATRIX OF FE-LSPIM

QUAD4U

For a four-node quadrilateral element, the parametric shape functions are given in Equations (4.3) – (4.6). It is important to note that the parametric shape functions are not appropriate as trial functions because some of the higher order completeness requirements are not satisfied under distorted mesh geometry. For example, for an element with a bilinear or quadratic geometry, none of the quadratic terms (involving x^2 or xy or y^2) of the displacement field can be reproduced by the parametric interpolation. Similarly, the meshfree shape functions are not suitable as test functions because the inter element continuity is violated under distorted mesh geometry. The test function and the corresponding strain and stress fields are written in terms of parametric coordinates as

$$\mathbf{v}^{(e)} \equiv \begin{Bmatrix} v_u^{(e)}(\xi, \eta) \\ v_v^{(e)}(\xi, \eta) \end{Bmatrix} = \begin{bmatrix} N_1, 0, N_2, 0, N_3, 0, N_4, 0 \\ 0, N_1, 0, N_2, 0, N_3, 0, N_4 \end{bmatrix} \mathbf{v}_n^{(e)} \equiv \mathbf{N} \mathbf{v}_n^{(e)} \quad (4.89)$$

$$\mathbf{L} \mathbf{v}^{(e)} = \mathbf{L} \mathbf{N} \mathbf{v}_n^{(e)} \equiv \mathbf{B}_L \mathbf{v}_n^{(e)} \quad (4.90)$$

$$\boldsymbol{\sigma}^{(e)} = \mathbf{D} \boldsymbol{\varepsilon}^{(e)} \equiv \mathbf{D} \mathbf{B}_L \mathbf{v}_n^{(e)} \quad (4.91)$$

where

$$\mathbf{B}_L = \mathbf{L} \mathbf{N} = \begin{bmatrix} N_{1,x} & 0 & N_{2,x} & 0 & N_{3,x} & 0 & N_{4,x} & 0 \\ 0 & N_{1,y} & 0 & N_{2,y} & 0 & N_{3,y} & 0 & N_{4,y} \\ N_{1,y} & N_{1,x} & N_{2,y} & N_{2,x} & N_{3,y} & N_{3,x} & N_{4,y} & N_{4,x} \end{bmatrix} \quad (4.92)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$\mathbf{N}$ is a matrix of parametric shape functions, $N_{i,x}$ and $N_{i,y}$ ($i=1,2,3,4$) are the Cartesian derivatives of parametric shape functions, and $\mathbf{v}_n^{(e)}$ is a vector of arbitrary nodal weight values.

4.3.2. FE-LSPIM QUAD4U ELEMENT STIFFNESS MATRIX AND LOAD VECTORS

The stiffness matrix and load vectors in the FE-LSPIM QUAD4U are derived according to the Petrov-Galerkin formulation. The stiffness matrix and force vectors can be expressed as follows:

$$\mathbf{K}^{(e)} = \int_{\Omega^{(e)}} \mathbf{B}_L^T \mathbf{D} \mathbf{B}_R d\Omega^{(e)} \quad (4.93)$$

$$\mathbf{f}^{(e)} = \mathbf{f}_b^{(e)} + \mathbf{f}_t^{(e)} \quad (4.94)$$

$$\mathbf{f}_b^{(e)} = \int_{\Omega^{(e)}} \mathbf{N}^T \mathbf{b}^{(e)} d\Omega^{(e)} \quad (4.95)$$

$$\mathbf{f}_t^{(e)} = \int_{\Gamma^{(e)}} \mathbf{N}^T \mathbf{t}^{(e)} d\Omega^{(e)} \quad (4.96)$$

Where $\mathbf{B}_L$ is from Equation (4.92), $\mathbf{B}_R$ is from Equation (4.83). The stiffness matrix, $\mathbf{K}$, turn out to be unsymmetric in view of different sets of shape functions used for test and trial functions, and hence an unsymmetric solver is needed for the solution of resulting finite element equations. For numerical integration, the isparametric mapping of geometry is, as usual, carried out using the parametric shape functions, and the numerical integration of stiffness and load integrals is by Gaussian quadrature. The nodal force vectors, $\mathbf{f}_b^{(e)}$ and $\mathbf{f}_t^{(e)}$, are due to body forces, $\mathbf{b}$, and surface traction forces, $\mathbf{t}$, acting on the element, respectively.

4.4. TEST PROBLEMS

A series of test problems are solved using the proposed element to study the efficacy of the proposed element, and the results are compared with exact solution, results of other researchers where available, and results obtained using other finite elements and meshfree methods. For numerical integration of stiffness, 2×2 Gaussian quadrature scheme is for all elements used except for QUAD16 for which 4×4 scheme is used.

4.4.1. PATCH TEST

The principal virtue of a patch test is that, if an element can produce the correct results for the test, the results for any problem solved with the element will convergence toward the correct solution as the mesh is continuously refined. The patch test was originally introduced by Iron [152] and Iron and Razzaque [153] to check whether a patch of elements subjected to a constant strain can reproduce exact strain when the patch becomes infinitesimally small. Three types of patch test, viz., type A, B and C are proposed by Zienkiewicz and Taylor [154].

Consider the following differential equation

$$\mathbf{K}\mathbf{u} - \mathbf{f} = 0 \quad (4.97)$$

where $\mathbf{u}$ is a vector of the nodal variables.

In Type A patch test as shown in Fig.4.6, the exact values of the displacements are inserted into the i th equation and the following condition is verified:

$$\mathbf{K}_{ij}u_j - \mathbf{f}_i \equiv 0 \quad (4.98)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

In Type B patch test (Fig. 4.7), only the values of displacement corresponding to boundaries of the patch are substituted into Equation (4.98). u_i is found as

$$u_i = \mathbf{K}_{ii}^{-1}(\mathbf{f}_i - \mathbf{K}_{ij}u_j) \quad j \neq i \quad (4.99)$$

and compared with the exact displacement.

In Type C patch test (Fig. 4.8), the patch of the elements is assembled as before, but subjected to prescribed displacement boundary conditions. The equations are solved for the remaining displacement values and compared with the exact solution.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.6. Type A patch test.

$\mathbf{u}$ prescribed on all nodes $K_{ij}u_j = f_i$ verified
at node i

Figure 4.7. Type B patch test.

$\mathbf{u}$ prescribed at edges of patch
 $u_i = \mathbf{K}_{ii}^{-1}(\mathbf{f}_i - \mathbf{K}_{ij}u_j) \quad (j \neq i)$ solved

Figure 4.8. Type C patch test.

4.4.1.1. Constant strain/stress patch test

The constant strain/stress patch test using irregular mesh is considered first. This patch test is proposed by Chen et al. [66] for a patch as shown in Fig. 4.9, for which, the Young's modulus $E = 1000$, Poisson's ratio $\nu = 0.25$ and unit thickness. The

rectangular domain is divided into three elements as shown and then, for subsequent mesh refinement, each element is divided through the midpoints of the element sides into four elements at every stage of mesh refinement. In addition to applying zero displacements on the nodes of displacement boundary shown in Fig. 4.9, the distributed reaction forces also applied on this boundary, which are equal and opposite to the applied forces on the opposite boundaries. The applied as well as reaction force was lumped on to the nodes.

Figure 4.9. Constant strain patch test by Chen et al [66].

Table 4.1 shows the results of the constant strain weak patch test. The results shown in the bracket are obtained by AGQ6-I [66] and are shown here for comparison purposes. The present element is able to reproduce the exact solution for every mesh and thus can pass this patch test even in the strong form.

Chen et al. [66] have only conducted constant strain/stress patch test. For the present work, it is intended to carry out linear and quadratic strain patch test using distorted as well as undistorted element geometries. The mesh used by Chen et al. [66] (shown in Fig. 4.9) has three distorted elements and it is difficult to construct undistorted mesh

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

with three elements, so studying the effect of mesh distortion is not possible with this mesh. Hence, a different mesh will be used for the linear and quadratic strain patch tests discussed in section 4.4.1.2 and section 4.4.1.3.

Table 4.1. Patch test results.

Mesh	v_2	v_3	u_4	v_4	u_5	v_5	u_6	v_6	u_7	v_7
3	-0.2500 (-0.2787)	-0.5000 (-0.4291)	2.0000 (2.0758)	-0.1250 (-0.4211)	2.5000 (2.4946)	0.0000 (-0.9491)	4.0000 (5.4819)	0.0000 (-0.8051)	4.0000 (4.9589)	-0.5000 (-1.6375)
12	-0.2500 (-0.2653)	-0.5000 (-0.4735)	2.0000 (2.0508)	-0.1250 (-0.1862)	2.5000 (2.4578)	0.0000 (-0.3988)	4.0000 (4.7955)	0.0000 (0.1313)	4.0000 (4.2688)	-0.5000 (0.8346)
48	-0.2500 (-0.2605)	-0.5000 (-0.4962)	2.0000 (2.0142)	-0.1250 (-0.1238)	2.5000 (2.4975)	0.0000 (-0.1044)	4.0000 (4.2432)	0.0000 (0.1049)	4.0000 (4.0753)	-0.5000 (-0.5683)
192	-0.2500 (-0.2544)	-0.5000 (-0.4996)	2.0000 (2.0042)	-0.1250 (-0.1231)	2.5000 (2.5007)	0.0000 (-0.0288)	4.0000 (4.0685)	0.0000 (-0.0383)	4.0000 (4.0233)	-0.5000 (-0.5203)
Exact	-0.2500	-0.5000	2.0000	-0.1250	2.5000	0.0000	4.0000	0.0000	4.0000	-0.5000

Note: values shown in bracket are obtained by AGQ6-I element and the other values by the present elements (FE-LSPIM QUAD4 and FE-LSPIM QUAD4U).

4.4.1.2. The linear strain patch tests in the strong form

A plane stress problem using the patch shown in Fig. 4.10 is considered for the linear strain patch test. The nodal coordinates are listed in Table 4.2. The length of each patch, $L = 2$, the height of each patch, $D = 2$, the thickness of each patch, $t = 1$. The material is assumed to be linear, isotropic elastic with properties $E = 1000$ and $\nu = 0.28$. Type B and Type C patch tests are performed in this section. For the linear strain patch test, the patch is assumed to be subjected to a constant moment $M = 2000$ on the right edge 3-6-9. The exact solutions for this problem are:

$$u = \frac{M}{EI}xy \quad (4.100)$$

$$v = -\frac{\nu M}{2EI}y^2 - \frac{M}{2EI}x^2 \quad (4.101)$$

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$\sigma_x = \frac{M}{I} y, \sigma_y = 0, \sigma_{xy} = 0 \quad (4.102)$$

$$I = \frac{D^3}{12} \quad (4.103)$$

Figure 4.10. Patch considered for patch test; (a) regular mesh; (b) distorted mesh.

Table 4.2. Coordinates for patch (a).

nodes		1	2	3	4	5	6	7	8	9
Patch (a)	y	0	1	2	0	1	2	0	1	2
	x	-1	-1	-1	0	0	0	1	1	1
Patch (b)	y	0	1	2	0	1.2	2	0	1	2
	x	-1	-1	-1	0	0.3	0	1	1	1

The results of the linear strain patch tests are listed in Tables 4.3-4.6. It is seen from Table 4.3 that all the three elements can pass Type B patch test for regular mesh. For distorted mesh, however, FE-LSPIM QUAD4U can pass the test while FE-LSPIM QUAD4 elements cannot (Table 4.4). It is can be seen from Table 4.4, the linear strain patch test result for the u displacement given by FE-LSPIM QUAD4 has a small error of $(1 - 0.996 = 0.004)$. This is rather unexpected because the shape functions used in

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

this element has been shown to satisfy compatibility and quadratic displacement completeness of the mesh (see section 4.2.4.4). This is because the FE-LSPIM QUAD4 does not fulfill node-to-edge Kronecker-delta property, so the essential boundary condition can not enforce directly.

Table 4.3. Normalized displacements at node 5 for linear strain type B Patch test (Regular mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	1	1	1	0
v	1	1	1	-1.5

Table 4.4. Normalised displacements at node 5 for linear strain type B Patch test (Distorted mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	0.852	0.996	1	0.78
v	0.968	1	1	-2.5518

Table 4.5 shows that all the three elements can pass type C linear strain patch test when regular meshes are used. However, for distorted meshes, both FE-LSPIM QUAD4 element and FE-LSPIM QUAD4U element can pass the test while QUAD4 element cannot (Table 4.6).

Table 4.5. Normalised displacements at node 5 for linear strain type C Patch test (Regular mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	1	1	1	0
v	0.894	1	1	-1.5

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.6. Normalised displacements at node 5 for linear strain type C Patch test (Distorted mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	0.746	1	1	0.78
v	0.86	1	1	-2.5518

Figures 4.11- 4.14 show u , v , σ_x contour plots of the two patches for the linear strain type B and type C patch tests for FE-LSPIM QUAD4, FE-LSPIM QUAD4U and QUAD4 as well as exact solution. The Figs.4.11-4.14 show that both the FE-LSPIM QUAD4, FE-LSPIM QUAD4U can give smooth and continuous stress contour plots as in exact solution for all the tests while QUAD4 cannot.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.11. The contour plot of linear strain type B patch test for regular mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.

Figure 4.12. The contour plot of linear strain type B patch test for distorted mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.13. The contour plot of linear strain type C patch test for regular mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.14. The contour plot of linear strain type C patch test for distorted mesh (a) u displacement; (b) v displacement; (c) σ_x stress.

4.4.1.3. Quadratic strain patch tests

The quadratic strain patch tests are given in this section. The same patches shown in Fig. 4.10 are considered here. The patch is assumed to be subjected to a shear force $P = 1000$ on the right edge 3-6-9. When Type B patch tests are performed, the exact displacements are applied on the boundary nodes. When Type C path tests are performed, the forces computed from the exact stresses are applied on the boundary. The exact solutions for this problem are:

$$u = -\frac{Py}{6EI}[(6L - 3x)x + (2 + \nu)(y^2 - \frac{D^2}{4})] \quad (4.104)$$

$$v = -\frac{P}{6EI}[3\nu y^2(L - x) + (4 + 5\nu)\frac{D^2 x}{4} + (3L - x)x^2] \quad (4.105)$$

$$\sigma_x(x, y) = -\frac{P(L - x)y}{I} \quad (4.106)$$

$$\sigma_y = 0 \quad (4.107)$$

$$\sigma_{xy}(x, y) = -\frac{Py}{2I}\left[\frac{D^2}{4} - y^2\right] \quad (4.108)$$

The results of the quadric strain type B patch tests are listed in Tables 4.7-4.8. It is seen that all of the elements can pass the proposed type B patch test under regular mesh (Table 4.7). However, not all of them can pass the patch test when distorted meshes were used (Table 4.8). The error given by the FE LSPIM QUAD4 and FE-LSPIM QUAD4U is small compared with the results given by QUAD4.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

For type C patch tests, FE-LSPIM QUAD4 can pass the test while the other elements cannot under regular mesh (Table 4.9). Also none of them can pass the patch tests when distorted meshes are used (Table 4.10). The reason that the two proposed elements cannot pass the quadratic strain patch tests under distorted mesh is obvious: The displacement field of the test problem includes x^3 term where as the present elements can only reproduce the terms $x, y, x^2, xy, y^2, x^2y, xy^2$. Although the proposed elements cannot pass quadratic strain patch test in the strong form, with mesh refinement, they can still pass this test in weak form. This is shown in section 4.4.7.

Table 4.7. Normalised displacements at node 5 for Quadratic strain type B Patch test
(Regular mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	1	1	1	0
v	1	1	1	2.6

Table 4.8. Normalised displacements at node 5 for Quadratic strain type B Patch test
(Distorted mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	0.993	0.821	0.835	0.4171
v	0.966	0.981	0.986	3.7466

Table 4.9. Normalised displacements at node 5 for Quadratic strain type C Patch test
(Regular mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	1	1	1	0
v	0.982	1	0.992	2.6

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.10. Normalised displacements at node 5 for Quadratic strain type C Patch test
(Distorted mesh).

Element	QUAD4	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact solution
u	0.880	0.831	0.835	0.4171
v	0.902	0.991	0.998	3.7466

Figures 4.15-4.18 show u , v , σ_x contour plots of the two patches for the quadratic strain type B and type C patch tests for FE-LSPIM QUAD4, FE-LSPIM QUAD4U and QUAD4 as well as exact solution. The Figs. show that both FE-LSPIM QUAD4, FE-LSPIM QUAD4U can give almost smooth and continuous stress contour plots for most of the tests while QUAD4 cannot.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.15. The contour plot of quadratic strain Type B patch test for regular mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.16. The contour plot of quadratic strain Type B patch test for distorted mesh, (a) u displacement; (b) v displacement; (c) σ_x stress.

Figure 4.17. The contour plot of quadratic strain Type C patch test for regular mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.18. The contour plot of quadratic strain Type C patch test for distorted mesh. (a) u displacement; (b) v displacement; (c) σ_x stress.

4.4.2. COOK'S SKEW BEAM PROBLEM

Figure 4.19. Cook's skew beam problem.

This problem (Fig. 4.19) was proposed by Cook et al. [155], and involves a skew cantilever with distributed shear load at the right end face. The problem is modelled using 2×2 , 4×4 , 8×8 meshes. The Poisson's ratio $\nu = 1/3$ and Young's modulus $E = 1$. The deflection at point A, the maximum principal stress at point B and the minimum principal stress at point C are computed and listed in Tables 4.11-4.13 along the results of other researchers. The results of 8-node serendipity element (QUAD8) and 16-node Lagrangian element (QUAD16) as computed by us are also listed for reference. The stresses at points A and B are obtained by averaging the two values given by the two adjacent elements. It is seen from Tables 4.11-4.13 that the present elements exhibit the good convergence.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.11. Computed v_C for Cook's skew beam.

Element	Mesh			
	2×2	4×4	8×8	16×16
Q4 [66]	11.845	18.299	22.079	23.430
Q6 [17]	22.94	23.48	23.80	23.91
QM6[18]	21.05	23.02	23.65	23.86
QUAD8	23.218	23.680	23.865	23.927
QUAD16	23.833	23.902	24.025	23.954
P2-Q6-I [26]	--	23.016	--	--
AGQ6-I [66]	23.07	23.68	23.87	23.93
FE-LSPIM QUAD4U (Present)	19.463	23.381	23.867	23.92
FE-LSPIM QUAD4 (Present)	21.572	23.570	23.856	23.918
Reference solution [156] = 23.9				

Table 4.12. $(\sigma_A)_{\max}$ of Cook's skew beam.

Element	Mesh			
	2×2	4×4	8×8	16×16
Q4 [66]	0.1078	0.1814	0.2227	0.2353
Q6 [17]	0.2029	0.2258	0.2334	0.2361
QM6[18]	0.1298	0.2243	0.2341	0.2364
QUAD8	0.2642	0.2425	0.2387	0.2374
QUAD16	0.2407	0.2374	0.2369	0.2369
P2-Q6-I [26]	--	0.2268	--	--
AGQ6-I [66]	0.2023	0.2275	0.2351	0.2365
FE-LSPIM QUAD4U (Present)	0.1860	0.2319	0.2370	0.2370
FE-LSPIM QUAD4 (Present)	0.1235	0.2325	0.2380	0.2372
Reference solution[156] = 0.236				

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.13. $(\sigma_B)_{\min}$ of Cook's skew beam.

Element	Mesh			
	2×2	4×4	8×8	16×16
Q4 [66]	-0.0776	-0.1429	-0.1840	-0.1995
Q6 [17]	-0.1734	-0.1915	-0.1997	-0.2028
QM6[18]	-0.1580	-0.1856	-0.1985	-0.2025
QUAD8	-0.2161	-0.2052	-0.2032	-0.2035
QUAD16	-0.1966	-0.2050	-0.2035	-0.2035
P2-Q6-I [26]	--	-0.1865	--	--
AGQ6-I [66]	-0.1758	-0.1972	-0.2016	-0.2033
FE-LSPIM QUAD4U (Present)	-0.2183	-0.2101	-0.1999	-0.2033
FE-LSPIM QUAD4 (Present)	-0.1796	-0.2079	-0.2008	-0.2033
Reference solution [156]= -0.201				

4.4.3. THICK CURVED BEAM

A thick curved cantilever beam is subjected to a transverse force at its tip as shown in Fig. 4.20. The geometry is meshed using 2×4 elements. The Young's modulus $E = 1000N/mm^2$ and Poisson's ratio $\nu = 0$. The thickness of the beam is unit. The computed vertical tip deflection at point A is given in Table 4.14. It is seen that the results given by FE-LSPIM QUAD4 and FE-LSPIM QUAD4U are better than those given by QUAD4 and QM6.

Table 4.14. Computed vertical deflection of a thick curved beam (Fig. 4.20).

Elements	QUAD4	QM6	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)	Exact
v_A	57.576	83.751	85.99	85.99	90.01

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.20. Bending of thick curved beam.

4.4.4. MACNEAL'S THIN CANTILEVER BEAM

MacNeal's thin cantilever beam [157] (Fig. 4.21) is a frequently used test problem due to its simplicity and the fact that the principal element deformation modes can be evoked by loads applied to the free end. This problem is modelled using three types of meshes with rectangular, parallelogram and trapezoidal elements (Fig. 4.21). Two types of loading, viz., unit tip shear force, P , and unit tip moment, M , are considered. The Young's modulus of the beam $E = 10^7$, Poisson's ratio $\nu = 0.3$ and thickness of the beam $t = 0.1$. The results computed using the present element are compared with the results obtained with other methods in Table 4.15. A six term basis (i.e., $1, x, y, xy, x^2, y^2$) is used to construct $\mathbf{P}$ and $\bar{\mathbf{P}}$ -matrices. In addition, a small perturbations to nodal coordinates as done by Liu and Gu [107] is necessary because the nodal density shown in Fig.4.21 cannot represent a second order polynomial in the y -

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

direction as there are only two distinct y -coordinate values in the support of a given node.

Figure 4.21. MacNeal's beam subjected to (1) unit shear force (2) constant moment.

Table 4.15. Normalized tip deflection for MacNeal's beam.

Element	Load P			Load M		
	Mesh(a)	Mesh(b)	Mesh(c)	Mesh(a)	Mesh(b)	Mesh(c)
Q4 [66]	0.093	0.035	0.003	0.093	0.031	0.022
Q6 [157]	0.993	0.667	0.106	1.000	0.759	0.093
QM6 [18]	0.993	0.623	0.044	1.000	0.759	0.093
QUAD4 [66]	0.904	0.080	0.071	-	-	-
QUAD8	0.987	0.983	0.872	1.000	0.997	0.947
QUAD16	0.997	0.997	0.997	1.000	1.000	1.000
P-S [28]	0.993	0.798	0.221	1.000	0.852	0.167
RGD20 [41]	0.981	0.625	0.047	-	-	-
AQ6-I [66]	0.993	0.994	0.994	1.000	1.000	1.000
AQ6-II [66]	0.993	0.994	0.994	1.000	1.000	1.000
FE-LSPIM QUAD4 (Present)	0.984	0.963	0.932	1.000	1.000	1.000
FE-LSPIM QUAD4U (Present)	0.816	0.783	0.495	1.000	1.000	1.000
Exact solution = -0.1081				Exact solution = 0.0054		

The results of QUAD8 and QUAD16 have been computed by the author's computer program whereas the rest have been taken from references.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

In order to check volumetric locking of the proposed element, the tip deflection has been computed under plain strain condition by varying the Poisson's ratio as $\nu = 0.3, 0.49, 0.499, 0.4999$ and the results are summarised in Table 4.16. It is seen that the present element tends to lock although the locking tendency is much less compared to QUAD4 element.

Table 4.16. Tip deflection for MacNeal's beam as $\nu \rightarrow 0.5$.

Element	Poisson's ratio	Load P			Load M		
		Mesh(a)	Mesh(b)	Mesh(c)	Mesh(a)	Mesh(b)	Mesh(c)
Present	0.3	-0.073869	-0.073690	-0.073537	-0.004919	-0.004908	-0.004898
	0.49	-0.061586	-0.059599	-0.061224	-0.004100	-0.003969	-0.004076
	0.499	-0.059221	-0.040129	-0.046108	-0.003942	-0.002673	-0.003070
	0.4999	-0.027486	-0.005372	-0.013397	-0.001830	-0.000356	-0.000890
	0.49999	-0.004104	-0.000994	-0.002428	-0.002726	-0.000067	-0.000162
QUAD4	0.3	-0.009856	-0.003371	-0.002754	-0.000492	-0.000150	-0.000113
	0.49	-0.004293	-0.001247	-0.001736	-0.000212	-0.000045	-0.000075
	0.499	-0.000701	-0.000429	-0.001026	-0.000031	-0.000014	-0.000050
	0.4999	-0.000154	-0.000166	-0.000765	-0.000003	-0.000004	-0.000040
	0.49999	-0.000096	-0.000115	-0.000715	-0.000000	-0.000001	-0.000037

4.4.5. A CANTILEVER BEAM SUBJECTED TO A PURE TIP MOMENT

The cantilever beam shown in Fig. 3.6 is considered again. The beam is now subjected to a pure bending moment, $M = -24000$, at the free end. The moment is applied as a linearly varied distributed load of intensity $\sigma_x = \frac{My}{I}$. In addition to applying zero displacements on the nodes of displacement boundaries shown in Fig. 3.6, the distributed reaction moment on the boundary is also applied, which is equal and opposite to the applied moment on the free end. The exact solution to this problem is given in Equations (3.59-3.62).

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.17 lists the computed vertical displacement at point A of the beam along with the exact vertical displacement given by Equation (3.60). The results show that both FE-LSPIM QUAD4 and FE-LSPIM QUAD4U (as well as QM6, QUAD8 and QUAD16) reproduces exact solution for all the five meshes used. QUAD4 and EFG give convergent results although the results of QUAD4 converge slowly compared to that of EFG. The results of QUAD8, QUAD16 and EFG have been computed by the author's computer program whereas the QM6 have been obtained using ANSYS.

Table 4.17. Vertical displacement* at the tip of the beam under pure tip moment.

Mesh	2×2	2×8	4×16	6×24	8×32
QUAD4	- 2.497	-5.709	-6.194	-6.301	-6.342
EFG	-6.406	-6.419	-6.407	-6.403	-6.402
QM6	-6.400	-6.400	-6.400	-6.400	-6.400
QUAD8	-6.400	-6.400	-6.400	-6.400	-6.400
QUAD16	-6.400	-6.400	-6.400	-6.400	-6.400
FE-LSPIM QUAD4 (Present)	-6.400	-6.400	-6.400	-6.400	-6.400
FE-LSPIM QUAD4U (Present)	-6.400	-6.400	-6.400	-6.400	-6.400
Exact=-6.400					

*Multiply all values by 10^{-3} .

Figure 4.22 shows the contour plots of σ_x stress for this problem. A 2×2 Gauss integration and a 16×4 mesh are used for the computation. As this problem is a constant bending moment problem, the normal stress must have a linear distribution across the depth of the beam (Equation (3.61)). If a given solution method were to faithfully reproduce the exact solution, the stress bands must be straight and horizontal, and also continuous across the element boundaries. As seen from the Fig. 4.22, all the methods give straight and continuous stress contours similar to the exact solution (Fig. 4.22a).

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.23 shows the contour plots of σ_x stress obtained with a distorted mesh. Here again, a 2×2 Gauss integration and 24×6 mesh are used. As seen from Fig. 4.23, FE-LSPIM QUAD4 as well as FE-LSPIM QUAD4U gives straight and continuous contour plots even for the distorted mesh too. However, QUAD4 gives a highly discontinuous contour plot. Although EFG gives a continuous contour plot, it exhibits some waviness.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.22. Contour plot of σ_x for pure tip moment problem under regular mesh.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.23. Contour plot of σ_x for pure tip moment problem under distorted mesh.

**4.4.6. A CANTILEVER BEAM SUBJECT TO PARABOLICALLY
DISTRIBUTED TIP SHEAR FORCE**

Considering the same cantilever beam geometry as shown in Fig. 3.9, the beam is now subjected to a tip shear force, $P = -1000$, at the free end. The shear force is applied as a linearly varied distributed load of intensity according to Equations (3.66-3.68). In addition to applying zero displacements on the nodes of displacement boundaries shown in Fig. 3.6, the distributed reaction shear force on the boundary is also applied, which is equal and opposite to the applied shear force on the free end.

Table 4.18 lists the computed vertical displacement at point A of the beam. A 2×2 gauss integration is used for the purpose. The results show a good agreement between the exact solution and the present elements (FE-LSPIM QUAD4 and FE-LSPIM QUAD4U). It can also be seen that the results obtained with the present element are generally more efficient than that of EFG, QM6 [18], QUAD4 and QUAD8, and comparable to that of QUAD16.

Table 4.18. Vertical displacement* at the tip of the beam under a tip shear load.

Mesh	2×2	2×8	4×16	6×24	8×32
QUAD4	-3.475	-7.964	-8.612	-8.759	-8.816
EFG	-7.7851	-8.875	-8.910	-8.905	-8.903
QM6	-8.129	-8.788	-8.863	-8.883	-8.892
QUAD8	-8.823	-8.871	-8.895	-8.900	-8.900
QUAD16	-8.855	-8.899	-8.900	-8.900	-8.900
FE-LSPIM QUAD4 (Present)	-7.750	-8.906	-8.898	-8.899	-8.900
FE-LSPIM QUAD4U (Present)	-7.599	-8.888	-8.899	-8.900	-8.900
Exact=-8.900					

*Multiply all values by 10^{-3} .

All the results reported in this table have been computed by the author's computer program.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.19 lists the computed vertical displacements at point A of the beam for different orders of Gaussian quadrature. It is seen that the accuracy of results changes very little with different orders of Gaussian quadrature, even for the undistorted mesh. It is interesting to note that even a 2×2 scheme gives fairly accurate results although the displacement interpolation is apparently of a higher order. The reason for this is not completely understood. Although the local approximations (nodal displacement functions) are of higher order over the nodal supports, their variation over the element domains is predominantly of lower order due to the relatively small size of elements as compared to the nodal supports. Thus, the effective order of integrand appears to be still low so that a 2×2 scheme can give fairly accurate results.

Table 4.19. Computed displacements* for different orders of Gaussian integration of stiffness matrix.

Gauss integration	2×2	3×3	4×4
Regular mesh	-8.9064729	-8.9052297	-8.9052270
Distorted mesh	-8.8814459	-8.8763094	-8.8762754
Exact displacement = -8.9 (mm)			

*Multiply all values by 10^{-3} .

Figure 4.24 shows the contour plot of σ_x stress obtained with a 16×4 regular mesh for the tip shear problem. As the normal stress distribution is a bilinear function, its magnitude should vary along the length and depth of the beam as shown in Fig. 4.24a. For an element that can faithfully reproduce the exact solution, the stress bands must be smooth and continuous across the element boundaries although they will not be straight lines because of the bilinear nature of their distribution. Figure 4.24 shows that the stress bands given by FE-LSPIM QUAD4 and FE-LSPIM QUAD4U are highly smooth

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

and continuous across the inter-element boundaries whereas that of QUAD4 are highly discontinuous. The stress bands of EFG are also smooth and continuous but slight deviations from exact solution (Fig. 4.24a) are observed near left top and bottom corners.

Figure 4.25 shows the contour plot of σ_x obtained with a 24×6 distorted mesh for the same problem. It is seen that, although the mesh is severely distorted, the stress contours given by FE-LSPIM QUAD4 and FE-LSPIM QUAD4U are still continuous and smooth whereas that of QUAD4 are highly discontinuous. The stress bands given by EFG exhibit much waviness although continuous at the inter-element boundaries. The performance of present element is insensitive to mesh distortion under the cubic displacement field of the present example.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.24. The contour plot of σ_x for tip shear problem under regular mesh.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.25. The contour plot of σ_x for tip shear problem under distorted mesh.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

The convergence of the present method has been studied for this problem. For this study, meshes with 2×8 , 4×16 , 6×24 and 8×32 elements. For EFG method, the meshes are used as the background integration meshes for integration purposes. The size of the influence domain of the EFG method is chosen as the double of the background element size. The error of error energy norm used for monitoring convergence is as follows.

where v is the vertical displacement at the tip of the beam (point A).

$$\text{Error energy norm} = \left\{ \frac{1}{2} \int_{\Omega} (\boldsymbol{\varepsilon}^{num} - \boldsymbol{\varepsilon}^{exact})^T \mathbf{D} (\boldsymbol{\varepsilon}^{num} - \boldsymbol{\varepsilon}^{exact}) \right\}^{1/2} \quad (4.109)$$

where $\mathbf{D}$ is the material constitutive matrix, for plane stress state

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \quad (4.110)$$

and $\boldsymbol{\varepsilon} = \{\varepsilon_x \quad \varepsilon_y \quad \varepsilon_{xy}\}^T$. A 4×4 Gaussian quadrature is found to be sufficient for

evaluating error norm integrals exactly. Figure 4.26 show that the present method in general gives a better convergence than EFG and QUAD4 for the same Gaussian integration order and same number of nodes. The symbol h refers to the characteristic size of elements and R refers to the convergence rate. The error energy norm of the cantilever beam subjected to shear force is shown in Table 4.20.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.20. Error energy norm of the cantilever beam subjected to shear force.

Mesh	h	QUAD4	EFG	FE-LSPIM QUAD4 (Present)	FE-LSPIM QUAD4U (Present)
2×8	6	0.7100	0.2389	0.1723	0.1736
4×16	3	0.3704	0.0836	0.0378	0.0388
6×24	2	0.2492	0.0457	0.0146	0.0151
8×32	1.5	0.1875	0.0298	0.0076	0.0078

Figure 4.26. Convergence of the error of energy norm.

4.4.7. A BEAM SUBJECT TO MID-SPAN POINT LOAD

Consider a beam of length, L , height, D , and unit thickness subjected to a mid-span point load, P (Fig. 4.27(a)). The beam is assumed to be in a state of plane stress. The mesh used for the purpose is shown in Fig. 4.27(b). The parameters are taken as $E = 3.0 \times 10^7$, $\nu = 0.3$, $D = 12$, $L = 48$, and $P = 10000$. Table 4.21 lists the computed maximum vertical displacement at the bottom surface of the beam (i.e., at point A). The results show a good agreement between the “overkill” solution and the present element.

Figure 4.27. (a) A beam subject to mid-span point load; (b) The finite element mesh used for problem shown in (a).

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.21. Maximum vertical displacement* of the beam at
point A under mid-span point load.

QUAD4	1.78
QM6	1.89
QUAD8	2.09
QUAD16	2.11
FE-LSPIM QUAD4 (Present)	2.08
FE-LSPIM QUAD4U (Present)	2.06
“Overkill” solution [#]	2.09

*Multiply all values by 10^{-3} .

[#]Obtained using ANSYS Plane 82 element with a 240×60 mesh.

4.4.8. A PLATE WITH A CENTRAL HOLE

Consider an infinite plate with a central hole under unidirectional tension in the x -direction (Fig. 4.28). The exact stress distribution for this problem is given by Timoshenko and Goodier [151] as follows:

$$\sigma_x(r, \theta) = 1 - \frac{a^2}{r^2} \left(\frac{3}{2} \cos 2\theta + \cos 4\theta \right) + \frac{3a^4}{2r^4} \cos 4\theta \quad (4.111)$$

$$\sigma_y(r, \theta) = -\frac{a^2}{r^2} \left(\frac{1}{2} \cos 2\theta - \cos 4\theta \right) - \frac{3a^4}{2r^4} \cos 4\theta \quad (4.112)$$

$$\sigma_{xy}(r, \theta) = -\frac{a^2}{r^2} \left(\frac{1}{2} \sin 2\theta + \sin 4\theta \right) + \frac{3a^4}{2r^4} \sin 4\theta \quad (4.113)$$

where a is the radius of the circular hole. The displacement components (with rigid-body displacement and rotation suppressed) are:

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

$$u(r, \theta) = \frac{a}{8\mu} \left[\frac{r}{a} (\kappa + 1) \cos \theta + 2 \frac{a}{r} ((1 + \kappa) \cos \theta + \cos 3\theta) - \frac{2a^3}{r^3} \cos 3\theta \right] \quad (4.114)$$

$$v(r, \theta) = \frac{a}{8\mu} \left[\frac{r}{a} (\kappa - 3) \sin \theta + 2 \frac{a}{r} ((1 - \kappa) \sin \theta + \sin 3\theta) - \frac{2a^3}{r^3} \sin 3\theta \right] \quad (4.115)$$

where μ is the shear modulus and κ is Kolosov constant, defined as

$$\mu = \frac{E}{2(1+\nu)} \quad \kappa = 3 - 4\nu \text{ (plane strain)} \quad \kappa = \frac{3-\nu}{1+\nu} \text{ (plane stress)} \quad (4.116)$$

Due to symmetry, only one quadrant of the problem, $0 \leq x \leq 5$ and $0 \leq y \leq 5$, is modelled. For numerical computations, the parameter value are taken $a = 1$, $\nu = 0.3$ and $E = 3 \times 10^7$. Traction-prescribed boundary conditions consistent with the exact solution are applied at the top and right edges and displacement-prescribed boundary conditions at the left and bottom edges as shown in Fig. 4.28.

Figure 4.28. Quarter model for the plate with a central hole.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Table 4.22. Comparison of error energy norm.

Method	Error energy norm
QUAD4	3.116982×10^{-05}
EFG	2.357944×10^{-05}
FE-LSPIM QUAD4	1.258103×10^{-05}
FE-LSPIM QUAD4U	1.636480×10^{-05}

Table 4.22 shows the error energy norm of the results for the four methods. The definition of the error energy norm is the same as Equation (4.110). It is observed that the errors given by the FE-LSPIM QUAD4 element and FE-LSPIM QUAD4U element are smaller than that of other elements although the order is the same.

The v -displacement and σ_x - stress along $x = 0$, obtained with all the three methods (present, QUAD4 and EFG) as well as the exact solution, are plotted in Figs. 4.29 and 4.30, respectively. It is seen that the displacement and stress values given by FE-LSPIM QUAD4 and FE-LSPIM QUAD4U as well as EFG are very close to the exact solution.

Figures 4.31-4.32 show the contour plots of displacements. These plots show that the contour plots obtained by all the methods are smooth and continuous.

Figures 4.33-4.35 show the contour plots of σ_x stress. These Figs. show that the contour plots obtained by the FE-LSPIM QUAD4 and FE-LSPIM QUAD4U are better than that of QUAD4 and comparable with that of EFG.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.29. v -displacement distribution for the plate with a central hole subjected to a uni-axial tensile load along the $x = 0$ edge.

Figure 4.30. σ_x -stress distribution for the plate with a central hole subjected to a uni-axial tensile load along the $x = 0$ edge.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.31. The contour plots of the u -displacement for the plate with a central hole.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.32. The contour plots of the v -displacement for the plate with a central hole.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.33. Contour plots of σ_x -stress for the plate with a central hole.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.34. The contour plots of σ_y -stress for the plane with a central hole.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Figure 4.35. The contour plots of σ_{xy} -stress for the plane with a central hole.

4.5. CONCLUDING REMARKS

In this chapter, two new QUAD4 elements termed as FE-LSPIM QUAD4 and FE-LSPIM QUAD4U have been developed based on the concept of PU-FEM. The shape functions developed in this paper possess inter-element compatibility properties inherited from finite element method and higher order (quadratic) polynomial completeness properties inherited from meshfree (PIM) methods. The following are the important observations from the present work:

1. The shape functions proposed in this paper have Kronecker property and ensure inter-element compatibility as in FEM. In view of the Kronecker property of shape functions, enforcing the prescribed nodal displacement boundary conditions is easy.
2. For the linear, quadratic and cubic displacement field problems tested, the performance of the present element is superior to that of QUAD4 element, and comparable to that of EFG (meshfree) method and QUAD16 element.
3. The present element exhibits excellent tolerance to the mesh distortions. When the mesh distortions are present, its performance is superior to that of EFG. This has been demonstrated for quadratic and cubic displacement field problems.
4. The results of several other elements (Q4, Q6, QM6, QUAD8, QUAD16, P2-Q6-I, AGQ6-I) have also been compared with that of the present elements. Same element density has been used for computing the results of different elements. It has been observed that the performance of the present elements is comparable to that of QUAD16. This suggests that if same nodal density were used, then the performance of the present elements could have been better.

CHAPTER 4. FE-MESHFREE FORMULATION BASED ON FEM SHAPE FUNCTIONS
AND LEAST-SQUARE PIM SHAPE FUNCTIONS

Although the shape functions developed in this paper possess Kronecker delta property and satisfy inter-element compatibility property, fulfillment of exact displacement boundary conditions all along the edges is not guaranteed by simply enforcing the displacement boundary conditions at the nodes on these edges. Thus, penalty method or Lagrange multiplier method may have to be used for the purpose. This aspect is rather similar to the meshfree methods. Furthermore, the order of continuity across the element boundaries remains that of finite element despite the use of composite shape functions in the interpolation.

The proposed hybrid approach of combining the concepts of finite element method and meshfree methods can be extended to other 2D elements as well as 3D elements. In spite of the advantages of the method, the requirement of a FE mesh for the present method is a demerit as compared to meshfree methods, particularly for 3D problems where the computer time spent on mesh generation is not trivial. Nevertheless, the quality of mesh (in terms of mesh distortion) is rather less critical in the proposed method than in the classical finite element method.

CHAPTER 5. FE-LSPIM QUAD4 ELEMENT FOR FREE VIBRATION ANALYSIS

The FE-LSPIM QUAD4 element, which was introduced for static analysis in Chapter 4, uses new shape functions that combine the meshfree and finite element shape functions so as to synergize the individual strengths of meshfree and finite element methods. Thereby the element inherits the completeness properties of the meshfree shape functions and compatibility properties of the finite element shape functions. In this chapter, the element is extended to the free vibration analysis of two-dimensional solids. Typical benchmark problems are solved to test the performance of the element. The element gives more efficient solution than other peer elements particularly under distorted mesh. The FE-LSPIM QUAD4U, which was also developed in Chapter 4, is not considered here because it is an unsymmetric element and may, therefore, give rise to complex eigenvalues that are difficult to interpret.

5.1. INTRODUCTION

In the last few decades, several powerful numerical techniques have been developed to solve vibration problems in solid and structural mechanics. Typical of them are Rayleigh–Ritz method, finite element method (FEM) and meshfree methods. Out of these, the finite element method has been used extensively by many engineers and researchers for solving many complex problems encountered in practice. Although the

finite element method based on isoparametric concepts has been extensively used by many engineers and researchers for solving complex problems encountered in practice, it still has the drawback of being sensitive to mesh distortions. The meshfree methods have emerged as a potential alternative to finite element method in this regard as they dispense with the very necessity of a mesh to discretize the problem.

Compared to the literature on the application of meshfree methods to static analysis, the literature on the application to vibration problems is quite limited. Ouatuati and Johnson [158] were the first to use EFG method to calculate natural frequencies and modes of beams and plates for the first time. Gu and Liu [109] extended the local radial point interpolation method (LRPIM) to free vibrations analysis. Zhou et al. [159] use the RKPM to calculate the natural frequencies of beams and plates with various boundary conditions such as free, simply supported and fully clamped edges. Liew et al. [160] applied the reproducing kernel particle method to free vibration analysis of rotating cylindrical shells. Li et al. [161] proposed a meshfree Local Kriging (LoKriging) method for the free/forced vibrations of 2-D structures and dynamic response of MEMS device. Bel et al. [162] proposed a new approach for the forced vibration of thin plates up to the medium frequency range. Their approach is based on the partition of unity method (PUM). Like finite element method, the meshfree methods too are not free from drawbacks. The lack of Kronecker delta property of meshfree shape functions, for example, renders implementation of essential boundary conditions awkward. The meshfree methods are also computationally more expensive than the finite element methods. The nodal shape functions of meshfree methods are in general

complex in nature and this renders difficult an accurate numerical evaluation of stiffness matrix using numerical methods like Gaussian quadrature.

There have been new efforts in recent years to develop hybrid methods synergising the merits of FE and meshfree methods. Examples include the moving particle method proposed by Hao et al. [144, 145] and the partition unity (PU) based finite element method (FEM) proposed by Tian and Yagawa [139]. This approach uses a meshfree approximation in combination with the usual finite element shape functions.

The FE-LSPIM QUAD4 element developed in Chapter 4 belongs to the class of hybrid elements. This element employs composite shape functions which are obtained as a product of finite element and meshfree shape functions. As a result, the shape functions possess inter-element compatibility properties inherited from the finite element method and higher order (quadratic) polynomial completeness in physical space inherited from the meshfree (PIM) methods. The element has excellent tolerance to mesh distortion and implementing this element in an existing FE codes is rather straightforward. The former renders the method more attractive than the finite element method and the latter offers an advantage over the meshfree methods.

In this Chapter, FE-LSPIM QUAD4 element is applied to the free vibration analysis of two dimensional solids. In Section 5.2, the mass and stiffness matrices are derived. In Section 5.3, some computational aspects are discussed. In Section 5.4, numerical tests are carried out to assess the performance of the proposed FE-LSPIM QUAD4 element. Finally, important observations are summarized in section 5.5.

5.2. THE EIGENVALUE PROBLEM FOR FREE VIBRATION ANALYSIS

Using the FE-LSPIM shape functions derived in Chapter 4, the eigenvalue problem associated with the free vibration analysis is obtained as

$$\mathbf{K}\boldsymbol{\phi} - \lambda\mathbf{M}\boldsymbol{\phi} = \mathbf{0}, \quad \lambda = \omega^2 \quad (5.1)$$

where λ and ω are the eigenvalue and circular frequency, respectively, corresponding to mode shape $\boldsymbol{\phi}$. The stiffness matrix, $\mathbf{K}$, and mass matrix, $\mathbf{M}$, are obtained by assembling the element level counterparts defined by

$$\mathbf{K}^{(e)} = \int_{V^{(e)}} \mathbf{B}^T \mathbf{D} \mathbf{B} dV^{(e)} \quad (5.2)$$

$$\mathbf{M}^{(e)} = \int_{V^{(e)}} \rho \mathbf{Q}^T \mathbf{Q} dV^{(e)} \quad (5.3)$$

$$\mathbf{Q} = \begin{bmatrix} \Psi_1 & 0 & \Psi_2 & 0 & \Psi_3 & 0 & \dots & \Psi_M & 0 \\ 0 & \Psi_1 & 0 & \Psi_2 & 0 & \Psi_3 & \dots & 0 & \Psi_M \end{bmatrix} \quad (5.4)$$

$$\mathbf{B} = \begin{bmatrix} \Psi_{1,x} & 0 & \Psi_{2,x} & 0 & \Psi_{3,x} & 0 & \dots & \Psi_{M,x} & 0 \\ 0 & \Psi_{1,y} & 0 & \Psi_{2,y} & 0 & \Psi_{3,y} & \dots & 0 & \Psi_{M,y} \\ \Psi_{1,y} & \Psi_{1,x} & \Psi_{2,y} & \Psi_{2,x} & \Psi_{3,y} & \Psi_{3,x} & \dots & \Psi_{M,y} & \Psi_{M,x} \end{bmatrix} \quad (5.5)$$

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \quad (\text{for plane stress problems}) \quad (5.6)$$

5.3. PSEUDO CODE FOR IMPLEMENTING FE-LSPIM QUAD4 FOR THE FREE VIBRATION ANALYSIS

Loop over the nodes in the mesh:

For each node, determine the number of nodes, N , in its support and save this information.

End loop over nodes

Loop over the elements in the mesh:

Loop over quadrature points in each element:

Determine the number of nodes, M , in the support of the element.

Compute Φ_i for each node of the element and assemble it as described

$1 \times N$

in Chapter 4 to form Φ .

$4 \times M$

Compute $\mathbf{N}$ and then compute Ψ .

1×4

$1 \times M$

Compute $\mathbf{N}_{,x}$ and $\mathbf{N}_{,y}$ as in the classical FEM.

1×4

1×4

Compute $\Phi_{,x}$ and $\Phi_{,y}$.

$4 \times M$

$4 \times M$

Construct $\mathbf{Q}$ using Equation (5.4).

$2 \times 2M$

Construct $\mathbf{B}$ using Equation (5.5).

$3 \times 2M$

Compute $\mathbf{B}^T \mathbf{D} \mathbf{B}$, add quadrature point contribution to element stiffness matrix

$\mathbf{K}^{(e)}$ as in FEM.

Compute $\rho \mathbf{Q}^T \mathbf{Q}$, add quadrature point contribution to mass matrix $\mathbf{M}^{(e)}$ as in FEM.

End quadrature point loop.

Assemble element stiffness matrix into the global stiffness matrix.

Assemble mass matrix into the global force vector.

End element loop.

Solve the eigenvalue problem for eigenvalues and eigenvectors.

5.4. NUMERICAL EXAMPLES

5.4.1. A CANTILEVER BEAM

(a)

(b)

(c)

Figure 5.1. Free vibration analysis of a cantilever beam. (a) The geometry of the cantilever beam; (b) Coarse mesh; (c) Fine mesh.

The FE-LSPIM QUAD4 element is used to analyze the free vibration of a cantilever beam (Fig. 5.1(a)) with a material Young's modulus of 21000 kgf/mm^2 , Poisson's ratio of 0.3, thickness of 1 mm and mass density of $8.0 \times 10^{-10} \text{ kgfs}^2/\text{mm}^4$. Two kinds of meshes are used to model the problem, a coarse mesh and a fine mesh with (2×40) and (5×50) elements, respectively, as shown in Figs. 5.1(b) and 5.1(c) respectively. This

problem has been analyzed by Nagashima [163] using the node-by-node meshfree (NBNM) method and Gu and Liu [109] using a local radial point interpolation method (LRPIM). The multi-quadrics (MQ) radial function and Gaussian (Exp) radial function are used in this paper. The frequencies computed for the two meshes are listed in Tables 5.1-5.2. The reference results shown in Tables 5.1 and 5.2 are obtained by Gu and Liu [109] using the commercial FE software, ABAQUS. A very fine mesh with 8000 degrees of freedom is used for the purpose. From Tables 5.1-5.2, it is observed that the results obtained using the present element are in very good agreement with those obtained using LRPIM [109] and NBNM [163] methods. Even for the coarse mesh, the results given by the present method are close to the reference solution. The present element is more efficient in predicting the frequencies than the other elements.

The first ten eigenmodes, obtained with the present element using the coarse mesh (mesh b) are plotted in Fig. 5.2. These mode shape plots compare well with that of Nagashima [163]. (The mode shapes of higher order bending modes shown in Fig. 5.2 seem to give a distorted picture that the zero-slope boundary condition at the left end of the beam is violated. This apparent discrepancy is due to the high scaling using for plotting the mode shapes.)

Table 5.1. Comparison of computed frequencies (Hz) for the cantilever beam with a coarse mesh (Mesh b).

Mode	LRPIM (MQ)		NBNM		FEM (ABAQUS)		FE-LSPIM QUAD4		Reference[109]
	[109] (126 d.o.f)		[163] (126 d.o.f)		(126 d.o.f)		(Present) (126 d.o.f)		(8000 d.o.f.)
	Frequency	% Error	Frequency	% Error	Frequency	% Error	Frequency	% Error	Frequency
1	889	7.97	926	12.53	870	5.71	824	0.17	823
2	5310	7.55	5484	11.08	5199	5.31	4971	0.68	4937
3	12829	0.04	12832	0.06	12830	0.05	12828	0.03	12824
4	13964	7.37	14201	9.20	13640	4.88	13177	1.32	13005
5	25311	7.11	25290	7.02	24685	4.46	24141	2.16	23632
6	38463	6.72	37350	3.60	37477	3.99	37198	3.21	36040
7	38488	0.12	38321	0.32	38378	0.17	38457	0.04	38442
8	52833	6.48	50819	2.42	51322	3.44	51947	4.69	49616
9	64012	0.09	63284	1.05	63584	0.58	64000	0.07	63955
10	67933	6.20	63994	0.04	65731	2.76	68363	6.87	63967

Table 5.2. Comparison of computed frequencies (Hz) for the cantilever beam with a fine mesh (Mesh c).

Mode	LRPIM (MQ)		NBNM		FEM (ABAQUS)		FE-LSPIM QUAD4		Reference[109]	
	[109] (126 d.o.f)	% Error	Frequency	% Error	[163] (126 d.o.f)	% Error	(Present) (126 d.o.f)	% Error	(8000 d.o.f.)	
	Frequency		Frequency		Frequency		Frequency		Frequency	
1	824	0.16	844	2.57	830	0.85	822	0.07	823	
2	4977	0.80	5051	2.31	4979	0.85	4934	0.05	4937	
3	12827	0.02	12828	0.03	12826	0.02	12825	0.01	12824	
4	13094	0.68	13258	1.95	13111	0.82	12999	0.05	13005	
5	23782	0.63	23993	1.53	23818	0.79	23625	0.03	23632	
6	36258	0.61	36432	1.09	36308	0.74	36036	0.01	36040	
7	38452	0.02	38436	0.01	38436	0.02	38447	0.01	38442	
8	49911	0.59	49937	0.65	49958	0.69	49622	0.01	49616	
9	63988	0.05	63901	0.08	63917	0.06	63980	0.03	63955

Figure 5.2. First ten vibration modes of the cantilever beam computed using present element.

In order to study the convergence of computed frequencies and the sensitivity of the present element to mesh distortion, a sequence of four distorted meshes are considered as shown in Fig. 5.3. The results obtained with three element types viz., the present element, QUAD4 and QM6 [18], are shown in Tables 5.3, 5.4 and 5.5, respectively. In this chapter, the results of QUAD4 and QM6 have been obtained using ANSYS. Figure

5.4 shows the plot of error in the first two natural frequencies obtained using the present element as well as QUAD4 and QM6. It is seen that the error given by the present element is generally less than those given by QUAD4 and QM6. The results obtained from the present element also exhibit a faster convergence than those from QUAD4 and QM6.

Figure 5.3. The sequence of meshes used for the convergence study.

Figure 5.4. Convergence of the error in the computed frequency for the first two modes.

Table 5.3. Convergence of computed natural frequencies (Hz) for the cantilever beam using the FE-LSPIM QUAD4 (**present**) element.

Mode	1×1 super element	2×2 super element	4×4 super element	8×8 super element	Reference[109] 8000 d.o.f.
1	982.24	995.9	852.7	824.5	823
2	10165	9219	6130	4994	4937
3	12850	12906	12852	12834	12824
4	46808	41094	18023	13448	13005
5	175647	75285	38656	25401	23632
6	181525	91988	40971	38479	36040
7	239756	160186	65928	40840	38442
8	293709	168085	98076	60309	49616
9	295621	168792	123219	64081	63955

Table 5.4. Convergence of computed natural frequencies (Hz) for the cantilever beam using QUAD4.

Mode	1×1 super element	2×2 super element	4×4 super element	8×8 super element	Reference[109] 8000 d.o.f.
1	3411.3	2318.4	1425.4	1012.8	823
2	13559	11781	8446	6107	4937
3	52496	13062	12907	12854	12824
4	57509	42576	22446	16366	13005
5	178012	80667	39695	30563	23632
6	188192	89335	39853	38800	36040
7	296843	128172	69248	48170	38442
8	305672	134010	101413	65445	49616
9	356988	165612	136564	68502	63955

Table 5.5. Convergence of computed natural frequencies (Hz) for the cantilever beam using QM6.

Mode	1×1 super element	2×2 super element	4×4 super element	8×8 super element	Reference[109] 8000 d.o.f.
1	2913.3	1992.1	1294.9	968.4	823
2	13281	10135	7321.6	5786.7	4937
3	17538	12970	12874	12841	12824
4	53049	30714	19126	15333	13005
5	162922	41619	35166	28261	23632
6	179778	69895	39440	38721	36040
7	221902	79924	64637	43980	38442
8	294092	105522	68137	62019	49616
9	352919	131182	95683	65157	63955

5.4.2. A SHEAR WALL

Figure 5.5 shows a shear wall with four openings, which has been solved using boundary element method by Brebbia et al. [164]. The problem has been solved assuming plane stress state with $E = 1000$, $\nu = 0.2$, $t = 1.0$ and $\rho = 1$. The problem has also been solved by them using ABAQUS. Natural frequencies of the first eight modes are computed using the present element and listed in Table 5.6. Here again, it is seen that the results given by the present element are in good agreement with those obtained by other methods.

Figure 5.5. A shear wall with four openings.

Table 5.6. Natural frequencies of the shear wall.

Mode	LRPIM(MQ) ⁺ (1118 d.o.f) [109]	ABAQUS (1118 d.o.f) [109]	Brebbia [164] (BEM)	QM6	FE-LSPIM QUAD4 (1118 d.o.f) (Present)
1	2.086	2.073	2.079	2.057	2.064
2	7.152	7.096	7.181	7.067	7.071
3	7.647	7.625	7.644	7.620	7.625
4	12.019	11.938	11.833	11.840	11.895
5	15.628	15.341	15.947	15.313	15.288
6	18.548	18.345	18.644	18.342	18.315
7	20.085	19.876	20.268	19.887	19.843
8	22.564	22.210	22.765	22.236	22.185

5.4.3. DISTORTION SENSITIVITY TEST

The present element has also been tested for its sensitivity to mesh distortion. The cantilever beam in Fig. 5.1(a) is considered again with the distorted mesh as shown in Fig. 5.6. Distortion parameters δ and η are introduced to characterize the extent of distortion. The natural frequency has been computed using the present element as well as QUAD4 and QM6 for different values of δ and η . A comparison of computed frequencies (Hz) is shown in Table 5.7. A comparison of percent error in the computed frequencies is shown in Fig. 5.7. It seen that the error given by the present element does not change appreciably with the increase in distortion parameters while the error with the QUAD4 and QM6 shows rapid increase. Thus, the present element is almost insensitive to mesh distortion.

Figure 5.6. The mesh used for the distortion sensitivity test.

Table 5.7. Computed natural frequencies (Hz) for the distortion sensitivity test.

	$\delta = 0, \eta = 0$	$\delta = 10, \eta = 1$	$\delta = 20, \eta = 1$	$\delta = 30, \eta = 1$
QUAD 4	2619.5	3411.3	3544.6	3693.9
QM6	852.52	2913.3	2602.3	2133.5
FE-LSPIM QUAD4 (Present)	979.51	982.24	984.49	986.48
Reference[109]	823

Figure 5.7. The percent error in computed frequencies for distortion sensitivity test.

5.4.4. RAYLEIGH-LAMB PROBLEM

To illustrate the numerical accuracy of the present method, the well known Rayleigh and Lamb problem is studied here [165,166]. The problem has been solved assuming plane strain state with $E = 210 \times 10^9$, $\nu = 0.3$, $t = 1.0$ and $\rho = 7860$. Rectangular domain is considered with length $1m$ (half of wave length λ) and depth $12m$. A periodic boundary condition is applied on the two vertical sides and the bottom edge is fixed on the y direction. The depth to length ratio has been chosen sufficiently high (12:1) so that the fixed boundary condition at the bottom edge does not affect the Lamb frequencies computed. Table 5.8 lists the relative errors in the computed Lamb frequencies as well as the computational times for different meshes for this problem. A MATLAB program

has been used for the purpose with a 2×2 Gaussian integration for computing the stiffness matrix. It is seen that the computational times of the present method are comparable to that of EFG while the accuracy of solution is far better than that of EFG. For the computational times of the same order (shown in bold face), the present method gives the least error which shows that the present method is computationally more efficient than the other two methods for this problem.

Table 5.8. Comparison of computed frequencies (Hz) and CPU time for the Rayleigh-Lamb problem.

Mesh	QUAD4		EFG		FE-LSPIM QUAD4 (Present)	
	Error	time	Error	time	Error	time
24×2	19.444	0.2	4.590	2.2	0.181	0.67
48×4	4.375	2.8	0.372	4.2	0.007	4.67
60×5	2.828	11	0.130	13.4	0.002	13.9
72×6	1.973	38.5	0.083	42.2	0.001	41
Rayleigh frequencies 1486.47 [165]

Figure. 5.8. The Rayleigh-Lamb problem.

5.5. CONCLUDING REMARKS

The FE-LSPIM QUAD4 element has been applied to the free vibration analysis of two-dimensional solids. The FE-LSPIM shape functions have been used for the formulation of this element. As the FE-LSPIM shape functions possess the *Kronecker delta* properties, the displacement boundary conditions can be easily applied. Numerical results obtained using FE-LSPIM QUAD4 has been compared with those obtained using other solution techniques. The following are some of the major observations:

1. For regular meshes, the results obtained using the FE-LSPIM QUAD4 are in very good agreement with those obtained using LRPIM [109] and NBNM [163] methods.
2. For coarse meshes, the accuracy of results given by the present element is generally better than those given by ABAQUS, LRPIM [109] and NBNM [163].
3. For distorted meshes, the results given by the present element show a faster convergence and are also more efficient than those obtained using QM6. The distortion sensitivity test conducted shows that the present element is highly tolerant to mesh distortion. This is attributed due to the fact that the element shape functions possess higher order completeness, i.e., they can reproduce all the monomial terms $(1, x, y, x^2, xy, y^2, x^2y, xy^2)$ considered in the interpolation, a property inherited from the meshfree method (PIM), and at the same time they satisfy the compatibility conditions, a property inherited from finite element method.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

The FE-LSPIM QUAD4 element developed in Chapter 4 is extended to geometrically nonlinear problems in this chapter. The geometrical nonlinear formulation is derived based on the updated Lagrangian formulation. The performance of the proposed element is studied under various numerical examples. The results are compared with those obtained from the commercial software, ANSYS. Numerical examples show that the performance of FE-LSPIM QUAD4 is at least as efficient as the QM6 and better than QUAD4 for regular meshes, and is less sensitive to the mesh distortions compared to the QM6 and QUAD4.

6.1. INTRODUCTION

In nonlinear solid mechanics problems, the relationship between the applied external forces and resulting displacements is nonlinear. The nonlinear relationship can be caused by nonlinear material response (*i.e.*, material nonlinearity) or by large displacements and rotations (*i.e.*, geometrical nonlinearity). In this chapter, the geometrically nonlinear behavior is studied, wherein changes in geometry have a significant effect on the load displacement characteristics of solids bodies. In the linear description of the motion of solids bodies, one assumes that the displacements and strains are very small and the material is linearly elastic. This assumption allows ignoring the changes in the geometry of body while proceeding to determine the deformation for the next load step. In addition, the equilibrium equations can still be derived using the undeformed configuration of the body.

Ever since the pioneering work of Kohnke [167] for large deformation analysis, which introduced the mathematical programming approach for the analysis of geometrically nonlinear structural behavior and applied it to the analysis of frame structures, many papers on nonlinear analysis have been published in the last decade. Following is some of them. Basar and Itskov [168] proposed a fully nonlinear shell theory including both the membrane and bending components in the field of large elastoplastic deformations. Stephane and Thomas [169] developed an object-oriented finite element program for the nonlinear structural and continuum analysis. Von Mises plasticity is used as an illustration of constitutive theory in this paper. Liu and Hong [170] developed the geometric nonlinear formulation for dynamic stiffening of a rectangular plate undergoing large overall motions. All of these analyses were based on the finite element method (FEM).

Although the finite element method has been the most popular tool for solving various non-linear problems, the remeshing procedure is still a main drawback for analyzing problems undergoing large deformation. When a continuum body deforms severely, the remeshing process is necessary to avoid severe mesh distortions. However, this process is not only a burdensome task but also reduces the accuracy of the numerical solutions. Due to the flexibility in solving practical engineering problems, meshfree methods have been introduced to nonlinear analysis in recent years. Liu and coworkers [171,172] applied the Reproducing Kernel Particle Method (RKPM) to large deformation problems. In their work, the concept of a multiple-scale measure is proposed and an efficient algorithm for simulation of large deformation problems has been developed. Li et al. [173] extended the RKPM to Large deformation analysis of non-linear elastic and

inelastic structures. Chen et al. [174] presented the meshfree simulations of large deformation of thin shell structures. Liew et al. [175] applied the RKPM to the analysis of two-dimensional large deformation problems. In their work, the displacement boundary conditions are imposed using the penalty method. In the above analyses, Lagrangian and updated Lagrangian reproducing kernel (RK) shape functions are used to approximate the field variables.

The FE-LSPIM QUAD4 element has been developed in Chapter 4 and extended to free vibration analysis in Chapter 5. In this Chapter, the FE-LSPIM QUAD4 element is applied to two dimensional geometrically nonlinear analyses. The effectiveness of the present element is evaluated using typical numerical problems.

The outline of this chapter is as follows. Section 6.2 introduces the notation used. Section 6.3 summarises the updated Lagrangian formulation for continuum problems. Section 6.4 presents several numerical examples to test the performance of the FE-LSPIM QUAD4. Finally, important observations are summarised in section 6.5.

6.2. NOTATION

In the geometrically nonlinear analysis, the final configuration C is determined from a known initial configuration C_0 by assuming that the total load is applied in increments so that the body eventually occupies the final configuration. In the determination of an intermediate configuration C_i , the previously known configurations $C_0, C_1 \dots C_{i-1}$ are used as the reference configuration C_R . If the initial configuration is used as reference configuration, it is called the total Lagrangian description. If the latest known

configuration C_{i-1} is used as the reference configuration, it is called the updated Lagrangian description.

Figure 6.1. Initial and two consecutive configurations of a body.

The notation used for the positions, displacements, strains, stresses, etc are introduced in this section. Three equilibrium configurations of the body, namely, C_0 , C_1 and C_2 are considered in this chapter as shown in Fig. 6.1. The three configurations of the body refer to the initial undeformed configuration C_0 , the last known deformed configuration C_1 , and the current configuration C_2 . It is assumed that the deformation of the body from C_1 to C_2 is small and the accumulated deformation of the body from C_0 to C_1 can be arbitrarily large but continuous.

The following notation is used in this chapter. A left superscript refers to the configuration in which the quantity occurs, and a left subscript denotes the

configuration with respect to which the quantity is measured. Thus ${}^i_j Q$ indicate that the quantity Q occurs in configuration C_i but measured in C_j . When the quantity is measured in the same configuration in which it occurs, the left subscript may be omitted. The left superscript will be omitted on incremental quantities that occurs between C_1 and C_2 . The right subscripts refer to components in rectangular Cartesian coordinate systems.

The total displacement of the particle $\mathbf{x}$ in the two configurations C_1 and C_2 can be written as

$${}^1_0 \mathbf{u} = {}^1 \mathbf{x} - {}^0 \mathbf{x} \quad (6.1)$$

$${}^2_0 \mathbf{u} = {}^2 \mathbf{x} - {}^0 \mathbf{x} \quad (6.2)$$

and the displacement increment of the point from C_1 to C_2 is written as

$$\mathbf{u} = {}^2_0 \mathbf{u} - {}^1_0 \mathbf{u} \quad (6.3)$$

6.3. UPDATED LAGRANGIAN FORMULATION

6.3.1. PRINCIPLE OF VIRTUAL DISPLACEMENTS

The derivation of the updated Lagrangian formulation follows the work of Reddy [176]. The equation of the Lagrangian incremental description of motion can be derived from the principle of virtual work. The principle requires that the sum of the external virtual work and internal virtual work should be equal to zero. It can be expressed in terms of the displacements as

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

$$\delta W \equiv \int_{\Omega} {}^2\boldsymbol{\sigma} : \delta({}_2\mathbf{e}) d^2\Omega - \int_{\Omega} {}^2\mathbf{f} \cdot \delta\mathbf{u} d^2\Omega - \int_{\Gamma} {}^2\mathbf{t} \cdot \delta\mathbf{u} d^2\Gamma = 0 \quad (6.4a)$$

$$= \int_{\Omega} {}^2\sigma_{ij} \delta({}_2e_{ij}) d^2\Omega - \int_{\Omega} {}^2f_i \delta u_i d^2\Omega - \int_{\Gamma} {}^2t_i \delta u_i d^2\Gamma = 0 \quad (6.4b)$$

where ${}_2e_{ij}$ is the infinitesimal strain tensor

$${}_2e_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (6.5)$$

where ${}^2\Omega$ and ${}^2\Gamma$ are the domain and boundary, and ${}^2\mathbf{f}$ and ${}^2\mathbf{t}$ are the body force and the surface traction, respectively, in the current configuration C_2 . Equation (6.4) cannot be solved directly because the configuration C_2 is unknown. This is different from the linear analysis, in which the displacements are assumed infinitesimally small. In a large deformation analysis, the configuration of the body is changing continuously. This change in configuration is dealt with by defining appropriate stress and strain measures. The *second Piola-Kirchhoff stress tensor* and the *Green-Lagrange strain tensor* are used for this purpose. They are defined as:

$$\mathbf{E} = \frac{1}{2} (\mathbf{F}^T \mathbf{F} - \mathbf{I}) \quad (6.6a)$$

$$= \frac{1}{2} \left[\left(\frac{\partial \mathbf{u}}{\partial \mathbf{X}} \right)^T + \left(\frac{\partial \mathbf{u}}{\partial \mathbf{X}} \right) + \left(\frac{\partial \mathbf{u}}{\partial \mathbf{X}} \right)^T \left(\frac{\partial \mathbf{u}}{\partial \mathbf{X}} \right) \right] \quad (6.6b)$$

$\mathbf{F}$ is the *deformation gradient tensor*, which gives the relationship of between a material line $d\mathbf{X}$ before deformation to the line $d\mathbf{x}$ after deformation. It is defined as

$$\mathbf{F} = \left(\frac{\partial \mathbf{x}}{\partial \mathbf{X}} \right)^T \quad (6.7)$$

The Cartesian component form of the *Green-Lagrange strain tensor* can be written as

$$E_{ij} = \frac{1}{2} \left(\frac{\partial x_m}{\partial X_i} \frac{\partial x_m}{\partial X_j} - \delta_{ij} \right) \quad (6.8a)$$

$$= \frac{1}{2} \left(\frac{\partial u_i}{\partial X_j} + \frac{\partial u_j}{\partial X_i} + \frac{\partial u_k}{\partial X_i} \frac{\partial u_k}{\partial X_j} \right) \quad (6.8b)$$

The second *Piola-Kirchhoff stress* is defined as

$$\mathbf{S} = J\mathbf{F}^{-1} \cdot \boldsymbol{\sigma} \cdot \mathbf{F}^{-T} \quad (6.9)$$

where J denote the determinant of $\mathbf{F}$.

6.3.2. CONSTITUTIVE EQUATIONS

If the constitutive behavior of the materials is only a function of the current state of deformation, the materials are known as *elastic*. In the special case, if constitutive behavior of the materials is dependent on the initial state and the current configuration, the material is called *hyperelastic*. For hyperelastic materials, a stored strain energy function, U_0 , per unit underformed volume, is assumed. In this case, the components of the *material elasticity tensor* $\mathbf{C}$ are given by

$$C_{ijkl} = \frac{\partial S_{ij}}{\partial E_{kl}} \quad (6.10)$$

When deriving the finite element models of incremental nonlinear analysis of solid continua, it is necessary to specify the stress-strain relations in incremental form. In the undated Lagrangian formulation, it can be expressed in terms of the updated Kirchhoff stress increment tensor components ${}_1S_{ij}$ and updated Green-Lagrange strain increment tensor components ${}_1\varepsilon_{kl}$:

$${}_1S_{ij} = {}_1C_{ijkl} {}_1\varepsilon_{kl} \quad (6.11)$$

where ${}_1C_{ijkl}$ is the incremental constitutive tensors with respect to the configurations C_1 . The following transformation rules hold between the components of elasticity tensor $\mathbf{C}$ in different configurations.

$${}_1C_{ijkl} = \frac{{}_1\rho}{{}_0\rho} \frac{\partial^1 x_i}{\partial^0 x_p} \frac{\partial^1 x_j}{\partial^0 x_q} \frac{\partial^1 x_k}{\partial^0 x_r} \frac{\partial^1 x_l}{\partial^0 x_s} {}_0C_{pqrs} \quad (6.12)$$

With this equation, only ${}_1C_{ijkl}$ or ${}_0C_{pqrs}$ need to be given, the other sets of coefficients can be obtained by transformation. $\mathbf{C}$ is often assumed constant in the updated Lagrangian finite element formulations. The error introduced by this assumption can be negligible if the strains are relatively small but the difference can be significant in the large deformation problems. For two dimensional plain stress problem,

$$\mathbf{C} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \quad (6.13)$$

6.3.3. UPDATED LAGRANGIAN FORMULATION

In the updated Lagrangian formulation, all quantities are referred to the latest known configuration C_1 . Hence, the virtual work statement in Equation (6.4b) must be rewritten in terms of quantities referred to C_1 .

$$\begin{aligned} \delta W &= \int_{{}_1\Omega} {}^2S_{ij} \delta({}_1^2\varepsilon_{ij}) d^1\Omega - \int_{{}_1\Omega} {}^2f_i \delta u_i d^1\Omega - \int_{{}_1\Gamma} {}^2t_i \delta u_i d^1\Gamma \\ &= \int_{{}_1\Omega} {}^2S_{ij} \delta({}_1^2\varepsilon_{ij}) d^1\Omega - \delta({}_1^2R) = 0 \end{aligned} \quad (6.14)$$

where, 2f_i and 2t_i are body force and boundary traction components referred to the configuration C_1 , and the Cartesian components of the updated Green-Lagrange strain tensors, ${}_1^2\varepsilon_{ij}$, in the configurations C_2 with respect to configuration C_1 are defined by

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

$$\begin{aligned} {}^2_1\varepsilon_{ij} &= \frac{1}{2} \left(\frac{\partial u_i}{\partial {}^1x_j} + \frac{\partial u_j}{\partial {}^1x_i} + \frac{\partial u_k}{\partial {}^1x_i} \frac{\partial u_k}{\partial {}^1x_j} \right) \\ &\equiv {}_1e_{ij} + {}_1\eta_{ij} \end{aligned} \quad (6.15)$$

where,

$$u_i = {}^2x_i - {}^1x_i \quad (6.16)$$

$${}_1e_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial {}^1x_j} + \frac{\partial u_j}{\partial {}^1x_i} \right) \quad (6.17)$$

$${}_1\eta_{ij} = \frac{1}{2} \left(\frac{\partial u_k}{\partial {}^1x_i} \frac{\partial u_k}{\partial {}^1x_j} \right) \quad (6.18)$$

The virtual strain is given by

$$\delta({}^2_1\varepsilon_{ij}) = \delta({}_1e_{ij}) + \delta({}_1\eta_{ij}) \quad (6.19)$$

$$\delta({}_1e_{ij}) = \frac{1}{2} \left(\frac{\partial \delta u_i}{\partial {}^1x_j} + \frac{\partial \delta u_j}{\partial {}^1x_i} \right) \quad (6.20)$$

$$\delta({}_1\eta_{ij}) = \frac{1}{2} \left(\frac{\partial \delta u_k}{\partial {}^1x_i} \frac{\partial u_k}{\partial {}^1x_j} + \frac{\partial u_k}{\partial {}^1x_i} \frac{\partial \delta u_k}{\partial {}^1x_j} \right) \quad (6.21)$$

and

$$\delta({}^2_1R) = \int_{{}_1\Omega} {}^2_1f_i \delta u_i d^1\Omega + \int_{{}_1\Gamma} {}^2_1t_i \delta u_i d^1\Gamma \quad (6.22)$$

The updated *Kirchhoff stress* is decomposed as

$${}^2_1S_{ij} = {}^1\sigma_{ij} + {}_1S_{ij} \quad (6.23)$$

where ${}^1\sigma_{ij} = {}^1S_{ij}$ are Cauchy stress components in C_1 configuration and ${}_1S_{ij}$ are the components of *updated Kirchhoff stress increment tensor*.

$${}^1\sigma_{ij} = {}_1C_{ijkl} {}^1\varepsilon_{kl} \quad (6.24)$$

$${}_1S_{ij} = {}_1C_{ijkl} {}^2_1\varepsilon_{kl} \quad (6.25)$$

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

$${}^1\varepsilon_{ij} = \frac{1}{2} \left(\frac{\partial_0^1 u_i}{\partial^1 x_j} + \frac{\partial_0^1 u_j}{\partial^1 x_i} - \frac{\partial_0^1 u_k}{\partial^1 x_i} \frac{\partial_0^1 u_k}{\partial^1 x_j} \right) \quad (6.26)$$

Substituting Equations (6.19) and (6.23) into Equation (6.14)

$$\begin{aligned} 0 &= \int_{{}^1\Omega} {}^2S_{ij} \delta({}_1^2\varepsilon_{ij}) d^1\Omega - \delta({}_1^2R) \\ &= \int_{{}^1\Omega} ({}^1\sigma_{ij} + {}^1S_{ij}) \delta({}_1^2\varepsilon_{ij}) d^1\Omega - \delta({}_1^2R) \\ &= \int_{{}^1\Omega} \{ {}^1S_{ij} \delta({}_1^2\varepsilon_{ij}) + {}^1\sigma_{ij} [\delta({}_1e_{ij}) + \delta({}_1\eta_{ij})] \} d^1\Omega - \delta({}_1^2R) \\ &= \int_{{}^1\Omega} {}^1S_{ij} \delta({}_1^2\varepsilon_{ij}) d^1\Omega + \int_{{}^1\Omega} {}^1\sigma_{ij} \delta({}_1\eta_{ij}) d^1\Omega + \delta({}_1^1R) - \delta({}_1^2R) \end{aligned} \quad (6.27)$$

where

$$\delta({}_1^1R) = \int_{{}^1\Omega} {}^1\sigma_{ij} \delta({}_1e_{ij}) d^1\Omega \quad (6.28)$$

is the virtual internal energy stored in the body at configuration C_1 .

Assuming the displacements u_i are small, the *updated Kirchhoff stress increment tensor* can be expressed as

$${}_1S_{ij} = {}_1C_{ijkl} {}_1^2\varepsilon_{kl} \approx {}_1C_{ijkl} e_{kl} \quad \delta({}_1^2\varepsilon_{ij}) \approx \delta({}_1e_{ij}) \quad (6.29)$$

${}_1C_{ijkl}$ denote the incremental constitutive tensors with respect to the configuration C_1 .

Substituting Equation (6.29) into Equation (6.27), the following equation can be obtained

$$\int_{{}^1\Omega} {}_1C_{ijkl} e_{kl} \delta({}_1e_{ij}) d^1\Omega + \int_{{}^1\Omega} {}^1\sigma_{ij} \delta({}_1\eta_{ij}) d^1\Omega = \delta({}_1^2R) - \int_{{}^1\Omega} {}^1\sigma_{ij} \delta({}_1e_{ij}) d^1\Omega \quad (6.30)$$

The total Cauchy stress components ${}^1\sigma_{ij}$ are evaluated using the constitutive relation.

6.3.4. FINITE ELEMENT MODEL FOR TWO-DIMENSIONAL CONTINUA

For simplicity, the following notations are introduced in the following section:

$$^1x_1 = x, \quad ^1x_2 = y, \quad {}^1_0u_1 = u, \quad {}^1_0u_2 = v, \quad u_1 = \bar{u}, \quad u_2 = \bar{v}$$

the first expression of Equation (6.30) can be written in the alternate form

$$\begin{aligned} \int_{^1\Omega} {}^1C_{ijkl} e_{kl} \delta({}^1e_{ij}) d^1\Omega &= \int_{^1\Omega} \delta\{{}_1\mathbf{e}\}^T [{}_1\mathbf{C}] \{{}_1\mathbf{e}\} d^1\Omega \\ &= \int_{^1\Omega} \{\delta\bar{\mathbf{u}}\}^T [\mathbf{D}]^T [{}_1\mathbf{C}] [\mathbf{D}] \{\bar{\mathbf{u}}\} d^1\Omega \end{aligned} \quad (6.31)$$

where,

$$\{{}_1\mathbf{e}\} = \begin{Bmatrix} {}^1e_{xx} \\ {}^1e_{yy} \\ 2{}^1e_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial \bar{u}}{\partial x} \\ \frac{\partial \bar{v}}{\partial y} \\ \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} \bar{u} \\ \bar{v} \end{Bmatrix} \equiv [\mathbf{D}] \{\bar{\mathbf{u}}\} \quad (6.32)$$

where

$$\mathbf{D} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \quad (6.33)$$

)

$$\{\delta {}_1\mathbf{e}\} = \begin{Bmatrix} \delta {}^1e_{xx} \\ \delta {}^1e_{yy} \\ 2\delta {}^1e_{xy} \end{Bmatrix} = [\mathbf{D}] \{\delta \bar{\mathbf{u}}\} \quad (6.34)$$

The second expression of Equation (6.30) can be written as

$$\int_{^1\Omega} {}^1\sigma_{ij} \delta({}^1\eta_{ij}) d^1\Omega = \int_{^1\Omega} \{\delta {}_1\boldsymbol{\eta}\}^T \{{}_1\boldsymbol{\sigma}\} d^1\Omega \quad (6.35)$$

$${}^1\boldsymbol{\eta} = \begin{Bmatrix} {}^1\eta_{xx} \\ {}^1\eta_{yy} \\ 2 {}^1\eta_{xy} \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} \frac{\partial \bar{u}}{\partial x} \frac{\partial}{\partial x} & \frac{\partial \bar{v}}{\partial x} \frac{\partial}{\partial x} \\ \frac{\partial \bar{u}}{\partial y} \frac{\partial}{\partial x} & \frac{\partial \bar{v}}{\partial y} \frac{\partial}{\partial x} \\ \frac{\partial \bar{u}}{\partial y} \frac{\partial}{\partial y} & \frac{\partial \bar{v}}{\partial y} \frac{\partial}{\partial y} \\ \frac{\partial \bar{u}}{\partial y} \frac{\partial}{\partial x} + \frac{\partial \bar{u}}{\partial x} \frac{\partial}{\partial y} & \frac{\partial \bar{v}}{\partial y} \frac{\partial}{\partial x} + \frac{\partial \bar{v}}{\partial x} \frac{\partial}{\partial y} \end{bmatrix} \begin{Bmatrix} \bar{u} \\ \bar{v} \end{Bmatrix} = \frac{1}{2} [\mathbf{D}_u] \bar{\mathbf{u}} \quad (6.36)$$

$$\{{}^1\boldsymbol{\sigma}\} = \begin{Bmatrix} {}^1\sigma_{xx} \\ {}^1\sigma_{yy} \\ {}^1\sigma_{xy} \end{Bmatrix} = \begin{bmatrix} {}^1C_{11} & {}^1C_{12} & 0 \\ {}^1C_{21} & {}^1C_{22} & 0 \\ 0 & 0 & {}^1C_{66} \end{bmatrix} \begin{Bmatrix} {}^1\varepsilon_{xx} \\ {}^1\varepsilon_{yy} \\ 2 {}^1\varepsilon_{xy} \end{Bmatrix} \quad (6.37)$$

$$\{{}_1\boldsymbol{\varepsilon}\} = \begin{Bmatrix} {}^1\varepsilon_{xx} \\ {}^1\varepsilon_{yy} \\ 2 {}^1\varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} - \frac{1}{2} \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial x} \right)^2 \right] \\ \frac{\partial v}{\partial y} - \frac{1}{2} \left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 \right] \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} - \left(\frac{\partial u}{\partial x} \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \frac{\partial v}{\partial y} \right) \end{Bmatrix} \quad (6.38)$$

However, the expression (6.35) is not convenient for the finite element development as the vector $\delta^1 \boldsymbol{\eta}$ is a nonlinear function of the vector of displacement increments $\bar{\mathbf{u}}$. It is written in an alternate form to facilitate the construction of finite element matrices.

$$\begin{aligned} \int_{\Omega} \{\delta^1 \boldsymbol{\eta}\}^T \{{}^1\boldsymbol{\sigma}\} d^1\Omega &= \int_{\Omega} \begin{Bmatrix} \frac{\partial \delta \bar{u}}{\partial x} \\ \frac{\partial \delta \bar{u}}{\partial y} \\ \frac{\partial \delta \bar{v}}{\partial x} \\ \frac{\partial \delta \bar{v}}{\partial y} \end{Bmatrix}^T \begin{bmatrix} {}^1\sigma_{xx} & {}^1\sigma_{xy} & 0 & 0 \\ {}^1\sigma_{xy} & {}^1\sigma_{yy} & 0 & 0 \\ 0 & 0 & {}^1\sigma_{xx} & {}^1\sigma_{xy} \\ 0 & 0 & {}^1\sigma_{xy} & {}^1\sigma_{yy} \end{bmatrix} \begin{Bmatrix} \frac{\partial \bar{u}}{\partial x} \\ \frac{\partial \bar{u}}{\partial y} \\ \frac{\partial \bar{v}}{\partial x} \\ \frac{\partial \bar{v}}{\partial y} \end{Bmatrix} d^1\Omega \\ &= \int_{\Omega} \{\delta^1 \bar{\mathbf{u}}\}^T [\bar{\mathbf{D}}]^T [{}^1\boldsymbol{\sigma}] [\bar{\mathbf{D}}] \{\bar{\mathbf{u}}\} d^1\Omega \end{aligned} \quad (6.39)$$

where

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

$$\bar{\mathbf{D}} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ \frac{\partial}{\partial y} & 0 \\ 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial y} \end{bmatrix} \quad (6.40)$$

$$[{}^1\boldsymbol{\sigma}] = \begin{bmatrix} {}^1\sigma_{xx} & {}^1\sigma_{xy} & 0 & 0 \\ {}^1\sigma_{xy} & {}^1\sigma_{yy} & 0 & 0 \\ 0 & 0 & {}^1\sigma_{xx} & {}^1\sigma_{xy} \\ 0 & 0 & {}^1\sigma_{xy} & {}^1\sigma_{yy} \end{bmatrix} \quad (6.41)$$

$$\int_{\Omega} {}^1\sigma_{ij} \delta({}_1e_{ij}) d\Omega = \int_{\Omega} \{{}^1\boldsymbol{\sigma}\}^T \delta({}_1\mathbf{e}) d\Omega = \int_{\Omega} \{{}^1\boldsymbol{\sigma}\}^T [\mathbf{D}] \{\delta\bar{\mathbf{u}}\} d\Omega \quad (6.42)$$

Using the FE-LSPIM shape function $\Psi(\mathbf{x})$ defined in Chapter 4, the following functions can be obtained

$$\{\bar{\mathbf{u}}\} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{Bmatrix} \sum_{j=1}^M \bar{u}_j \Psi_j \\ \sum_{j=1}^M \bar{v}_j \Psi_j \end{Bmatrix} = [\mathbf{M}] \{\bar{\boldsymbol{\Delta}}\} \quad (6.43)$$

where

$$\mathbf{M} = \begin{bmatrix} \Psi_1 & 0 & \Psi_2 & 0 & \cdots & \Psi_M & 0 \\ 0 & \Psi_1 & 0 & \Psi_2 & \cdots & 0 & \Psi_M \end{bmatrix} \quad (6.44)$$

M is the total number of nodes in the support of an element. The definition of the FE-LSPIM shape function Ψ_j is as in Chapter 4 Equation (4.15).

$$\{\bar{\boldsymbol{\Delta}}\}^T = \{\bar{u}_1 \quad \bar{v}_1 \quad \bar{u}_1 \quad \bar{v}_1 \quad \cdots \quad \bar{u}_1 \quad \bar{v}_1\} \quad (6.45)$$

Substituting Equations (6.31), (6.40), (6.42), (6.43) into Equation (6.30)

$$\begin{aligned} & \{\delta \bar{\Delta}\}^T \left(\int_{\Gamma_{\Omega}} [\mathbf{B}_L]^T \mathbf{C} [\mathbf{B}_L] d^1 \Omega + \int_{\Gamma_{\Omega}} [\mathbf{B}_{NL}]^T [\mathbf{1} \boldsymbol{\sigma}] [\mathbf{B}_{NL}] d^1 \Omega \right) \{\bar{\Delta}\} \\ &= \{\delta \bar{\Delta}\}^T \left(\int_{\Gamma_{\Omega}} [\mathbf{B}_L]^T \{\mathbf{1} \boldsymbol{\sigma}\} d^1 \Omega + \int_{\Gamma_{\Omega}} [\mathbf{M}]^T \mathbf{f} d^1 \Omega + \int_{\Gamma_{\Gamma}} [\mathbf{M}]^T \mathbf{t} d^1 \Gamma \right) \end{aligned} \quad (6.46)$$

where

$$[\mathbf{B}_L] = [\mathbf{D}]^e [\boldsymbol{\Psi}] = \begin{bmatrix} \frac{\partial \Psi_1}{\partial x} & 0 & \frac{\partial \Psi_2}{\partial x} & 0 & \dots & \frac{\partial \Psi_M}{\partial x} & 0 \\ 0 & \frac{\partial \Psi_1}{\partial y} & 0 & \frac{\partial \Psi_2}{\partial y} & \dots & 0 & \frac{\partial \Psi_M}{\partial y} \\ \frac{\partial \Psi_1}{\partial y} & \frac{\partial \Psi_1}{\partial x} & \frac{\partial \Psi_2}{\partial y} & \frac{\partial \Psi_2}{\partial x} & \dots & \frac{\partial \Psi_M}{\partial y} & \frac{\partial \Psi_M}{\partial x} \end{bmatrix} \quad (6.47)$$

$$[\mathbf{B}_{NL}] = [\bar{\mathbf{D}}]^e [\boldsymbol{\Psi}] = \begin{bmatrix} \frac{\partial \Psi_1}{\partial x} & 0 & \frac{\partial \Psi_2}{\partial x} & 0 & \dots & \frac{\partial \Psi_M}{\partial x} & 0 \\ \frac{\partial \Psi_1}{\partial y} & 0 & \frac{\partial \Psi_2}{\partial y} & 0 & \dots & \frac{\partial \Psi_M}{\partial y} & 0 \\ 0 & \frac{\partial \Psi_1}{\partial x} & 0 & \frac{\partial \Psi_2}{\partial x} & \dots & 0 & \frac{\partial \Psi_M}{\partial x} \\ 0 & \frac{\partial \Psi_1}{\partial y} & 0 & \frac{\partial \Psi_2}{\partial y} & \dots & 0 & \frac{\partial \Psi_M}{\partial y} \end{bmatrix} \quad (6.48)$$

From Equation (6.46), the finite element model based on the updated Lagrangian formulation can be simply written as

$$([\mathbf{K}_L] + [\mathbf{K}_{NL}]) \{\bar{\Delta}\} = \mathbf{F} - \mathbf{F}_1 \quad (6.49)$$

where

$$[\mathbf{K}_L] = \int_{\Gamma_{\Omega}} [\mathbf{B}_L]^T [\mathbf{C}] [\mathbf{B}_L] d^1 \Omega \quad (6.50)$$

$$[\mathbf{K}_{NL}] = \int_{\Gamma_{\Omega}} [\mathbf{B}_{NL}]^T [\mathbf{1} \boldsymbol{\sigma}] [\mathbf{B}_{NL}] d^1 \Omega \quad (6.51)$$

$$\{\mathbf{F}_1\} = \int_{\Gamma_{\Omega}} [\mathbf{B}_L]^T \{\mathbf{1} \boldsymbol{\sigma}\} d^1 \Omega \quad (6.52)$$

$$\{\mathbf{F}_1^2\} = \int_{\Omega} [\mathbf{M}]^T \{\mathbf{f}_1^2\} d\Omega + \int_{\Gamma} [\mathbf{M}]^T \{\mathbf{t}_1^2\} d\Gamma \quad (6.53)$$

6.3.5. FLOWCHART OF THE COMPUTER IMPLEMENTATION

The pseudo-code for the formulation of the element stiffness matrix is given as follows:

Loop over nodes in the mesh:

For each node, determine the number of nodes in its support, N , and save this information.

End loop over nodes

Do While $\|\mathbf{F}_1 - \mathbf{F}_1^1\| / \|\mathbf{F}_1^1\| > tolerance$

Loop over the elements in the mesh:

Loop over quadrature points in each element:

Determine the number of nodes, M , in the support of the element.

Loop over the four nodes of the element:

Compute Ψ as outlined in Chapter 4.

End loop over nodes of the elements.

Construct $\mathbf{B}_L$ and matrix $\mathbf{B}_{NL}$ as in Equations (6.47) and (6.48).

Compute the $\mathbf{K}_L$ as in Equation (6.50).

Read the values of the current stresses and form the $\{\mathbf{\sigma}_1^1\}$ and $[\mathbf{\sigma}_1^1]$ according to Equations (6.37) and (6.41).

Compute the $\mathbf{K}_{NL}$ as in Equation (6.51).

Compute $\mathbf{F}_1^1$ and $\mathbf{F}_1^2$ as in Equations (6.52) and (6.53).

End quadrature point loop.

Assemble element stiffness matrix into the global stiffness matrix.

Assemble element force vector into the global force vector.

End element loop.

Solve the stiffness equations for displacements/displacement parameters.

Update geometry $\mathbf{x} = \mathbf{x} + \mathbf{u}$.

End Do

6.4. NUMERICAL EXAMPLES

To investigate the performance of the FE-LSPIM QUAD4 element, several numerical examples of geometric nonlinear problems are examined. The results are compared with those obtained from ANSYS. A L_2 norm of residual forces less than 0.001 is used as the convergence criterion.

6.4.1. CANTILEVER BEAM UNDER SHEAR FORCE

A beam subjected to shear force is considered first (Fig. 6.2). The beam is of length $L = 10m$ and height $D = 1m$ and unit thickness. The beam is modeled using 2×8 regular and distorted meshes. The material properties are $E = 1.2 \times 10^4$ and $\nu = 0.2$. The beam is assumed to be in the plane state of stress. For numerical integration of stiffness, a 2×2 Gaussian scheme is used. Table 6.1 compares the deflection at point A obtained using the present element with those obtained using two well-known elements, viz., the isoparametric quadrilateral element called the QUAD4 (alias Q4) element and the QM6 element of Taylor et al. [18]. QUAD4 is a conforming element while QM6 is a non-

conforming element with two additional shape functions. Figure 6.3 shows a graphical comparison of the results. Table 6.2 and Fig. 6.4 show the corresponding results obtained using a distorted mesh. The results in Table 6.2 and Fig. 6.4 show that, for a regular mesh, the results of present element agree very well with that of QM6 element and are better than that of QUAD4 element. However, it is seen from Table 6.2 and Fig. 6.4 that the results given by the present element are almost unaffected by mesh distortion while the results obtained from QUAD4 as well as QM6 element show a high sensitivity to the mesh distortion. Thus, the present element has an excellent tolerance to mesh distortion.

Figure 6.2. Cantilever beam under shear force (a) Geometry; (b) Regular mesh, (c) Distorted mesh.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.1. Deflection at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 (For regular mesh).

Load P	QM6	QUAD4	FE-LSPIM QUAD4 (Present)
0	0	0	0
5	1.6202	0.9970	1.5748
10	3.0261	1.9377	2.9753
15	4.1293	2.7850	4.0980
20	4.9677	3.5255	4.9507
25	5.5991	4.1648	5.5977
30	6.0785	4.6991	6.0952
35	6.4524	5.1523	6.4850
40	6.7494	5.5173	6.7964
45	6.9679	5.8396	7.0499

Figure 6.3. Load versus deflection curve for point A the beam subjected to a shear force at the end as shown in Fig. 6.2 under regular mesh.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.2. Deflection (m) at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 (For distorted mesh).

Load P (KN)	QM6	QUAD4	FE-LSPIM QUAD4 (Present)	Reference solution*
0	0.0000	0.0000	0.0000	0.0000
5	1.3885	0.7597	1.5807	1.6202
10	2.6324	1.4935	2.9864	3.0261
15	3.6622	2.1830	4.1017	4.1293
20	4.4797	2.8159	4.9502	4.9677
25	5.1156	3.3863	5.5947	5.5991
30	5.6254	3.8991	6.0897	6.0785
35	6.0338	4.3480	6.4764	6.4524
40	6.3657	4.7478	6.7842	6.7494
45	6.6395	5.1011	7.0335	6.9679

Reference solution* given by QM6 under regular mesh.

Figure 6.4. Load versus deflection curve at point A of the beam subjected to a shear force at the end as shown in Fig. 6.2 under distorted mesh.

In order to study the convergence rate of solution for the cantilever shear force problems of Fig. 6.2, the beam is analyzed using a series of meshes. In this study, the length of the beam is taken as 1000 m and the height of the beam as 39.37 m . The applied shear force is 1000 N and the material properties are taken as: $E = 1.8 \times 10^5$, $\nu = 0.3$. The results are given in Table 6.3 and plotted in Fig. 6.5. It is seen from Fig. 6.5 that the present element shows a faster convergence than QM6 and QUAD4.

Figure 6.5. Convergence of vertical deflection for the cantilever shear force problems (Fig. 6.2).

Table 6.3. Deflection of the cantilever beam under shear force at the end.

Mesh	QM6	QUAD4	FE-LSPIM QUAD4 (Present)
2×4	-313.52	22.031	-245.11
2×8	-322.53	-73.925	-309.76
2×16	-324.07	-177.27	-322.08
2×32	-324.44	-267.69	-324.53
2×64	-324.53	-303.69	-324.64
Overkill solution*		-324.61	

*Overkill solution is obtained from ANSYS (PLANE 82) using a very fine (8×128) mesh

6.4.2. CLAMPED BEAM UNDER CENTRAL POINT FORCE

Figure 6.6. Clamped beam under central point force.

A beam with two-fixed ends as shown in Fig. 6.6 is considered. The length of the beam is $L = 200$ mm and depth is $D = 10$ mm. The material properties used are: $E = 206.85$ GPa, $\nu = 0.3$. The beam is modelled with 2×20 regular as well as distorted elements, as shown in Fig. 6.6. For numerical integration of stiffness, 2×2 Gaussian quadrature is used. The computed results of vertical displacement at point A are listed in Table 6.4 and Table 6.5 for a series of load values. The load deflection curves for point A are plotted in Figs. 6.7 and 6.8. It is seen that the results obtained using FE-LSPIM QUAD4 are in a good agreement that of QM6 and better than QUAD4.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.4. The computed results of vertical displacement (mm) at point A of the beam subjected to central point force as shown in Fig. 6.6 (For regular mesh).

Load P	QM6	QUAD4	FE-LSPIM QUAD4 (Present)
6000	9.338	8.072	9.506
9000	11.707	10.567	11.904
12000	13.570	12.417	13.793
15000	15.133	14.029	15.379
20000	17.319	16.278	17.603
23000	18.459	17.448	18.764
25000	19.165	18.172	19.485
30000	20.784	19.826	21.139
35000	22.239	21.308	22.629

Figure 6.7. The load versus deflection curve at point A of the beam subjected to central point force as shown in Fig. 6.6 (For regular mesh).

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.5. The computed results of vertical displacement (mm) at point A of the beam subjected to central point force as shown in Fig. 6.6 (For distorted mesh).

Load P	QM6	QUAD4	FE-LSPIM QUAD4 (Present)	Reference solution*
3000	5.672	4.314	6.050	5.930
6000	9.084	7.547	9.497	9.338
9000	11.472	9.970	11.890	11.707
12000	13.351	11.908	13.780	13.570
15000	14.925	13.537	15.364	15.133
20000	17.126	15.809	17.584	17.319
23000	18.272	16.989	18.740	18.459
25000	18.981	17.718	19.464	19.165
30000	20.607	19.383	20.784	20.784
35000	22.065	20.873	22.239	22.239

Reference Solution* obtained with ANSYS Plane 42 element with bubble functions under regular mesh

Figure 6.8. The load versus deflection curve at point A of the beam subjected to central point force as shown in Fig. 6.6 (For distorted mesh).

6.4.3. A COLUMN UNDER AN ECCENTRIC AXIAL COMPRESSIVE LOAD

Figure 6.9. A column under an eccentric compressive load: (a) geometry; (b) regular mesh; (c) distorted mesh.

A column with an eccentric compressive load is shown in Fig.6.9. The column is modelled with 2×10 elements using regular as well as distorted meshes as shown in Figs. 6.9(b) and Fig. 6.9(c), respectively. The numerical values for the material properties are: Young's modulus, $E = 1000000$ and Poisson's ratio, $\nu = 0$. The height of the beam $L = 10$ m and depth $H = 1$ m and thickness is unit value. The problem is solved in plane-stress condition for the horizontal and vertical tip displacements at point A under regular as well as distorted meshes. The results are summarised in Tables 6.6-6.9. The loads versus displacement response at point A are plotted in Figs. 6.10-6.13. The displacements given by the present element agree well with that obtained using QM6 element when regular mesh is used. When the distorted mesh is used, the results

obtained with the present element are better than that of QM6 element. This example again suggests the high distortion tolerance of the present element.

Table 6.6. The horizontal displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).

P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)
0	0	0	0
400	0.0922	0.1495	0.1494
800	0.2169	0.3939	0.3926
1000	0.2972	0.5805	0.5806
1200	0.3943	0.8544	0.8475
1400	0.5136	1.2628	1.2469
1600	0.6635	1.8891	1.8779
1800	0.8561	2.9662	2.9087
2000	1.0888	4.2588	4.2672
2200	1.4509	5.3833	5.452
2400	1.9078	6.1777	6.2763
2600	2.4946	6.7107	6.8066
2800	3.1943	7.0419	7.1383

Figure 6.10. Load versus horizontal displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.7. The vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).

P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)
0	0.0000	0.0000	0.0000
400	0.0137	0.0201	0.0200
800	0.0319	0.0549	0.0546
1000	0.0439	0.0848	0.0846
1200	0.0588	0.1352	0.1335
1400	0.0782	0.2268	0.2224
1600	0.1043	0.4092	0.4048
1800	0.1413	0.8558	0.8256
2000	0.1917	1.6629	1.6650
2200	0.2830	2.7056	2.7743
2400	0.4228	3.7598	3.8933
2600	0.6458	4.7461	4.9104
2800	0.9821	5.6098	5.8219

Figure 6.11. Load versus vertical displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For regular mesh).

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.8. The horizontal displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).

P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)	Reference solution*
0	0.0000	0.0000	0.0000	0.0000
400	0.0416	0.1394	0.1490	0.1495
800	0.0894	0.3596	0.3902	0.3939
1000	0.1160	0.5231	0.5747	0.5805
1200	0.1446	0.7429	0.8347	0.8544
1400	0.1756	1.0618	1.2206	1.2628
1600	0.2092	1.5275	1.8411	1.8891
1800	0.2458	2.4119	2.9007	2.9662
2000	0.2857	3.4648	4.2673	4.2588
2200	0.3294	4.6149	5.4543	5.3833
2400	0.3774	5.6128	6.2782	6.1777
2600	0.4304	6.2597	6.8084	6.7107
2800	0.4891	6.6989	7.1470	7.0419

*Solution given by QM6 under regular mesh.

Figure 6.12. Load versus horizontal displacement curve at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.9. The vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).

P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)	Reference solution*
0	0.0000	0.0000	0.0000	0.0000
400	0.0085	0.0191	0.0200	0.0201
800	0.0178	0.0508	0.0544	0.0549
1000	0.0228	0.0769	0.0839	0.0848
1200	0.0281	0.1162	0.1317	0.1352
1400	0.0337	0.1833	0.2171	0.2268
1600	0.0397	0.3044	0.3943	0.4092
1800	0.0462	0.6180	0.8226	0.8558
2000	0.0532	1.1566	1.6646	1.6629
2200	0.0609	2.0088	2.7695	2.7056
2400	0.0694	3.0856	3.8787	3.7598
2600	0.0788	4.0819	4.8708	4.7461
2800	0.0893	5.0059	5.7547	5.6098

*Solution given by QM6 under regular mesh.

Figure 6.13. Load versus vertical displacement at point A of the column under an eccentric compressive load as shown in Fig. 6.9 (For distorted mesh).

6.4.4. THRUST BEAM UNDER CENTRAL LOAD

Figure 6.14. Beam subject to central transverse load.

△ Restraint in y direction

Next a beam subject to a central transverse load as shown in Fig. 6.14 is considered. Young's modulus and Poisson's ratio are taken as 10 and 0, respectively. For numerical integration of stiffness, a 2×2 Gaussian scheme is used. The computed results of vertical displacement at point A are listed in Table 6.10 and Table 6.11 for regular and distorted meshes, respectively, and the corresponding load deflection curve for point A are presented in Fig. 6.15 and Fig. 6.16. It is seen that the results obtained using the present element agree well with that obtained using QM6 element, and are better than that obtained with QUAD4 element. However, the results obtained using the present elements are better than that given by both QUAD4 and QM6 under distorted mesh.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.10. The vertical displacement of point A as shown in Fig. (6.14) (For regular mesh).

Load P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)
0	0	0	0
1	9.277	10.19	10.27
2	16.056	17.373	17.59
3	21.557	23.183	23.18
4	25.903	27.630	27.63

Table 6.11. The vertical displacement of point A as shown in Fig. (6.14) (For distorted mesh).

Load P	QUAD4	QM6	FE-LSPIM QUAD4 (Present)	Reference solution*
0	0	0	0	0
1	7.766	9.006	10.14	10.192
2	13.838	15.616	17.38	17.373
3	18.492	21.073	22.98	23.183
4	22.802	25.442	27.49	27.630

*Solution given by QM6 under regular mesh.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Figure 6.15. Load versus deflection curve for point A as shown in Fig. (6.14) under regular mesh.

Figure 6.16. Load versus deflection curve for point A as shown in Fig. (6.14) under distorted mesh.

6.4.5. COOK'S SKEW BEAM

The cook's skew beam problem (Fig. 4.19) proposed by Cook et al. [155] for linear analysis is studied here for geometrical nonlinear analysis. The problem is modelled here using 8×4 mesh. The computed vertical displacements at point A for typical load values are listed in Table 6.12, and the corresponding load deflection curve is presented in Fig. 6.17. It is seen that the results obtained using the present element are rather comparable to that obtained using QM6 element (available in ANSYS), and are better than that obtained with QUAD4 element. In all the problems tested in section 6.4.1 to 6.4.4, the performance of the present element has been comparable to QM6 element for regular mesh or better than QM6 element for distorted mesh. However, for this problem, the performance of the present element is worse than that of QM6 element. The reason seems to be that the present implementation of the updated Lagrangian formulation works well for small strain problems (*i.e.*, the thin beam problems discussed in section 6.4.1-6.4.4) and not very much so for large strain problem (*i.e.*, the present Cook's skew beam, which is actually a thick beam problem). In order to verify this conjecture, a modification of Cook's skew beam problem is now considered. The length dimensions are multiplied by a factor of 10 to change it to a thin beam problem, and the problem is solved again. The results are summarized in Table 6.13, which show that the results of the present element and that of QM6 element are almost the same. This confirms that for small strain problems the present formulation works well and not very much for large strain problems.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Table 6.12. The vertical displacement at point A of the Cook's skew beam as shown in Fig. 4.19.

Load P	Q4	QM6	FE-LSPIM QUAD4	Reference*
0.0	0.000	0.000	0.000	0.000
0.2	3.986	4.423	4.448	4.457
0.4	7.461	8.211	8.315	8.265
0.6	10.474	11.428	11.645	11.495
0.8	13.093	14.170	14.515	14.245
1.0	15.388	16.532	17.008	16.616
1.1	16.434	17.598	18.136	17.692
1.2	17.422	18.598	19.195	18.692
1.3	18.358	19.542	20.194	19.662
1.4	19.248	20.435	21.140	20.554

Reference* is obtained by QM6 using a finer mesh (32×64 element mesh)

Figure 6.17. Load versus deflection curve at point A of the Cook's skew beam as shown in Fig. 4.19.

Table 6.13. The vertical displacement at point A of the elongated Cook's skew beam

P (N)	QUAD 4	QM6 (m)	Present (m)
0.001	9.6057	10.508	10.510
0.002	19.129	20.912	20.914
0.003	28.538	31.169	31.173
0.005	46.903	51.097	51.112
0.01	89.201	96.301	96.360
0.012	104.41	112.26	112.300

6.4.6. DISTORTION SENSITIVITY TEST

The cantilever beam in Fig. 6.2(a) used earlier for the convergence study is considered here for the distortion sensitivity test. The distorted patch as shown in Fig. 6.18 is used for the purpose. The distortion parameter δ is used to control the extent of distortion. The deflection at point A has been computed using the present element as well as, QUAD4 and QM6 for different values of δ . A comparison of computed deflections is shown in Table 6.14. A comparison of percent error in the computed deflections is shown in Fig. 6.19. The results reveal that the present element is far less sensitive to mesh distortion compared to QUAD4 and QM6.

CHAPTER 6. FE-LSPIM QUAD4 ELEMENT FOR GEOMETRIC NONLINEAR ANALYSIS

Figure 6.18. A patch of elements with geometric distortions.

Figure 6.19. The present error of deflection of point A for distortion sensitivity test.

Table 6.14. Comparison of deflection of point A for a cantilever beam with distorted mesh.

	0	$\delta = 10$	$\delta = 20$	$\delta = 30$
QUAD4	-212.34	-135.71	-104.95	-90.659
QM6	-324.19	-191.88	-134.87	-121.64
FE-LSPIM QUAD4 (Present)	-322.5	-322.2	-295.07	-248.95
Overkill solution*		-324.61		

6.5. CONCLUDING REMARKS

In this chapter, the effectiveness FE-LSPIM QUAD4 element has been explored for geometric nonlinear applications. Updated Lagrangian formulation has been used for the purpose. The accuracy of the element has been investigated by comparing the performance of the present element with that of QUAD4 and QM6 elements for typical numerical examples. The results show that, for regular meshes, the performance of the present element is comparable to QM6 element. For distorted meshes, the element yields more efficient results compared to QUAD4 as well as QM6 elements. Out of all the three elements tested, the present element has the best tolerance to mesh distortion.

CHAPTER 7. CRITICAL ASSESSMENT OF FE-LSPIM QUAD4 ELEMENT

In Chapters 4-6, the FE-LSPIM QUAD4 element has been developed and applied to static and free vibration analyses of linear elastic problems and also to geometrical nonlinear problems. The element has been observed to perform better than the classical elements. While the element has advantages, it does have a few disadvantages too. A critical review of advantages and disadvantages is presented in this chapter. Discussion is limited to some important aspects such as mesh-distortion tolerance, linear independence problem, singularity problem, locking problem, computational efficiency and relative merits of FE-LSPIM QUAD4 compared to classical FEM and meshfree methods.

7.1. MESH DISTORTION TOLERANCE

High mesh distortion tolerance is an attractive feature of FE-LSPIM QUAD4. The reason why this element has better mesh distortion tolerance can be traced back to the fact that this element has been designed to satisfy two important properties:

1. Mesh-distortion independent compatibility property.
2. Mesh-distortion independent higher order completeness (reproducibility) property.

Property 1 refers to the fact that the interpolation is continuous across the element boundaries irrespective of the geometry of the adjoining elements. This has been made

possible by the fact that the QUAD4 isoparametric shape functions are used for partition of unity. The proof for this property has already been given in section 4.2.4.3.

Property 2 refers to the fact that the completeness of higher order monomial terms is assured irrespective of the geometry of the elements in the mesh. This means not only the linear terms $\{1, x, y\}$ but also all the higher order terms $\{x^2, xy, y^2, x^2y, xy^2\}$ considered in the interpolation are producible. The proof for this property has already been given in section 4.2.4.4.

By virtue of this property, the element should be able to reproduce any field involving the monomial terms $\{1, x, y, x^2, xy, y^2, x^2y, xy^2\}$ irrespective of mesh distortion. Although the results reported in Chapters 4-6 generally conform to this expectation, some anomalous results have also been observed. One of them appears in the patch test results shown in Table 4.4. Although the problem involves just a quadratic displacement field, the u - displacement is not reproduced exactly; a small error (1.000-0.996) is observed. However for the regular mesh, this problem does not arise (Table 4.3). Presently, the reason for this is not quite obvious and need to be explored in future.

7.2 LINEAR INDENDENCE PROBLEM

A well known issue of the PU based methods is the so-called ‘linear dependence problem’ that leads to singular global stiffness matrices. This problem occurs when both the PU functions and the local functions are taken as explicit polynomials [135,136]. In order to eliminate the linear dependence problem, Oden et al. [142] suggested excluding

the space spanned by the terms of finite element shape functions from that of the local approximation functions. The mapped quadrilateral shape functions were used in the work of Strouboulis et al. [137, 138], but they could not eliminate the linear dependencies completely. Tian and Yagawa [139] proposed a 4-node quadratic tetrahedral element and 3-node quadratic triangular element. They point out the triangular meshes help to avoid the linear dependence problem.

Hence, so far, no successful attempts at eliminating the linear dependence problem have been made.

An investigation into whether FE-LSPIM QUAD4 has linear dependence problem was taken up. The eigenvalue analysis of FE-LSPIM QUAD4 shows that the nullity for the present element does not change with mesh geometry or mesh refinement. This is an important merit of the present element.

7.2.1 EIGENVALUE ANALYSIS

The linear dependence check is carried out here in two steps [139, 140]. Firstly, eigenvalue analysis of global stiffness is carried out before applying the boundary conditions in order to determine the number of zero eigenvalues. Next, the eigenvalue analysis is repeated after applying the minimum number of boundary conditions to restrain the rigid body modes. If there are no more zero eigenvalues after applying the boundary conditions, then it suggests that the method has no linear dependency problem.

For the meshes and boundary conditions shown in Fig. 7.1, element stiffness matrices are computed for plane stress condition with an Young's modulus of $E = 1.0$ and a Poisson's ratio of $\nu = 0.25$. The counts of computed zero eigenvalues are listed in Table 7.1. Before applying boundary condition, a total of 3 zero eigenvalues (corresponding to the three rigid body modes) are found for all the meshes. The number of zero eigenvalues is also found to be invariant with respect to the mesh geometry and mesh refinement. Moreover, by applying the boundary conditions, there are no more zero eigenvalues left. This suggests that the proposed FE-LSPIM QUAD4 element is free of linear dependence problem.

Table 7.1. Eigenvalue analysis of global stiffness matrix.

Mesh		Before Applying Boundary conditions	After Applying boundary conditions	After Applying boundary conditions	
		Nullity/total dofs	Nullity/total dofs	Max. eigenvalue	Min. eigenvalue
Mesh (1)	(a)	3/18	0/18	3.2584	0.0247
	(b)	3/18	0/18	4.2490	0.0240
	(c)	3/18	0/18	3.8010	0.0206
Mesh (2)	(a)	3/32	0/32	3.6324	0.0115
	(b)	3/32	0/32	6.5365	0.0161
	(c)	3/32	0/32	3.8001	0.0103
Mesh (3)	(a)	3/50	0/50	3.7200	0.0068
	(b)	3/50	0/50	4.6884	0.0069
	(c)	3/50	0/50	3.9537	0.0060

Nullity refers to the number of zero eigenvalues of the stiffness matrix.

Figure 7.1. Patches for linear dependence check.

7.3. SINGULARITY PROBLEM

The other issue that scores against FE-LSPIM QUAD4 is the possibility of singularity of $\mathbf{P}$ -matrix (section 4.2.2). When $\mathbf{P}$ -matrix is singular, the shape functions of FE-LSPIM QUAD4 do not exist. Singularities of $\mathbf{P}$ -matrix may result due to improper choice of monomial terms or nodal positions.

Liu and Gu [107] encountered similar singularity problems in connection with the development of their point interpolation method. For alleviating this problem, they propose some strategies such as perturbing the nodes by a small distance before computing $\mathbf{P}$ -matrix [107], using radial basis functions (RBF) [112] instead of polynomial basis functions, and using matrix triangularization algorithm (MTA) [113].

Nevertheless the above mentioned methods are only fixes rather than solutions to completely overcome the singularity problem. Better methods are yet to appear.

The singularity problem has appeared in the development of FE-LSPIM QUAD4 too. For most of the test problems considered in this thesis, an eight term polynomial has in general been used in the least-square fit procedure to construct the FE-LSPIM QUAD4 shape functions (section 4.2.1). When the number of nodes in the support is less than 8, this would lead to a singular $\mathbf{P}$ -matrix (as well as singular $\bar{\mathbf{P}}$ -matrix in the least-square fit procedure). Such difficulties have often been encountered with the boundary nodes as their support includes less number of nodes. To solve this problem, a second order nodal connectivity has been defined so as to include more number of nodes in the support. Agreeably, this is a cumbersome procedure as different support sizes are used for the nodes at the boundary and the nodes at the interior regions. An alternative is to use second order support uniformly for all the nodes. This will, of course, demand more computational effort. However, the benefit is that the book-keeping of boundary and interior nodes is avoided.

Even with the use the second order connectivity, it is sometimes possible that $\mathbf{P}$ -matrix still does not exist in some situations. For example, in Fig. 7.2(a), only six nodes appear in the domain of node 1 even with second order connectivity, and the six nodes are arranged in two lines parallel to the x-axis. Since there are only six nodes, eight term basis cannot be considered anymore. A six term basis (i.e., $1, x, y, xy, x^2, y^2$) may seem appropriate. However, as the nodal density has only two distinct y-coordinate values (Fig. 7.2(a)), the $\mathbf{P}$ -matrix will still be singular even with six term basis. To solve this

problem, small perturbations to nodal coordinates as done by Liu and Gu [119] can be used (shown exaggerated in Fig. 7.2(b)). A simple approach to circumvent these difficulties altogether is to use a higher nodal density, i.e., a fine mesh as shown in Fig. 7.2(c).

Figure 7.2. Nodal density.

The approaches to handle singularity problem discussed above are practical although not elegant. Better methods are yet to come.

Sometimes, the nodal locations may lead to a near singular $\mathbf{P}$ -matrix rather than singular matrix itself. Under such instances, the inverse of the $\mathbf{P}$ -matrix may exist but may be ill-conditioned. This will then result in bad results. Ooi [177] proposed a method to avoid this type of near singularity problem. In this approach, the nodal coordinates are rotated by some arbitrary angle; the stiffness is then recomputed based on the new coordinates. Upon computing the element stiffness matrix, the matrix is transformed back to its original configuration. The near singularity problem, however, has not been experienced in the computations reported in this thesis.

7.4 LOCKING PROBLEM

Locking is defined as a condition of excessive stiffness for a particular deformation state MacNeal [178]. The isoparametric QUAD4 element is a well-known example where shear locking is experienced.

The reason for locking can be understood easily by considering a rectangular shaped element. For a four-node rectangular element, the displacement field can be expressed as:

$$u = a_1 + a_2x + a_3y + a_4xy \quad (7.1)$$

$$v = b_1 + b_2x + b_3y + b_4xy \quad (7.2)$$

For the case of pure bending in xy plane, the shear strain $r_{xy} = u_{,y} + v_{,x} = 0$. This is only possible when

$$a_3 + b_3 = 0 \quad (7.3)$$

$$a_4 = 0 \quad (7.4)$$

$$b_4 = 0 \quad (7.5)$$

Equation (7.3) contains coefficients from both u and v displacement fields and hence can represent true constraint condition. However, Equations (7.4)-(7.5) involve isolated coefficients and hence lead to spurious constraints responsible for locking [179].

As known from the Literature Review chapter, some remedies to alleviate shear locking problem of QUAD4 include reduced integration of element stiffness matrix, addition of ‘bubble functions’ (Wilson’s incompatible modes) and drilling freedom to the displacement interpolation.

In order to study the shear locking phenomenon of FE-LSPIM QUAD4, the cantilever beam as shown in Fig. 7.3 is considered. Three aspect ratios, $L/h = 30, 300, \text{ and } 3000$, are considered. When computing, $M = h, F = 1$ is used. With the increase in aspect ratio, the elements become thinner in the depth directions and the slenderness of the elements became more and more severe. For results reported here, a 2×2 Gaussian quadrature is used. The results are summarized in Tables 7.2-7.3. The tables show that QUAD4 and QM6 tend to lock with the increase in aspect ratio under constant moment while FE-LSPIM QUAD4 does not. The small error in the results of FE-LSPIM QUAD4 is due to near singularity problem. For the beam subjected to tip shear force, all the elements show locking tendency. Although FE-LSPIM QUAD4 tends to lock, the displacement does not deteriorate much with the increase in aspect ratio.

Figure 7.3. Cantilever beam for the study of shear locking.

Table 7.2. Normalized tip deflection for the beam subjected to constant moment.

L/h	QUAD4	QM6	FE-LSPIM QUAD4	Exact
3	0.892	1	1.000	5.4×10^{-5}
30	0.094	0.994	1.006	0.0054
300	0.001	0.954	1.007	0.54
3000	0.000	0.857	1.003	540

Normalized with respect to exact solution shown on the test column.

Table 7.3. Normalized tip deflection for the beam subjected to shear force.

L/h	QUAD4	QM6	FE-LSPIM QUAD4	Exact
3	1.123	1.02	1.01	1.1625×10^{-4}
30	0.094	0.984	0.824	-0.1081
300	0.001	0.983	0.756	-108.0
3000	0.000	0.496	0.752	-108000

Normalized with respect to exact solution shown on the Exact column.

7.5 THE INF-SUP TEST

The *consistency* and *stability* are the two important requirements of a finite element formulation so as to assure convergence. The consistency is guaranteed by choosing continuous test functions and trial functions. The well known Babuska-Brezzi (or simply, the BB) condition [180,181] is accepted as a crucial requirement to ensure the stability of an effective finite element method. The BB condition is also named as the ‘inf-sup’ condition [182,183]. For practical purposes, checking the validity of the inf-sup stability condition, at a given refinement level, provides the necessary information to assess whether an element is stable and accurate. However, an analytical proof of whether the inf-sup condition is satisfied by a specific element is difficult, so the numerical test is valuable.

In the following discussion, the numerical inf-sup condition is presented in the general context of variational statement of elliptic problems in linear elasticity, viz.

$a(\mathbf{u}, \mathbf{w}) = \langle \mathbf{f}, \mathbf{w} \rangle$. For the bilinear form $a(\mathbf{u}, \mathbf{w}): U \times W \rightarrow R$, the inf-sup condition is given by

$$\inf_{\mathbf{u} \in U} \sup_{\mathbf{w} \in W} \frac{|a(\mathbf{u}, \mathbf{w})|}{\|\mathbf{u}\|_U \|\mathbf{w}\|_W} \geq \beta > 0 \quad (7.6)$$

where the constant β is the so-called BB inf-sup constant, which is independent of mesh refinement, and physical features such as boundary condition and loading. $\|\mathbf{u}\|_U$ and $\|\mathbf{w}\|_W$ are the norms defined on space U and W , respectively. The spaces U and W are defined as

$$U = \left\{ \mathbf{u} \mid \mathbf{u} \in L^2(\Omega), \frac{\partial u_i}{\partial x_j} \in L^2(\Omega), i, j = 1, 2, u_i|_{\Gamma_u} = 0, i = 1, 2 \right\} \quad (7.7)$$

$$W = \left\{ \mathbf{w} \mid \mathbf{w} \in L^2(\Omega), \frac{\partial w_i}{\partial x_j} \in L^2(\Omega), i, j = 1, 2, w_i|_{\Gamma_u} = 0, i = 1, 2 \right\} \quad (7.8)$$

where $L^2(\Omega)$ is the space of square integrable functions over the domain Ω and Γ_u is the essential boundary. In this thesis, the following semi-norm [182, 183] is used:

$$\|\mathbf{u}\|_U = \sqrt{\int_{\Omega} \left[\sum_{i=1}^2 \sum_{j=1}^2 \left(\frac{\partial u_i}{\partial x_j} \right)^2 \right] d\Omega} \quad (7.9)$$

$$\|\mathbf{w}\|_W = \sqrt{\int_{\Omega} \left[\sum_{i=1}^2 \sum_{j=1}^2 \left(\frac{\partial w_i}{\partial x_j} \right)^2 \right] d\Omega} \quad (7.10)$$

The finite element version of Equation (7.6) for a given mesh is written as.

$$\inf_{\mathbf{U} \in R^N} \sup_{\mathbf{W} \in R^N} \frac{\mathbf{U}^T \mathbf{K} \mathbf{U}}{\mathbf{U}^T \mathbf{S} \mathbf{U}} = \beta_h > \beta > 0 \quad (7.11)$$

where β_h , called the *inf-sup* value, is an estimate of β , $\mathbf{U}$ are the global nodal displacement vectors, N is the number global degrees of freedom, $\mathbf{K}$ is the global stiffness matrix and $\mathbf{S}$ are the symmetric positive-definite global norm matrices obtained by assembling their element counterparts, *viz.*

$$\mathbf{S} = (\mathbf{Q}\Psi)^T (\mathbf{Q}\Psi) \quad (7.12)$$

Ψ are shape function matrices defined in Chapter 4 and $\mathbf{Q}$ is a differential operator matrix arising from the definition of norm in Equation (7.9) given by

$$\mathbf{Q} = \begin{bmatrix} \partial/\partial x & \partial/\partial x & 0 & 0 \\ 0 & 0 & \partial/\partial x & \partial/\partial x \end{bmatrix}^T \quad (7.13)$$

It is hard to obtain the inf-sup constant β_h and check its convergence with mesh refinement. If β_h converges to a constant positive value, the element is said to have passed the numerical *inf-sup* test and suggesting that the formulation is possibly stable. But passing of numerical inf-sup tests cannot assure the stability of a given formulation.

It can be shown (see Reference [183]) that $\beta_h \equiv \sqrt{|\lambda_{\min}|}$ where $\lambda_{\min}$ is the smallest eigen-value of the eigenproblem

$$(\mathbf{K}^T \mathbf{S}^{-1} \mathbf{K})\varphi = \lambda \mathbf{S}\varphi \quad (7.14)$$

excluding the zero eigenvalues corresponding to the rigid modes of the finite element model. The inf-sup convergence of the present element has been studied for the following test problems. In the following section the symbol Δ means constraints in both the x and y direction and the symbol $\circ$ means constraints in y direction.

7.5.1. TEST PROBLEM 1

A square geometry with boundary conditions shown in Fig. 7.5 and Fig. 7.6 is considered first, the inf-sup values are computed for plane stress condition with an Young's modulus of $E = 2.1 \times 10^{11}$, Poisson's ratio $\nu = 0.3$ and 2×2 Gaussian

integration. The superelement consisting of four elements (see Fig. 7.4) is used for mesh refinement. A sequence of four meshes with 1, 2, 4, 8 super elements along the length as well as depth of the geometry is considered for the convergence study. Figures 7.7 and 7.8 show that the inf-sup value converges to a steady constant value for both distorted mesh as well as undistorted meshes under different boundary conditions.

(a) Regular mesh

(b) Distorted mesh

Figure. 7.4. Superelements used for test problem 1.

(a) Regular mesh

(b) Distorted mesh

Figure.7.5. Meshes used for boundary condition 1.

CHAPTER 7. CRITICAL ASSESSMENT OF FE-LSPIM QUAD4 ELEMENT

Figure. 7.6. Meshes used for boundary condition 2.

Figure.7.7. Convergence of inf-sup value with mesh refinement for the boundary condition shown in Fig. 7.5.

Figure.7.8 Convergence of inf-sup value with mesh refinement for the boundary condition shown in Fig.7.6.

7.5.2. TEST PROBLEM 2

This problem was proposed by Chen et al. [66] for a constant strain patch as shown in Figs. 7.9 and 7.10, for which, the Young's modulus $E = 1.0 \times 10^7$, Poisson's ratio $\nu = 1/3$ and the thickness of the plate $t = 1$. Here it is used for inf-sup test. The rectangular domain is divided into three elements as shown and then for subsequent mesh refinement, each element is divided through the midpoints of the element sides into 2,4,6,8 at every stage of mesh refinement. Figure 7.11 suggest that, for both boundary conditions, the inf-sup value converge to a steady value with the mesh refinement.

Figure. 7.9. Test problem 2 with boundary condition 1.

Figure. 7.10. Test problem 2 with boundary condition 2.

Figure. 7.11. Convergence of inf-sup value with mesh refinement for test problem 2.

7.5.3. TEST PROBLEM 3

This problem (Fig. 4.19) was proposed by Cook et al. [155]. The problem is modelled using 2×2 , 4×4 , 8×8 and 16×16 meshes. The Poisson's ratio $\nu = 1/3$ and Young's modulus $E = 1.0 \times 10^7$. The left side is fixed for boundary condition 1 and the left and right side are fixed for boundary condition 2. It is seen from Fig. 7.12 that the inf-sup value converged to a steady value under both boundary conditions.

Figure. 7.12. Convergence of inf-sup value with mesh refinement for test problem 3.

7.5.4. TEST PROBLEM 4

A thick curved cantilever beam is subjected to a transverse force at its tip as shown in Fig. 7.13. The geometry is meshed using 2, 4, 6, 8 elements along the depth of the beam and correspondingly double the number of the elements along the length. The Young's modulus $E = 1.0 \times 10^7$ and Poisson's ratio $\nu = 0.3$. The thickness of the beam is 1mm . Figure 7.14 shows that the inf-sup value does convergence with 3×3 as well as 2×2 Gaussian integration.

CHAPTER 7. CRITICAL ASSESSMENT OF FE-LSPIM QUAD4 ELEMENT

Figure 7.13. Bending of thick curved beam.

Figure. 7.14. Convergence of inf-sup value with mesh refinement for test problem 4.

7.6 COMPUTATIONAL EFFICIENCY

Table 7.4 lists the relative errors in the computed displacement as well as the computational times for different meshes for test problem (Fig. 3.9). A MATLAB program has been used for the purpose with a 2×2 Gaussian integration for computing the stiffness matrix. It is seen that the computational times of the present method are comparable to that of EFG while the accuracy of solution is better than that of EFG. For the computational time of the same order (see the bold-faced entries of Table 7.4), the present method gives a more efficient solution (0.067% error) than QUAD4 element (3.236% error) as well as the EFG method (0.281% error).

Table 7.4. Computational efficiency for the proposed element.

Mesh	QUAD4		EFG		FE-LSPIM QUAD4 (Present)	
	% Error	Time(s)	% Error	Time(s)	% Error	Time(s)
2×2	60.955	0.06	12.527	0.07	12.921	0.15
2×8	10.517	0.18	0.281	0.24	0.067	0.25
4×16	3.236	0.23	0.112	0.64	0.023	0.62
6×24	1.584	0.34	0.056	1.34	0.011	1.38
8×32	0.944	0.54	0.034	2.61	0.004	2.60

7.7. RELATIVE MERITS OF PRESENT METHOD (FE-LSPIM), FEM AND MESHFREE METHODS

Based on the observations in this thesis, the comparison of FE-LSPIM versus FEM can be summarised as in Table 7.5 and that of FE-LSPIM versus meshfree method as in Table 7.6.

Table 7.5. FE-LSPIM versus FEM.

Advantages	Disadvantages
<ul style="list-style-type: none"> ✧ High mesh distortion tolerance ✧ High order completeness properties ✧ Higher order global approximation can be obtained (by increasing the support size) without the necessarily of adding new nodes to the underlying mesh ✧ Computationally more efficient. 	<ul style="list-style-type: none"> ✧ Possible singularity of $\mathbf{P}$-matrix ✧ Shape functions more complex; Code development more difficult

Table 7.6. FE-LSPIM versus Meshfree methods.

Advantages	Disadvantages
<ul style="list-style-type: none"> ✧ Compatibility properties ✧ Free from the difficulties associated with numerical integration ✧ Establishing nodal connectivity is easy ✧ The shape functions possess the much desired <i>Kronecker delta</i> property 	<ul style="list-style-type: none"> ✧ Mesh based method

7.8. CONCLUSIONS

In this chapter, the formulation of FE-LSPIM QUAD4 has been critically reviewed and its merits and demerits have been highlighted. One of the demerits is the possibility of $\mathbf{P}$ -matrix becoming singular under some conditions. Remedial approaches have been suggested. An important merit is that FE-LSPIM QUAD4 seems to be free from linear

CHAPTER 7. CRITICAL ASSESSMENT OF FE-LSPIM QUAD4 ELEMENT

independence problem which is a serious problem in other PU based methods. Also, FE-LSPIM QUAD4 is computationally more efficient than QUAD4. Furthermore, the inf-sup test reveals that FE-LSPIM QUAD4 is computationally stable at the least for the problems investigated in section 7.5. Although the numerical problem investigated can not guarantee an outright stability for all the problems, the test conducted here suggest that there is no serious stability issues in the formulation.

CHAPTER 8. CONCLUDING REMARKS

8.1. SUMMARY

In this thesis, a few hybrid FE-meshfree elements have been developed in an attempt to synergize the strengths of FE and meshfree methods while avoiding their weaknesses. The performances of these elements have been tested using a variety of numerical examples. The focus of this research, in addition to a detailed review of literature, has been on:

1. Development of a FE-EFG method.
2. Development of a FE-PIM method.
3. Development a FE-LSPIM QUAD4 element and a FE-LSPIM QUADU element.
4. Extension of the FE-LSPIM QUAD4 to free vibration analysis of 2-D solids.
5. Extension of the FE-LSPIM QUAD4 to geometric nonlinear analysis.
6. A critical review of FE-LSPIM QUAD4.

8.2. MAJOR CONCLUSIONS

After a detailed review of related literature in Chapter 2, the FE-EFG method has been developed in Chapter 3. The EFG shape functions have been used as trial functions and FEM shape function as test functions in this approach. This method has been found to perform well even under distorted meshes. However, the EFG shape functions constructed by MLS approximation lack *Kronecker delta* properties, so it is difficult to implement the displacement boundary conditions. The FE-EFG method inherits this problem too. To resolve this problem, the PIM shape functions have been tried as trial

CHAPTER 8. CONCLUDING REMARKS

functions in the second part of Chapter 3. The resulting hybrid method has been labelled as FE-PIM. The PIM shape functions satisfy *Kronecker delta* properties, so it is easy to implement the boundary conditions. The point interpolation technique used in PIM is also more accurate and easier to implement than the MLS approximation, especially for regular nodal distributions. As a result, the FE-PIM method also inherits these advantages. Although, the FE-PIM has some improvement compared to FE-EFG, two drawbacks still exist:

- 1) The unsymmetry of the stiffness matrix is a serious disadvantage for vibration analysis because the eigenvalues of the unsymmetric matrices can then be complex even for the undamped free vibration problems, and this can pose difficulty in their interpretation.
- 2) The interpolation matrix (**P**-matrix) can become singular for certain nodal arrangements and thus positive definiteness of PIM interpolations cannot be guaranteed.

In order to overcome these drawbacks, an improved element has been derived using the FE-LSPIM method proposed in the first part of Chapter 4. In the formulation of this element, the same set of shape functions are used for test and trial functions, and they are obtained by multiplying the finite element and LSPIM shape functions appropriately. (The LSPIM shape functions are the PIM shape functions obtained in a least-square sense.) The composite shape functions so obtained have superior properties compared to the FE shape functions as well as PIM shape functions. Compared to the elements proposed in Chapter 3, the FE-LSPIM QUAD4 has the following advantages:

CHAPTER 8. CONCLUDING REMARKS

- 1) The element yields symmetric stiffness matrices and hence is easy to apply to vibration problems.
- 2) The shape functions of the proposed FE-LSPIM QUAD4 have been specially developed so that they possess mesh-distortion-independent inter-element compatibility properties inherited from finite element method and also mesh-distortion-independent higher order (quadratic) polynomial completeness in physical space inherited from meshfree (PIM) methods. These two properties enable the element produce efficient results even with extremely distorted meshes.
- 3) The FE-LSPIM QUAD4 is as efficient as EFG method but has a finite element format in terms of its implementation. The equation solver of FEM (which is based on the element-by-element assembly concept) can straightaway be used with very little modification. Thus, unlike the meshfree methods, the code development is as straightforward as FEM.
- 4) In contrast to meshfree methods like EFG, the element shape functions of FE-LSPIM QUAD4 have *Kronecker delta* properties.
- 5) The accuracy of meshfree methods depends to a great extent on node connectivity and correct choice of nodal connectivity is crucial to get accurate results. In the present method, the nodal connectivity is easily established using element connectivity.

But the advantages of FE-LSPIM QUAD4 do not come without a cost. As the shape functions of FE-LSPIM QUAD4 are constructed by combining FEM QUAD4 shape functions and LSPIM shape functions, it leads to an increase in computing time. The FE-LSPIM QUAD4U element has been developed in the second part of Chapter 4. It is

CHAPTER 8. CONCLUDING REMARKS

an unsymmetric element following an approach similar to that of FE-EFG and FE-PIM discussed in Chapter 3. The difference is that the FE-LSPIM QUAD4 shape functions are used as trial functions. This element has been observed to have all the advantages of the FE-LSPIM QUAD4. However, the disadvantage of this element is that the stiffness matrix is unsymmetric.

Several benchmark problems have been analyzed to assess the performance of both elements. It has been found that the accuracy of FE-LSPIM QUAD4 and FE-LSPIM QUAD4U is as good as the meshfree methods (e.g., EFG) and better than QUAD4 and QM6. Furthermore, both of the elements have excellent tolerance to the mesh distortions.

The FE-LSPIM QUAD4 element has been applied to the free vibration analysis of the cantilever beam and a shear wall with four opens in Chapter 5. The natural frequencies obtained with FE-LSPIM QUAD4 were compared with published results. The results of FE-LSPIM QUAD4 were comparable to published results under regular mesh and better than published results under coarse mesh.

In Chapter 6, FE-LSPIM QUAD4 has been extended to geometrical nonlinear analyses based on the updated Lagrangian formulation. The accuracy of the element has been assessed by comparing the results with those obtained using ANSYS elements. The proposed element is comparable to ANSYS element under regular mesh. However, the element yields better results under distorted meshes.

8.3. FUTURE WORK

An attempt has been made in this thesis to develop hybrid FE-Meshfree methods that can synergize the strengths of the FE and meshfree methods while avoiding their weaknesses. A few elements have been developed based on these methods. Static, vibration and geometric nonlinear applications have been tested. There remain some unsolved problems which may be taken up for future investigations in order to further exploit the full potential of the proposed FE-Meshfree method. A broad outline of the suggested future research is as follows:

1. Extension of the proposed elements to material nonlinear analysis. Conventional finite element analysis of metal forming process often breaks down due to severe mesh distortion. The FE-LSPIM QUAD4 has been found to yield results that are highly tolerant to mesh distortions. As a future direction, the application of the FE-LSPIM can be explored for the metal forming simulation. The focus would be on simulating large plastic deformations leading to severe mesh distortions.
2. Extension of the proposed elements to 3-D dimensional analysis: All the numerical examples have been considered in this thesis are 2-D problems. The proposed elements can be extended to 3-D analysis and also for beam/plate/shell applications.
3. All the formulations in this thesis have been restricted to QUAD4 elements. The hybrid FE-Meshfree method can also be extended to other element types like triangular elements.
4. The advantages of the proposed elements do not come without a cost. Currently, the computational time taken by the present element is as high as in meshfree methods. The computational efficiency may be improved by more efficient implementation.

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

APPENDIX: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

-----
% Matlab program of FE-LSPIM QUAD4 for cantilever beam
-----
clear all;
-----
% Set the overall dimensional, material and load parameters
-----
Lb = 48;
D=12;
young = 30e6; nu=0.3;
P=1000;
M=24000;
-----
% The type of loading
-----
problem=input('problem? (type: 1 for uniform tensile load; 2 for constant moment; 3 for
shear force')
-----
% Choose the type of contour plot
-----
mm=input('type of contour plot? (type: 1 for ux; 2 for uy; 3 for sxx; 4 for syy; 5 for sxy')
-----
% Set up plane stress D-matrix
-----
Dmat = (young/(1-nu^2))*[1 nu 0;nu 1 0;0 0 (1-nu)/2];
-----
% Read the mesh data (from ANSYS files)
-----

load nlist_P1m3.m; nlist=nlist_P1m3; clear nlist_P1m3;
load elist_P1m3.m; elist=elist_P1m3; clear elist_P1m3;
load dlist_P1m3.m; dlist=dlist_P1m3; clear dlist_P1m3;
-----
% Read nodal coordinates
-----
tmp=size(nlist);
numnod=tmp(1);
x(1:2,:)=nlist(:,2:3)';
xx(1:2,:)=nlist(:,2:3)';
-----
% Read element connectivity
-----
tmp=size(elist);
numcell=tmp(1);
conn(1:4,:)=elist(:,6:9)';
if mesh==5
    conn(1:4,:)=elist(:,7:10)';
end
-----
% Read displacement boundary conditions
-----

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

tmp=size(dlist);
nnu=tmp(1);
iffix(1:numnod,1:2)=0;
iffix1(1:numnod*2)=0;
for i=1:nnu
    node=dlist(i,1);
    i1=(node-1)*2+1; i2=i1+1;
    if dlist(i,2)==1, iffix(node,1)=1; iffix1(i1)=1; end
    if dlist(i,2)==2, iffix(node,2)=1; iffix1(i2)=1; end
end

ng = input('Howmany Gauss points?');
[gauss] = gausspt(ng);
-----
% Establish element-node connectivities
-----

[list1,list2,list4] = list(conn,numcell,numnod);
-----
% Compute the stiffness matrix of elements and assemble them
-----

Kmat = zeros(numnod*2,numnod*2);

for iel=1:numcell % loop over no. of elements
    x1=x(1,conn(1,iel)); x2=x(1,conn(2,iel)); x3=x(1,conn(3,iel)); x4=x(1,conn(4,iel));
    y1=x(2,conn(1,iel)); y2=x(2,conn(2,iel)); y3=x(2,conn(3,iel)); y4=x(2,conn(4,iel));

    for inode=1:4 %Extract nodal coords for 'iel_th' element
        elcod(inode,1)=x(1,conn(inode,iel));
        elcod(inode,2)=x(2,conn(inode,iel));
    end

    for ig=1:ng %loop1 over gauss points
        r=gauss(1,ig);
        w1=gauss(2,ig);
        for jg=1:ng %loop2 over gauss points
            s=gauss(1,jg);
            w2=gauss(2,jg);
            shape1=shapel(r,s);
            gpos=shape1*elcod;
            [Bmat1,SHY1,jac,N,DNX,DNY] = Bmatl(iel,r,s,conn,elcod,numnod);
            [Bmat2,SHY]=Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,iffix1);
            Kmat=Kmat+(w1*w2*jac)*Bmat2'*Dmat*Bmat2;
        end %loop2 over gauss points ends here
    end %loop1 over gauss points ends here
end % end loop over no. of elements
-----
% Compute the element load vector due to traction and assemble them
-----

force = zeros(1,numnod*2);
lmo = (1/12)*D^3;
count=0
for iel=1:numcell % loop over no. of elements
    iyes=0;

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

for inode=1:4
    for j=1:count
        if iel==niel(j);
            iyes=1;
        end
    end
end

if x(1,conn(inode,iel))==0 & iyes==0
    count=count+1;
    niel(count)=iel;
end

for k=1:count
    if iel==niel(k)
        iyes=1;
    end
end
if x(1,conn(inode,iel))==48 & iyes==0
    count=count+1;
    niel(count)=iel;
end
end %loop for k
end % loop over no. of element

k=length(niel);

for i=1:k
    iel=niel(i);
    for inode=1:4 %Extract nodal coords for 'iel_th' element
        elcod(inode,1)=x(1,conn(inode,iel));
        elcod(inode,2)=x(2,conn(inode,iel));
        if x(1,conn(inode,iel))==0
            r=-1; PP=-P; MM=-M;
        end

        if x(1,conn(inode,iel))==48
            r=1; PP=P; MM=M;
        end
    end
    % loop over k

    for jg=1:ng %loop2 over gauss points
        s=gauss(1,jg);
        w2=gauss(2,jg);
        shape1=shapel(r,s);
        gpos=shape1*elcod;
        [Bmat1,SHY1,jac,N,DNX,DNY] = Bmatl(iel,r,s,conn,elcod,numnod);
        [Bmat2,SHY] = Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,ifix1);

        sderi(2,1)=-0.25*(1.0-r);
        sderi(2,2)=-0.25*(1.0+r);
        sderi(2,3)= 0.25*(1.0+r);
        sderi(2,4)= 0.25*(1.0-r);
        jacc=sderi(2,:)*elcod(:,2);
    end
end

%-----
%uniform tensile load

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

-----
if problem==1
    tx= PP/D;
    ty=0;
end
-----
%   Constant bending moment
-----

if problem==2
    tx=(MM/lmo)*gpos(1,2);
    ty=0;
end

-----
%   Parabolically varying tip shear load
-----
if problem==3
    tx=0;
    if r==1
        tx=-P*(Lb-gpos(1,1))*gpos(1,2)/lmo;
    end
    ty=-(PP/(2*lmo))*((D^2)/4-gpos(1,2)^2);
end

    for k=1:numnod
        j2=2*k;
        j1=j2-1;
        force(j1)=force(j1)+ (w2*jacc)*SHY(k)*tx;
        force(j2)=force(j2)+ (w2*jacc)*SHY(k)*ty;

    end

    end %loop2 over gauss points ends here
end % end loop over no. of elements
-----
% Apply boundary conditions to the stiffness matrix
-----
kkk=Kmat;
for i=1:numnod*2
    if iff1x1(i) == 1
        for j=1:numnod*2
            if iff1x1(j) == 1
                kkk(i,j)=0;
                if i==j, kkk(i,j)=1000000000000000000; end
            end
        end
    end
end
end
end

disp=kkk\force';
for i=1:numnod
    if x(1,i)==48 & x(2,i)==0
        disp(2*i)
        if problem==1
            disp(2*i-1)
        end
    end
end

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

    end
end
end
-----
%Compute nodal displacements from nodal displacement parameters
-----
for i=1:numnod
    disp1(i)=disp(2*i-1);
    disp2(i)=disp(2*i);
end
% solve for output variables displacement
for iel=1:numcell % loop over no. of elements
    x1=x(1,conn(1,iel)); x2=x(1,conn(2,iel)); x3=x(1,conn(3,iel)); x4=x(1,conn(4,iel));
    y1=x(2,conn(1,iel)); y2=x(2,conn(2,iel)); y3=x(2,conn(3,iel)); y4=x(2,conn(4,iel));

    for inode=1:4 %Extract nodal coords for 'iel_th' element
        elcod(inode,1)=x(1,conn(inode,iel));
        elcod(inode,2)=x(2,conn(inode,iel));
    end
    for kk=1:4 %loop1 over element nodes
        a=[-1 1 1 -1; -1 -1 1 1];
        r=a(1,kk);
        s=a(2,kk);
        shape1=shapel(r,s);
        gpos=shape1*elcod;
        [Bmat1,SHY1,jac,N,DNX,DNY] = Bmatl(iel,r,s,conn,elcod,numnod);
        [Bmat2,SHY] = Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,ifix1);
        j1=2*conn(kk,iel)-1;
        j2=j1+1;
        u(j1)=SHY*disp1';
        u(j2)=SHY*disp2';
    end % loop kk
end %loop element

for i=1:numnod
    if xx(1,i)==48 & xx(2,i)==0
        u(2*i)
        if problem==1
            u(2*i-1)
        end
    end
end
end
-----
%Decide contour levels to be plotted for displacements
-----
uxmin=1e+32; uxmax=-1e+32;
uymin=1e+32; uymax=-1e+32;

for i=1:numnod
    i1=(i-1)*2+1; i2=i1+1;
    if disp(i1) > uxmax, uxmax=disp(i1); end
    if disp(i1) < uxmin, uxmin=disp(i1); end
    if disp(i2) > uymax, uymax=disp(i2); end
    if disp(i2) < uymin, uymin=disp(i2); end
end

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

clear ux;
uxmax=uxmax-(uxmax-uxmin)/150000;
uxmin=uxmin+(uxmax-uxmin)/150000;
kount=0;
for ux=uxmin:(uxmax-uxmin)/15:uxmax
    kount=kount+1;
    Vux(kount)=ux;
end

```

```

clear uy;
uymax=uymax-(uymax-uymin)/150000;
uymin=uymin+(uymax-uymin)/150000;
kount=0;
for uy=uymin:(uymax-uymin)/15:uymax
    kount=kount+1;
    Vuy(kount)=uy;
End

```

```

%Decide contour levels to be plotted for stresses

```

```

sxxmin=1e+32; sxxmax=-1e+32;
syymin=1e+32; syymax=-1e+32;
sxymin=1e+32; sxymin=-1e+32;

```

```

for iel=1:numcell

```

```

    x1=x(1,conn(1,iel)); x2=x(1,conn(2,iel)); x3=x(1,conn(3,iel)); x4=x(1,conn(4,iel));
    y1=x(2,conn(1,iel)); y2=x(2,conn(2,iel)); y3=x(2,conn(3,iel)); y4=x(2,conn(4,iel));
    for inode=1:4 %Extract nodal coords for 'iel_th' element

```

```

        elcod(inode,1)=x(1,conn(inode,iel));
        elcod(inode,2)=x(2,conn(inode,iel));
    end
    ip=0;
    for r=-1:1:1 %loop over 'r' values
        ip=ip+1;
        jp=0;
        for s=-1:1:1 %loop over 's' values
            jp=jp+1;

```

```

%   Compute coordinates of sampling points

```

```

        shape1=shapel(r,s);
        gpos=shape1*elcod;
        X(ip,jp)=gpos(1);    Y(ip,jp)=gpos(2);

```

```

%   Compute B-matrices at sampling points

```

```

[Bmat1,SHY1,deter,N,DNX,DNY] = Bmatl(iel,r,s,conn,elcod,numnod);
[Bmat2,SHY] = Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,ifix1);

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

% Compute displacements at sampling points

```

ux(ip,jp)=0; uy(ip,jp)=0;
for i=1:numnod
    i1=i*2-1; i2=i1+1;
    ux(ip,jp)=ux(ip,jp)+SHY(i)*disp(i1);
    uy(ip,jp)=uy(ip,jp)+SHY(i)*disp(i2);
end

```

% Compute stresses at sampling points

```

DBmat=Dmat*Bmat2;
stress=DBmat*disp;
sxx(ip,jp)=stress(1); syy(ip,jp)=stress(2); sxy(ip,jp)=stress(3);
end %loop over 'r' values
end %loop over 's' values

```

```

if max(max(sxx)) > sxxmax, sxxmax=max(max(sxx)); end
if min(min(sxx)) < sxxmin, sxxmin=min(min(sxx)); end
if max(max(syy)) > syymax, syymax=max(max(syy)); end
if min(min(syy)) < syymmin, syymmin=min(min(syy)); end
if max(max(sxy)) > sxymax, sxymax=max(max(sxy)); end
if min(min(sxy)) < sxymin, sxymin=min(min(sxy)); end

```

end

```

clear sxx;
sxxmax=sxxmax-(sxxmax-sxxmin)/150000;
sxxmin=sxxmin+(sxxmax-sxxmin)/150000;
kount=0;
for sxx=sxxmin:(sxxmax-sxxmin)/15:sxxmax
    kount=kount+1;
    Vsxx(kount)=sxx;
end

```

```

clear syy;
syymax=syymax-(syymax-syymmin)/150000;
syymmin=syymmin+(syymax-syymmin)/150000;
kount=0;
for syy=syymmin:(syymax-syymmin)/15:syymax
    kount=kount+1;
    Vsyy(kount)=syy;
end

```

```

clear sxy;
sxymax=sxymax-(sxymax-sxymin)/150000;
sxymin=sxymin+(sxymax-sxymin)/150000;
kount=0;
for sxy=sxymin:(sxymax-sxymin)/15:sxymax
    kount=kount+1;
    Vsxy(kount)=sxy;
end

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```
% Plot stress and displacement contours
```

```
-----
for iel=1:numcell
```

```
    x1=x(1,conn(1,iel)); x2=x(1,conn(2,iel)); x3=x(1,conn(3,iel)); x4=x(1,conn(4,iel));
    y1=x(2,conn(1,iel)); y2=x(2,conn(2,iel)); y3=x(2,conn(3,iel)); y4=x(2,conn(4,iel));
    x0=(x1+x2+x3+x4)/4; y0=(y1+y2+y3+y4)/4;
```

```
    for inode=1:4 %Extract nodal coords for 'iel_th' element
        elcod(inode,1)=x(1,conn(inode,iel));
        elcod(inode,2)=x(2,conn(inode,iel));
    end
```

```
    ip=0;
    for r=-1:0.2:1 %loop over 'r' values
        ip=ip+1;
        jp=0;
        for s=-1:0.2:1 %loop over 's' values
            jp=jp+1;
```

```
-----
% Compute coordinates of sampling points
```

```
-----
        shape1=shapel(r,s);
        gpos=shape1*elcod;
        X(ip,jp)=gpos(1);      Y(ip,jp)=gpos(2);
```

```
-----
%      Compute B-matrices at sampling points
```

```
-----
        [Bmat1,SHY1,deter,N,DNX,DNY] = Bmatl(iel,r,s,conn,elcod,numnod);
        [Bmat2,SHY] = Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,ifix1);
```

```
-----
%      Compute displacements at sampling points
```

```
-----
        ux(ip,jp)=0; uy(ip,jp)=0;
        for i=1:numnod
            i1=i*2-1; i2=i1+1;
            ux(ip,jp)=ux(ip,jp)+SHY(i)*disp(i1);
            uy(ip,jp)=uy(ip,jp)+SHY(i)*disp(i2);
        end
```

```
-----
%      Compute stresses at sampling points
```

```
-----
        DBmat=Dmat*Bmat2;
        stress=DBmat*disp;
        sxx(ip,jp)=stress(1); syy(ip,jp)=stress(2); sxy(ip,jp)=stress(3);
    end %loop over 'r' values
end %loop over 's' values
```

```

    if mm==1,
        contourf(X,Y',ux',Vux,'k');
    end
    if mm==2,
        contourf(X,Y',uy',Vuy,'k'); end
    if mm==3
```


APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

    contourf(X',Y',sxx,Vsxx,'k'); end
if mm==4,
    contourf(X',Y',syy,Vsyy,'k');end
if mm==5,
    contourf(X',Y',sxy,Vsxy,'k');end
if iel==1, hold; end
    plot([x1,x2,x3,x4,x1],[y1,y2,y3,y4,y1],'w')
    plot([x1,x2,x3,x4,x1],[y1,y2,y3,y4,y1],'k:')
    colorbar
if iel==numcell, hold; end
end

```

```

-----
%Compute axes lengths for contour plots
-----

```

```

xmin=min(x(1,:)); xmax=max(x(1,:));
ymin=min(x(2,:)); ymax=max(x(2,:));
axis([xmin,xmax,ymin,ymax]);
axis tight;

```

```

-----
% ANSYS output file nlist
-----

```

1	.000000000000	-6.000000000000	.000000000000
2	48.000000000000	-6.000000000000	.000000000000
3	3.000000000000	-6.000000000000	.000000000000
4	6.000000000000	-6.000000000000	.000000000000
5	9.000000000000	-6.000000000000	.000000000000
6	12.000000000000	-6.000000000000	.000000000000
7	15.000000000000	-6.000000000000	.000000000000
8	18.000000000000	-6.000000000000	.000000000000
9	21.000000000000	-6.000000000000	.000000000000
10	24.000000000000	-6.000000000000	.000000000000
11	27.000000000000	-6.000000000000	.000000000000
12	30.000000000000	-6.000000000000	.000000000000
13	33.000000000000	-6.000000000000	.000000000000
14	36.000000000000	-6.000000000000	.000000000000
15	39.000000000000	-6.000000000000	.000000000000
16	42.000000000000	-6.000000000000	.000000000000
17	45.000000000000	-6.000000000000	.000000000000
18	48.000000000000	6.000000000000	.000000000000
19	48.000000000000	-3.000000000000	.000000000000
20	48.000000000000	.000000000000	.000000000000
21	48.000000000000	3.000000000000	.000000000000
22	.000000000000	6.000000000000	.000000000000
23	45.000000000000	6.000000000000	.000000000000
24	42.000000000000	6.000000000000	.000000000000
25	39.000000000000	6.000000000000	.000000000000
26	36.000000000000	6.000000000000	.000000000000
27	33.000000000000	6.000000000000	.000000000000
28	30.000000000000	6.000000000000	.000000000000
29	27.000000000000	6.000000000000	.000000000000
30	24.000000000000	6.000000000000	.000000000000
31	21.000000000000	6.000000000000	.000000000000
32	18.000000000000	6.000000000000	.000000000000

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

33	15.0000000000	6.0000000000	.000000000000
34	12.0000000000	6.0000000000	.000000000000
35	9.0000000000	6.0000000000	.000000000000
36	6.0000000000	6.0000000000	.000000000000
37	3.0000000000	6.0000000000	.000000000000
38	.000000000000	3.0000000000	.000000000000
39	.000000000000	.000000000000	.000000000000
40	.000000000000	-3.0000000000	.000000000000
41	3.0000000000	-3.0000000000	.000000000000
42	3.0000000000	.000000000000	.000000000000
43	3.0000000000	3.0000000000	.000000000000
44	6.0000000000	-3.0000000000	.000000000000
45	6.0000000000	.000000000000	.000000000000
46	6.0000000000	3.0000000000	.000000000000
47	9.0000000000	-3.0000000000	.000000000000
48	9.0000000000	.000000000000	.000000000000
49	9.0000000000	3.0000000000	.000000000000
50	12.0000000000	-3.0000000000	.000000000000
51	12.0000000000	.000000000000	.000000000000
52	12.0000000000	3.0000000000	.000000000000
53	15.0000000000	-3.0000000000	.000000000000
54	15.0000000000	.000000000000	.000000000000
55	15.0000000000	3.0000000000	.000000000000
56	18.0000000000	-3.0000000000	.000000000000
57	18.0000000000	.000000000000	.000000000000
58	18.0000000000	3.0000000000	.000000000000
59	21.0000000000	-3.0000000000	.000000000000
60	21.0000000000	.000000000000	.000000000000
61	21.0000000000	3.0000000000	.000000000000
62	24.0000000000	-3.0000000000	.000000000000
63	24.0000000000	.000000000000	.000000000000
64	24.0000000000	3.0000000000	.000000000000
65	27.0000000000	-3.0000000000	.000000000000
66	27.0000000000	.000000000000	.000000000000
67	27.0000000000	3.0000000000	.000000000000
68	30.0000000000	-3.0000000000	.000000000000
69	30.0000000000	.000000000000	.000000000000
70	30.0000000000	3.0000000000	.000000000000
71	33.0000000000	-3.0000000000	.000000000000
72	33.0000000000	.000000000000	.000000000000
73	33.0000000000	3.0000000000	.000000000000
74	36.0000000000	-3.0000000000	.000000000000
75	36.0000000000	.000000000000	.000000000000
76	36.0000000000	3.0000000000	.000000000000
77	39.0000000000	-3.0000000000	.000000000000
78	39.0000000000	.000000000000	.000000000000
79	39.0000000000	3.0000000000	.000000000000
80	42.0000000000	-3.0000000000	.000000000000
81	42.0000000000	.000000000000	.000000000000
82	42.0000000000	3.0000000000	.000000000000
83	45.0000000000	-3.0000000000	.000000000000
84	45.0000000000	.000000000000	.000000000000
85	45.0000000000	3.0000000000	.000000000000

% ANSYS output file elist

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

1	1	1	1	0	1	3	41	40
2	1	1	1	0	3	4	44	41
3	1	1	1	0	4	5	47	44
4	1	1	1	0	5	6	50	47
5	1	1	1	0	6	7	53	50
6	1	1	1	0	7	8	56	53
7	1	1	1	0	8	9	59	56
8	1	1	1	0	9	10	62	59
9	1	1	1	0	10	11	65	62
10	1	1	1	0	11	12	68	65
11	1	1	1	0	12	13	71	68
12	1	1	1	0	13	14	74	71
13	1	1	1	0	14	15	77	74
14	1	1	1	0	15	16	80	77
15	1	1	1	0	16	17	83	80
16	1	1	1	0	17	2	19	83
17	1	1	1	0	40	41	42	39
18	1	1	1	0	41	44	45	42
19	1	1	1	0	44	47	48	45
20	1	1	1	0	47	50	51	48
21	1	1	1	0	50	53	54	51
22	1	1	1	0	53	56	57	54
23	1	1	1	0	56	59	60	57
24	1	1	1	0	59	62	63	60
25	1	1	1	0	62	65	66	63
26	1	1	1	0	65	68	69	66
27	1	1	1	0	68	71	72	69
28	1	1	1	0	71	74	75	72
29	1	1	1	0	74	77	78	75
30	1	1	1	0	77	80	81	78
31	1	1	1	0	80	83	84	81
32	1	1	1	0	83	19	20	84
33	1	1	1	0	39	42	43	38
34	1	1	1	0	42	45	46	43
35	1	1	1	0	45	48	49	46
36	1	1	1	0	48	51	52	49
37	1	1	1	0	51	54	55	52
38	1	1	1	0	54	57	58	55
39	1	1	1	0	57	60	61	58
40	1	1	1	0	60	63	64	61
41	1	1	1	0	63	66	67	64
42	1	1	1	0	66	69	70	67
43	1	1	1	0	69	72	73	70
44	1	1	1	0	72	75	76	73
45	1	1	1	0	75	78	79	76
46	1	1	1	0	78	81	82	79
47	1	1	1	0	81	84	85	82
48	1	1	1	0	84	20	21	85
49	1	1	1	0	38	43	37	22
50	1	1	1	0	43	46	36	37
51	1	1	1	0	46	49	35	36
52	1	1	1	0	49	52	34	35
53	1	1	1	0	52	55	33	34
54	1	1	1	0	55	58	32	33

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

55 1 1 1 0 58 61 31 32
56 1 1 1 0 61 64 30 31
57 1 1 1 0 64 67 29 30
58 1 1 1 0 67 70 28 29
59 1 1 1 0 70 73 27 28
60 1 1 1 0 73 76 26 27
61 1 1 1 0 76 79 25 26
62 1 1 1 0 79 82 24 25
63 1 1 1 0 82 85 23 24
64 1 1 1 0 85 21 18 23

```

```
% ANSYS output file dlist
```

```

1 1 .000000000 .000000000
22 1 .000000000 .000000000
39 1 .000000000 .000000000
39 2 .000000000 .000000000

```

```
% Function to compute gauss point
```

```
function v = gausspt(k)
```

```

if k==6,
% 4 gauss points and their weights
v(1,1)=-0.932469514203152;
v(1,2)=-0.661209386466265;
v(1,3)=-0.238619186083197;
v(1,4)=-v(1,3);
v(1,5)=-v(1,2);
v(1,6)=-v(1,1);

v(2,1)=0.171324492379170;
v(2,2)=0.360761573048139;
v(2,3)=0.467913934572691;
v(2,4)=v(2,3);
v(2,5)=v(2,2);
v(2,6)=v(2,1);
end

```

```

if k==5,
% 4 gauss points and their weights
v(1,1)=-0.906179845938664;
v(1,2)=-0.538469310105683;
v(1,3)=0.0;
v(1,4)=-v(1,2);
v(1,5)=-v(1,1);
v(2,1)=0.236926885056189;
v(2,2)=0.478628670499366;
v(2,3)=0.568888888888889;
v(2,4)=v(2,2);
v(2,5)=v(2,1);
end

```

```

if k==4,

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```
% 4 gauss points and their weights
v(1,1)=-.861136311594052575224;
v(1,2)=-.339981043584856264803;
v(1,3)=-v(1,2);
v(1,4)=-v(1,1);
v(2,1)=.347854845137453857373;
v(2,2)=.652145154862546142627;
v(2,3)=v(2,2);
v(2,4)=v(2,1);
end
```

```
if k==3,
% 3 gauss points and their weights
v(1,1)=-0.774596669241483;
v(1,2)=0.0000000000000000;
v(1,3)=-v(1,1);
v(2,1)=0.5555555555555555;
v(2,2)=0.8888888888888888;
v(2,3)=v(2,1);
end
```

```
if k==2,
% 2 gauss points and their weights
v(1,1)=-0.57735026918962;
v(1,2)=-v(1,1);
v(2,1)=1.0000000000000000;
v(2,2)=1.0000000000000000;
end
```

```
if k==1,
% 1 gauss points and their weights
v(1,1)=0.0000000000000000;
v(2,1)=2.0000000000000000;
end
```

```
% Function to Establish element-node connectivity
```

```
% list1=elements connected to nodes
% list2=nodes connected to nodes (1st order nodal support)
% list3=nodes connected to elements (element support)
% list4=nodes connected to nodes connected to nodes (2nd order nodal support)
```

```
function [list1,list2,list4] = list(conn,numcell,numnod)
```

```
% create the list of elements connected each node
list1=0;
for inode=1:numnod
kount=0;
for iel=1:numcell
for j=1:4
if conn(j,iel)==inode;
kount=kount+1;
```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

        list1(inode,1)=kount;
        list1(inode,kount+1)=iel;
    end % if
    end %j
    end % iel
    end % end inode

list2=0;
-----
%   create the list of nodes connected each node
-----
        for inode=1:numnod
            kount=0;
            for i=1:list1(inode,1)
                iel=list1(inode,i+1);
                for j=1:4
-----
%       check if this node is already listed
-----
-       iyes=0;
            if conn(j,iel)==inode;
                iyes=1;
            end
            for k=1:kount
                if list2(inode,k+1)==conn(j,iel)
                    iyes=1;
                end
            end
            end

-----
%       if not enlist it
-----
                if iyes==0,
                    kount=kount+1;
                    list2(inode,1)=kount;
                    list2(inode,kount+1)=conn(j,iel);
                    end %if

            end %j
        end %i
    end % inode

list4=0;
-----
%creat the list node connected to the node.a bigger support
-----
        for inode=1:numnod
            kount=0;
            for j=1:list2(inode,1)
                n=list2(inode,j+1);
                for k=1:list2(n,1)
                    %check if this node is already listed
                    iyes=0;
                    if list2(n,k+1)==inode;
                        iyes=1;

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

        end
    for kk=1:kount

        if list4(inode, kk+1)==list2(n, k+1);
            iyes=1;
        end

    end % for kk

-----
    %if not enlist it
    -----
    --    if iyes==0,
        kount=kount+1;
        list4(inode, 1)=kount;
        list4(inode, kount+1)=list2(n, k+1);
    end %if
    end %for k
    end %for j
end %for inode

-----
% Function to Establish FEM shape functions
-----
function shape1=shapel(r,s)
shape1(1)=0.25*(1.0-r)*(1.0-s);
shape1(2)=0.25*(1.0+r)*(1.0-s);
shape1(3)=0.25*(1.0+r)*(1.0+s);
shape1(4)=0.25*(1.0-r)*(1.0+s);

-----
% Function to Establish FEM shape functions and derivative matrix
-----
function [Bmat1, SHY1, deter, N, DNX, DNY] = Bmatl(iel, r, s, conn, elcod, numnod)

Bmat1=zeros(3, 2*numnod);
SHY1=zeros(1, numnod);

nnode=4;
ndime=2;
ndof=2;

N1=0.25*(1.0-r)*(1.0-s);
N2=0.25*(1.0+r)*(1.0-s);
N3=0.25*(1.0+r)*(1.0+s);
N4=0.25*(1.0-r)*(1.0+s);

N=[N1 N2 N3 N4];

sderi(1,1)=-0.25*(1.0-s);
sderi(1,2)= 0.25*(1.0-s);
sderi(1,3)= 0.25*(1.0+s);
sderi(1,4)=-0.25*(1.0+s);

sderi(2,1)=-0.25*(1.0-r);
sderi(2,2)=-0.25*(1.0+r);

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

sderi(2,3)= 0.25*(1.0+r);
sderi(2,4)= 0.25*(1.0-r);

%form the jacobian matrix
for i=1:ndime,
    for j=1:ndime,
        xjaco(i,j)=0.0;
        for k=1:nnode,
            xjaco(i,j)=xjaco(i,j)+sderi(i,k)*elcod(k,j);
        end
    end
end
% find the inverse of the jacobian matrix
x11=xjaco(1,1);
x12=xjaco(1,2);
x21=xjaco(2,1);
x22=xjaco(2,2);

-----
%find the determinant of the jacobian matrix
-----
deter=x11*x22-x12*x21;
-----
-%find the inverse of jacobian matrix
-----
ocajx(1,1)= x22/deter;
ocajx(1,2)=-x12/deter;
ocajx(2,1)=-x21/deter;
ocajx(2,2)= x11/deter;
%find the cartesian derivatives from the derivatives
%in natural coordinates
for i=1:ndime,
    for j=1:nnode,
        cartd(i,j)=0.0;
        for k=1:ndime,
            cartd(i,j)=cartd(i,j)+ocajx(i,k)*sderi(k,j);
        end
    end
end

for j=1:nnode,
    node=conn(j,iel);
    j2 = ndof*node;
    j1 = j2 - 1;
    Bmat1(1,j1)=cartd(1,j);
    Bmat1(1,j2)=0.0;
    Bmat1(2,j1)=0.0;
    Bmat1(2,j2)=cartd(2,j);
    Bmat1(3,j1)=cartd(2,j);
    Bmat1(3,j2)=cartd(1,j);
    SHY1(node)=N(j);
end

DNX=[cartd(1,1) cartd(1,2) cartd(1,3) cartd(1,4) ];
DNY=[cartd(2,1) cartd(2,2) cartd(2,3) cartd(2,4) ];

```


APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

-----
% Function to Establish FE-LSPIM shape functions and derivative matrix
-----

function [Bmat2,SHY] = Bmatr(gpos,xx,conn,list1,list2,list4,iel,N,DNX,DNY,numnod,ifix1)

% LL = number of nodes in the influence domain of a given node

PHI=zeros(4,numnod);
DPHIX=zeros(4,numnod);
DPHIY=zeros(4,numnod);
Bmat2=zeros(3,numnod*2);
SHY=zeros(1,numnod);
for node=1:4
    inode=conn(node,iel);
    LL=list4(inode,1);
    vv=list4(inode,2:LL+1);

    vv(LL+1)=inode; % add node i to the support of node i

    won = ones(1,LL);
    xq=gpos(1);yq=gpos(2);

    x2=xx(1,vv(1:LL));
    y2=xx(2,vv(1:LL));
    x=x2;y=y2;

    clear pmat;
    clear qmat;

    pmat1=[x;y; x.*y];
    pmat2=[pmat1; x.*x; y.*y];
    pmat3=[pmat2; (x.*x).* y; (y.*y).* x];
    rhs=[xq,yq,xq*yq,xq*xq,yq*yq,xq*xq*yq,xq*yq*yq];
    rhsx=[1,0,yq,2*xq,0,2*xq*yq,yq*yq];
    rhsy=[0,1,xq,0,2*yq,xq*xq,2*xq*yq];

    xk=xx(1,inode); yk=xx(2,inode);
    pmatk=[xk*won; yk*won; xk*yk*won; xk*xk*won; yk*yk*won; xk*xk*yk*won; yk*yk*xk*won];
    pmat3=pmat3-pmatk;
    rhsk=[xk,yk,xk*yk,xk*xk,yk*yk,xk*xk*yk,xk*yk*yk];
    rhs=rhs-rhsk;

    qmat=pmat3';

    pmat=inv(qmat'*qmat)*qmat';
    phi=rhs*pmat;
    phi=phi';
    dphix=rhsx*pmat;
    dphiy=rhsy*pmat;

    phi(LL+1,1)=1.0-rhs*pmat*won';
    dphix(1,LL+1)=-rhsx*pmat*won';
    dphiy(1,LL+1)=-rhsy*pmat*won';

```

APPENDIX I: TWO DIMENSIONAL FE-LSPIM QUAD4 PROGRAM

```

LL=LL+1;

en1=vv(1:LL);
for j=1:LL
    PHI(node,en1(j))=phi(j);
    DPHIX(node,en1(j))=dphix(j);
    DPHIY(node,en1(j))=dphiy(j);
end % end nodes connectivity
end % end element nodes loop

SHY=N*PHI;
DSHX=DNX*PHI+N*DPHIX;
DSHY=DNY*PHI+N*DPHIY;

for k=1:numnod
    j2=2*k;
    j1=j2-1;
    Bmat2(1,j1)=DSHX(k);
    Bmat2(1,j2)=0.0;
    Bmat2(2,j1)=0.0;
    Bmat2(2,j2)=DSHY(k);
    Bmat2(3,j1)=DSHY(k);
    Bmat2(3,j2)=DSHX(k);
end;

```

LIST OF PUBLICATIONS

1. S. Rajendran, B. R. Zhang, A 'FE-MESHFREE' QUAD4 element based on partition of unity, Computer Methods in Applied Mechanics and Engineering. 2007; **197**: 128-147.
2. B. R. Zhang and S. Rajendran, A 'FE-MESHFREE' synergetic QUAD4 element for the free vibration analysis of 2-D solids. Computer Methods in Applied Mechanics and Engineering. (Accepted).
3. B. R. Zhang and S. Rajendran, 'The FE-MESHFREE' QUAD4 element for the geometric nonlinear analysis. Finite Elements in Analysis and Design (Submitted).

REFERENCES

1. T. J. R. Hughes, M. Cohen, M. Haroun, “Reduced and selective integration techniques in finite element analysis of plates”, *Nuclear Engineering and Design*, 1978; **46**:203-222.
2. T. J. R. Hughes, T. E. Tezduyar, “Finite elements based upon mindlin plate theory with particular reference to the four-node bilinear isoparametric element”, *Journal of Applied Mechanics*, 1981; **48**:587-596.
3. R. H. Macneal, “A simple quadrilateral shell element”, *Computers and Structures*, 1978; **8**:175-183.
4. R. L. Taylor, “Finite element for general shell analysis”, *5th International Seminar on Computational Aspects of the Finite Element Method*, Berlin, 1979.
5. T. J. R. Hughes, R. L. Taylor, W. Kanoknukulchai, “A simple and efficient finite element for plate bending”, *International Journal for Numerical Methods in Engineering*, 1977; **11**:1529–1543.
6. J. L. Batoz, G. Dhatt, “Modelisation des Structures par Elements Finis”, *Poutres et Plaques*. Editions Hermite: Paris, 1990 vol. 2.
7. N. Bicanic, E. Hinton, “Spurious modes in two-dimensional isoparametric elements”, *International Journal for Numerical Methods in Engineering*, 1979; **14**:1545–1557.
8. A. K. Noor, C. M. Anderson, “Mixed models and reduced/selective integration displacement models for nonlinear shell analysis”, *International Journal for Numerical Methods in Engineering*, 1982; **18**:1429–1454.
9. K. T. Kavanagh, S. W. Key, “A note on selective and reduced integration techniques in finite element method”, *International Journal for Numerical Methods in Engineering*, 1972; **4**:148–150.

10. D. P. Flanagan, T. Belytschko, “A uniform strain hexahedron and quadrilateral with orthogonal hourglass control”, *International Journal for Numerical Methods in Engineering*, 1981; **17**:679–706.
11. T. Belytschko, C. S. Tsay, “A stabilisation procedure for the quadrilateral plate element with one-point quadrature”, *International Journal for Numerical Methods in Engineering*, 1983; **19**:405–419.
12. T. Belytschko, H. Stolarski, N. Carpenter, “A C^0 triangular plate element with one-point quadrature”, *International Journal for Numerical Methods in Engineering*, 1984; **20**:787–802.
13. T. Belytschko, W. K. Liu, E. Engelmann, “The gamma-elements and related developments”, In “Finite Element Methods for Plate and Shell Structures”, vol. 1, T. J. R. Hughes, E. Hinton (eds), Pineridge Press: Swansea, U.K., 1986; 316-347.
14. G. Prathap, S. Vishwanath, “An optimally integrated four-node quadrilateral plate bending element”, *International Journal for Numerical Methods in Engineering*, 1983; **19**:831–840.
15. G. Prathap, B. R. Somashekar, “Field-and edge-consistency synthesis of a 4-node quadrilateral plate bending element”, *International Journal for Numerical Methods in Engineering*, 1988; **26**:1693–1708.
16. T. J. R. Hughes, T. E. Tezduyar, “Finite elements based upon Mindlin plate theory with particular reference to the four-node bilinear isoparametric element”, *Journal of Applied Mechanics (ASME)*, 1981; **48**:587–596.
17. E. L. Wilson, R. L. Taylor, W. P. Doherty, J. Ghabussi, “Incompatible displacement models”, in “Numerical and Computer Methods in Structural Mechanics”, Academic Press, New York, 1973; 43-57.

-
18. R. L. Taylor, P. J. Beresford, E. L. Wilson, "A non-conforming element for stress analysis", *International Journal for Numerical Methods in Engineering*, 1976; **10**:1211–1219.
 19. R. L. Taylor, J. C. Simo, O. C. Zienkiewicz, A. C. H. Chan, "The patch test - a condition for assessing FEM convergence", *International Journal for Numerical Methods in Engineering*, 1986; **22**:39–62.
 20. C. C. Wu, M. G. Huang, T. H. H. Pian, "Consistency condition and convergence criteria of incompatible elements: general formulation of incompatible functions and its application", *Computers and Structures*, 1987; **27**:639–644.
 21. E. L. Wilson, A. Ibrahimbegovic, "Use of incompatible displacement modes for the calculation of element stiffnesses or stresses", *Finite Element in Analysis and Design*, 1990; **7**:229-241.
 22. U. Hueck, B. D. Reddy, P. Wriggers, "One the stabilization of the rectangular 4-node quadrilateral element", *Communications in Numerical Methods in Engineering*, 1994; **10**:555-563.
 23. U. Hueck, P. Wriggers, "A formulation for the 4-node quadrilateral element", *International Journal for Numerical Methods in Engineering*, 1995; **38**:3007-3037.
 24. P. Wriggers, U. Hueck, "A formulation of the QS6 element for large elastic deformation", *International Journal for Numerical Methods in Engineering*, 1996; **39**:1437-1454.
 25. C. C. Wu, Y. Q. Huang, E. Ramm, "A further study on incompatible models: revise-stiffness approach and completeness of trial functions", *Computer Methods in Applied mechanics and Engineering*, 2001; **190**:5923-5934.
 26. Y. Q. Huang, Q. S. Li, "Four-node incompatible plane and axisymmetric elements

-
- with quadratic completeness in physical space”, *International Journal for Numerical Methods in Engineering*, 2004; **61**:1603-1624.
27. T. H. H. Pian, “Derivation of element stiffness matrices by assumed stress distributions”, *AIAA Journal*, 1964; **2**:1333-1336.
 28. T. H. H. Pian, K. Sumihara, “Rational approach for assumed stress finite elements”, *International Journal for Numerical Methods in Engineering*, 1984; **20**:1685-1695.
 29. T. H. H. Pian, P. Tong, “Relation between incompatible displacement model and hybrid stress model”, *International Journal for Numerical Methods in Engineering*, 1986; **22**:173-181.
 30. T. H. H. Pian, C. C. Wu, “A rational approach for choosing stress terms for hybrid finite element formulations”, *International Journal for Numerical Methods in Engineering*, 1988; **26**:2331-2343.
 31. K. Y. Yuan, Y. S. Huang, T. H. H. Pian, “New strategy for assumed stresses for 4-node hybrid stress membrane element”, *International Journal for Numerical Methods in Engineering*, 1993; **36**:1747-1763.
 32. K. Y. Yuan, J. C. Wen, T. H. H. Pian, “A unified theory for formulation of hybrid stress membrane elements”, *International Journal for Numerical Methods in Engineering*, 1994; **37**:457-474.
 33. W. J. Chen, Y. K. Cheung, “A new approach for the hybrid element method”, *International Journal for Numerical Methods in Engineering*, 1987; **24**:1697-1709.
 34. W. J. Chen, Y. K. Cheung, “A robust refined quadrilateral plane element”, *International Journal for Numerical Methods in Engineering*, 1995; **38**:649-666.
 35. M. C. Bouzeghoub, M. J. Gunn, “On stress interpolation for hybrid models”, *International Journal for Numerical Methods in Engineering*, 1994; **37**:895-904.

-
36. C. C. Wu, Y. K. Cheung, "On optimization approaches of hybrid stress elements", *Finite Elements in Analysis and Design*, 1995; **21**:111-128.
 37. K. Y. Sze, H. Fan, "An economical assumed stress brick element and its implementation", *Finite Elements in Analysis and Design*, 1996; **21**:179-200.
 38. S. T. Yeo, B. C. Lee, "New stress assumption for hybrid stress elements and refined four-node plane and eight-node brick elements", *International Journal for Numerical Methods in Engineering*, 1997; **40**:2933-2952.
 39. T. X. Zhou, Y. F. Nie, "Combined hybrid approach to finite element schemes of high performance", *International Journal for Numerical Methods in Engineering*, 2001; **51**:181-202.
 40. X. P. Xie, T. X. Zhou, "Optimization of stress modes by energy compatibility for 4-node hybrid quadrilaterals", *International Journal for Numerical Methods in Engineering*, 2004; **59**:293-313.
 41. D. Mijuca, "On hexahedral finite element HC8/27 in elasticity", *Computational Mechanics*, 2004; **33**:466-480.
 42. X. P. Xie, "An accurate hybrid macro-element with linear displacements", *Communications in Numerical Methods in Engineering*, 2005; **21**:1-12.
 43. J. C. Simo, M. S. Rifai, "A class of mixed assumed strain methods and the method of incompatible modes", *International Journal for Numerical Methods in Engineering*, 1990; **29**:1595-1638.
 44. H. K. Stolarski, Y. I. Chen, "Assumed strain formulation for the four-node quadrilateral with improved in-plane bending behaviour", *International Journal for Numerical Methods in Engineering*, 1995; **38**:1287-1305.

-
45. J. Korelc, P. Wriggers, “Improved enhanced strain four-node element with Taylor expansion of the shape functions”, *International Journal for Numerical Methods in Engineering*, 1997; **40**:407–421.
 46. U. Andelfinger, E. Ramm, “EAS-elements for two-dimensional, three-dimensional, plate and shell structures and their equivalence to HR-elements”, *International Journal for Numerical Methods in Engineering*, 1993; **36**:1311–1337.
 47. S. Glaser, F. Armero, “On the formulation of enhanced strain finite elements in finite deformations”, *Engineering Computations: International Journal for Computers – Aided Engineering*, 1997; **14**:759–791.
 48. S. Klinkel, W. Wagner, “A geometrical non-linear brick element based on the EAS-method”, *International Journal for Numerical Methods in Engineering*, 1997; **40**:4529–4545.
 49. R. Piltner, “An implementation of mixed enhanced finite elements with strains assumed in Cartesian and natural element coordinates using sparse B (overline)-matrices”, *Engineering Computations*, 2000; **17**:933–949.
 50. R. Hauptmann, K. Schweizerhof, S. Doll, “Extension of the ‘solid-shell’ concept for application to large elastic and large elastoplastic deformations”, *International Journal for Numerical Methods in Engineering*, 2000; **49**:1121–1141.
 51. R. Hauptmann, K. Schweizerhof, “A systematic development of ‘solid-shell’ element formulations for linear and non-linear analyses employing only displacement degrees of freedom”, *International Journal for Numerical Methods in Engineering*, 1998; **42**:49–69.

-
52. E. P. Kasperl, R. L. Taylor, "Mixed-enhanced formulation for finite deformation axisymmetric-torsion", *International Journal for Numerical Methods in Engineering*, 2004; **60**:341–360.
 53. A. Ibrahimbegovic, E. L. Wilson, "Thick shell and solid finite elements with independent rotation fields", *International Journal for Numerical Methods in Engineering*, 1991; **31**:1393–1414.
 54. A. Ibrahimbegovic, E. L. Wilson, "A unified formulation for triangular and quadrilateral flat shell finite elements with six nodal degrees of freedom", *Communications in Applied Numerical Methods*, 1991; **7**:1–9.
 55. D. J. Allman, "A compatible triangular element including vertex rotations for plane elasticity analysis", *Computers and Structures*, 1984; **19**:1–8.
 56. A. Ibrahimbegovic, R. L. Taylor, E. L. Wilson, "A robust quadrilateral membrane finite element with drilling degrees of freedom", *International Journal for Numerical Methods in Engineering*, 1990; **30**:445–457.
 57. C. K. Choi, W. H. Lee, "Transition membrane elements with drilling freedom for local mesh refinements", *Structural Engineering and Mechanics*, 1995; **3**:75–89.
 58. M. Iura, S. N. Atluri, "Formulation of a membrane finite element with drilling degrees of freedom", *Computational Mechanics*, 1992; **9**:417–428.
 59. C. K. Choi, K. Y. Chung, T. Y. Lee, "A direct modification method for strains due to non-conforming modes", *Structural Engineering and Mechanics*, 2001; **11**:325–340.
 60. C. K. Choi, T. Y. Lee, K. Y. Chung, "Direct modification for non-conforming elements with drilling DOF", *International Journal for Numerical Methods in Engineering*, 2002; **55**:1463–1476.

-
61. N. S. Lee, K. J. Bathe, “Effects of element distortions on the performance of isoparametric elements”, *International Journal for Numerical Methods in Engineering*, 1993; **36**:3553–3576.
 62. Y. Q. Long, J. X. Li, Z. F. Long, S. Cen, “Area co-ordinates used in quadrilateral elements”, *Communications in Numerical Methods in Engineering*, 1999; **15**:533–545.
 63. Z. F. Long, J. X. Li, S. Cen, Y. Q. Long, “Some basic formulae for area co-ordinates in quadrilateral elements”, *Communications in Numerical Methods in Engineering*, 1999; **15**:841–852.
 64. A. K. Soh, Y. Q. Long, S. Cen, “Development of eight-node quadrilateral membrane elements using the area coordinates method”, *Computational Mechanics*, 2000; **25**:376–384.
 65. A. K. Soh, Z. F. Long, S. Cen, “Development of a new quadrilateral thin plate element using area coordinates”, *Computer Methods in Applied Mechanics and Engineering*, 2000; **190**:979–987.
 66. X. M. Chen, S. Cen, Y. Q. Long, Z. H. Yao, “Membrane elements insensitive to distortion using the quadrilateral area coordinate method”, *Computers and Structures*, 2004; **82**:35–54.
 67. R. H. Macneal, “A theorem regarding the locking of tapered four-noded membrane elements”, *International Journal for Numerical Methods in Engineering*, 1987; **24**:1793–1799.
 68. S. Rajendran, K. M. Liew, “A novel unsymmetric 8-node plane element immune to mesh distortion under a quadratic displacement field”, *International Journal for Numerical Methods in Engineering*, 2003; **58**:1713–1748.

-
69. E. T. Ooi, S. Rajendran, J. H. Yeo, “A 20-node hexahedron element with enhanced distortion tolerance”, *International Journal for Numerical Methods in Engineering*, 2004; **60**:2501–2530.
 70. L. B. Lucy, “A numerical approach to the testing of the fission hypothesis”, *The Astronomical Journal*, 1977; **82**:1013-1024.
 71. J. J. Monaghan, “Why particle methods work”, *SIAM Journal on Scientific and Statistical Computing*, 1982; **3**:422-433.
 72. J. J. Monaghan, “An introduction to SPH”, *Computer Physics Communications*, 1988; **48**: 89-96.
 73. L. D. Libersky, A. G. Petschek, “Smooth particle hydrodynamics with strength of materials”, *Advances in the Free-Lagrange Method, Lecture Notes in Physics*, 1990; **395**:248-257.
 74. L. D. Libersky, A. G. Petschek, T. C. Carney, J. R. Hipp, F. A. Allahdadi, “High strain Lagrange hydrodynamics: A three-dimensional SPH code for dynamic material response”, *Journal of Computational Physics*, 1993; **109**:67-75.
 75. R. F. Stellingwerf, C. A. Wingate, “Impact modeling with smooth particle hydrodynamics”, *International Journal of Impact Engineering*, 1993; **14**:707-718.
 76. S. W. Attaway, M. W. Heinstein, J. W. Swegle, “Coupling of smooth particle hydrodynamics with the finite element methods”, *Nuclear Engineering and Design*, 1994; **150**:199-205.
 77. B. Nayroles, G. Touzot, P. Villon, “Generalizing the finite element method: Diffuse approximation and diffuse elements”, *Computational mechanics*, 1992; **10**:307-318.

-
78. T. Belytschko, L. Gu, Y. Y. Lu, "Fracture and crack growth by element free Galerkin methods", *Modelling and Simulation in Materials Science and Engineering*, 1994; **2**:519-534.
 79. T. Belytschko, Y. Y. Lu, L. Gu, "Element-free Galerkin methods", *International Journal for Numerical Methods in Engineering*, 1994; **37**:229-256.
 80. Y. Y. Lu, T. Belytschko, L. Gu, "A new implementation of the element free Galerkin method", *Computer Methods in Applied Mechanics and Engineering*, 1994; **113**:397-414.
 81. T. Belytschko, Y. Y. Lu, L. Gu, M. Tabbar, "Element-free Galerkin methods for static and dynamic fracture," *International Journal of Solids and Structures*, 1995; **32**:2547-2570.
 82. T. Belytschko, Y. Y. Lu, L. Gu, "Crack propagation by element-free Galerkin methods", *Engineering Fracture Mechanics*, 1995; **51**:295–315.
 83. T. Belytschko, Y. Krongauz, D. Organ, M. Fleming, P. Krysl, "Meshless methods: An overview and recent development", *Computer methods in applied Mechanics and Engineering*, 1996; **139**:3-47.
 84. T. Belytschko, Y. Krongauz, M. Fleming, D. Organ, W. K. S. Liu, "Smoothing and accelerated computations in the element free Galerkin method", *Journal of Computational and Applied Mathematics*, 1996; **74**:111-126.
 85. T. Belytschko, M. Tabbara, "Dynamic fracture using element-free Galerkin methods", *International Journal for Numerical Methods in Engineering*, 1996; **39**:923–938.
 86. Y. V. Krongauz, "Applications of meshless methods to solid mechanics", Thesis of Doctor of Philosophy, Northwestern University, Evanston, Illinois, USA, 1996.

-
87. M. Fleming, Y. A. Chu, B. Moran, T. Belytschko, “Enriched element-free Galerkin methods for crack tip fields”, *International Journal for Numerical Methods in Engineering*, 1997; **40**:1483-1504.
 88. J. Braun, M. Sambridge, “A numerical method for solving partial differential equations on highly irregular evolving grids”, *Nature*, 1995; **376**:655-660.
 89. N. Sukumar, B. Moran, T. Belytschko, “The natural element method in solid mechanics”, *International Journal for Numerical Methods in Engineering*, 1998; **43**:839-887.
 90. N. Sukumar, B. Moran, A. Y. Semenov, V. V. Belikoy, “Natural neighbour Galerkin methods”, *International Journal for Numerical Methods in Engineering*, 2001; **50**:1–27.
 91. R. I. Sergio, O. Eugenio, C. Nestor, D. P. Facundo, “The meshless finite element method”, *International Journal for Numerical Methods in Engineering*, 2003; **58**:893-912.
 92. Y. C. Cai, H. H. Zhu, “A local search algorithm for natural neighbours in the natural element method”, *International Journal of Solids and Structures*, 2005; **42**:6059–6070.
 93. W. K. Liu, S. Jun, S. Li, J. Adee, T. Belytschko, “Reproducing kernel particle methods for structural dynamics”, *International Journal for Numerical Methods in Engineering*, 1995; **38**:1655-1679.
 94. W. K. Liu, S. Jun, Y. F. Zhang, “Reproducing kernel particle methods”, *International Journal for Numerical Methods in Fluids*, 1995; **20**:1081-1106.
 95. W. K. Liu, Y. Chen, C. T. Chang, T. Belytschko, “Advances in multiple scale Kernel Particle Methods”, *Computational Mechanics*, 1996; **18**:73-111.

96. W. K. Liu, S. F. Li, T. Belytschko, "Moving least-square reproducing kernel methods (I) Methodology and convergence", *Computer Methods in Applied Mechanics and Engineering*, 1997; **143**:113-154.
97. J. Sukky, W. K. Liu, T. Belytschko, "Explicit reproducing kernel particle methods for large deformation problems", *International Journal for Numerical Methods in Engineering*, 1998; **41**:137-166.
98. T. Zhu, J. D. Zhang, S. N. Atluri, "A local boundary integral equation (LBIE) method in computational mechanics, and a meshless discretization approach", *Computational mechanics*, 1998; **21**:223-235.
99. S. N. Atluri, J. Sladek, V. Sladek, T. Zhu, "The local boundary integral equation (LBIE) and it's meshless implementation for linear elasticity", *Computational mechanics*, 2000; **25**:180-198.
100. S. N. Atluri, T. Zhu, "A new meshless Local Petrov-Galerkin (MLPG) approach in computational mechanics", *Computational mechanics*, 1998; **22**:117-127.
101. S. N. Atluri, J. Y. Cho, H. G. Kim, "Analysis of thin beams, using the meshless local Petrov-Galerkin method, with generalized moving least squares interpolations", *Computational mechanics*, 1999; **24**:334-347.
102. S. N. Atluri, H. G. Kim, J. Y. Cho, "A critical assessment of the truly Meshless Local Petrov-Galerkin (MLPG), and Local Boundary Integral Equation (LBIE) methods", *Computational mechanics*, 1999; **24**:348-372.
103. T. Zhu, J. Zhang, S. N. Atluri, "A meshless local boundary integral equation (LBIE) method for solving nonlinear problems", *Computational mechanics*, 1998; **22**:174-186.

-
104. J. Sladek, V. Sladek, S. N. Atluri, “Local boundary integral equation (LBIE) method for solving problems of elasticity with nonhomogeneous material properties”, *Computational Mechanics*, 2000; **24**:456-462.
 105. W. Barry, “A Wachspress Meshless Local Petrov–Galerkin method”, *Engineering Analysis with Boundary Elements*, 2004; **28**:509-523.
 106. P. Radek, “Efficient cubature formulae for MLPG and related methods”, *International Journal for Numerical Methods in Engineering*, 2005; **65**:566-593.
 107. G. R. Liu, Y. T. Gu, “A point interpolation method for two-dimensional solids”, *International Journal for Numerical Methods in Engineering*, 2001; **50**:937-951.
 108. G. R. Liu, Y. T. Gu, “A local point interpolation method for stress analysis of two-dimensional solids”, *International Journal of Structural Engineering and Mechanics*, 2001; **11**:221-236.
 109. Y. T. Gu, G. R. Liu, “A local point interpolation method for static and dynamic analysis of thin beams”, *Computer Methods in Applied Mechanics and Engineering*, 2001; **190**:5515-5528.
 110. Y. X. Mukherjee, S. Mukherjee, “The boundary node method for potential problems”, *International Journal for Numerical Methods in Engineering*, 1997; **40**:797-815.
 111. Y. T. Gu, G. R. Liu, “A boundary point interpolation method for stress analysis of solids”, *Computational Mechanics*, 2002; **28**:47-54.
 112. G. R. Liu, Y. T. Gu, “A local radial point interpolation method (LRPIM) for free vibration analyses of 2-D solids”, *Journal of Sound and Vibration*, 2001; **246**:29-46.

-
113. G. R. Liu, Y. T. Gu, “A matrix triangularization algorithm for the polynomial point interpolation method”, *Computer Methods in Applied Mechanics and Engineering*, 2003; **192**:2269–2295.
 114. K. M. Liew, X. L. Chen, “Mesh-free radial point interpolation method for the buckling analysis of Mindlin plates subjected to in-plane point loads”, *International Journal for Numerical Methods in Engineering*, 2004; **60**:1861–1877.
 115. G. R. Liu, Y. T. Gu, K. Y. Dai, “Assessment and applications of point interpolation methods for computational mechanics”, *International Journal for Numerical Methods in Engineering*, 2004; **59**:1373–1397.
 116. E. Onate, S. Idelsohn, O. C. Zienkiewicz, R. L. Taylor, “A finite point method in computational mechanics. Applications to convective transport and fluid flow”, *International Journal for Numerical Methods in Engineering*, 1996; **39**:3839–3866.
 117. E. Onate, S. Idelsohn, O. C. Zienkiewicz, R. L. Taylor, C. Sacco, “A stabilized finite point method for analysis of fluid mechanics problems”, *Computer Methods in Applied Mechanics and Engineering*, 1996; **139**:315–346.
 118. E. Onate, F. Perazzo, J. Miquel, “A finite point method for elasticity problems”, *Computers and Structures*, 2001; **79**:2151–2163.
 119. N. R. Aluru, G. Li, “Finite cloud method: a true meshless technique based on a fixed reproducing kernel approximation”, *International Journal for Numerical Methods in Engineering*, 2001; **50**:2373–2410.
 120. J. J. Benito, F. Urena, L. Gavete, “Influence of several factors in the generalized finite difference method”, *Applied Mathematical Modelling*, 2001; **25**:1039–1053.

-
121. X. Z. Jin, G. Li, N. R. Aluru, "Positivity conditions in meshless collocation methods", *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:1171–1202.
 122. H. Li, Y. T. Ng, J. Q. Cheng, K. Y. Lam, "Hermite-Cloud: a novel true meshless method", *Computational Mechanics*, 2003; **33**:30-41.
 123. T. Rabczuk, T. Belytschko, S. P. Xiao, "Stable particle methods based on Lagrangian kernels", *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:1035-1063.
 124. Y. Krongauz, T. Belytschko, "Consistent pseudo-derivatives in meshless methods", *Computer Methods in Applied Mechanics and Engineering*, 1997; **146**:371-386.
 125. K. Y. Lam, Q. X. Wang, H. Li, "A novel meshless approach - Local Kriging (LoKriging) method with two-dimensional structural analysis", *Computational Mechanics*, 2004; **33**:235-244.
 126. D. L. Young, K. H. Chen, C. W. Lee, "Novel meshless method for solving the potential problems with arbitrary domain", *Journal of Computational Physics*, 2005; **209**:290–321.
 127. S. C. Li, S. C. Li, Y. M. Cheng, "Enriched meshless manifold method for two-dimensional crack modeling", *Theoretical and Applied Fracture Mechanics*, 2005; **44**:234-248.
 128. T. Belytschko, D. Organ, Y. Krongauz, "A coupled finite element-element-free Galerkin method", *Computational Mechanics*, 1995; **17**:186-195.
 129. D. Hegen, "Element-free Galerkin methods in combination with finite element approaches", *Computer Methods in Applied Mechanics and Engineering*, 1996; **135**:143-166.

-
130. A. Huerta, S. Fernandez-Mendez, “Enrichment and coupling of the finite element and meshless methods”, *International Journal for Numerical Methods in Engineering*, 2000; **48**:1615-1636.
 131. Q. Z. Xiao, M. Dhanasekar, “Coupling of FE and EFG using collocation approach”, *Advances in Engineering Software*, 2002; **33**:507-515.
 132. G. R. Liu, Y. T. Gu, “Meshless local Petrov-Galerkin (MLPG) method in combination with finite element and boundary element approaches”, *Computational Mechanics*, 2000; **26**:536-546.
 133. T. Chen, I. S. Raju, “A coupled finite element and meshless local Petrov-Galerkin method for two-dimensional potential problems”, *Computer Methods in Applied Mechanics and Engineering*, 2003; **192**:4533-4550.
 134. M. Michael, D. Suvranu, “Towards an automatic discretization scheme for the method of finite spheres and its coupling with the finite element method”, *Computers and Structures*, 2005; **83**:1429-1447.
 135. J. M. Melenk, I. Babuska, “The partition of unity finite element method: Basic theory and applications”, *Computer Methods in Applied Mechanics and Engineering*, 1996; **139**:289-314.
 136. I. Babuska, J. M. Melenk, “Partition of unity method”, *International Journal for Numerical Methods in Engineering*, 1997; **40**:727-758.
 137. T. Strouboulis, I. Babuska, K. Copps, “The design and analysis of the Generalized Finite Element Method”, *Computer Methods in Applied Mechanics and Engineering*, 2000; **181**:43-69.
 138. T. Strouboulis, K. Copps, I. Babuska, “The generalized finite element method: an example of its implementation and illustration of its performance”, *International*

-
- Journal for Numerical Methods in Engineering*, 2000; **47**:1401-1417.
139. R. Tian, G. Yagawa, "Generalized nodes and high-performance elements", *International Journal for Numerical Methods in Engineering*, 2005; **64**:2039-2071.
 140. R. Tian, G. Yagawa, H. Terasaka, "Linear dependence problems of partition of unity-based generalized FEMs", *Computer Methods in Applied Mechanics and Engineering*, 2006; **195**:4768–4782.
 141. T. J. Liszka, C. A. M. Duarte, W. W. Tworzydło, "Hp-Meshless cloud method", *Computer Methods in Applied Mechanics and Engineering*, 1996; **139**:263-288.
 142. J. T. Oden, C. A. M. Duarte, O. C. Zienkiewicz, "A new cloud-based *hp* finite element method", *Computer Methods in Applied Mechanics and Engineering*, 1998; **153**:117-126.
 143. G. Yagawa, "Node-by-node parallel finite elements: a virtually meshless method", *International Journal for Numerical Methods in Engineering*, 2004; **60**:69-102.
 144. S. Hao, H. S. Park, W. K. Liu, "Moving particle finite element method", *International Journal for Numerical Methods in Engineering*, 2002; **53**:1937-1958.
 145. S. Hao, W. K. Liu, T. Belytschko, "Moving particle finite element method with global smoothness", *International Journal for Numerical Methods in Engineering*, 2004; **59**:1007-1020.
 146. W. K. Liu, W. M. Han, H. S. Lu, S. F. Li, J. Cao, "Reproducing kernel element method. Part I: Theoretical formulation", *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:933–951.
 147. S. F. Li, H. S. Lu, W. M. Han, W. K. Liu, D. C. Simkins, "Reproducing kernel element method Part II: Globally conforming I^m / C^n hierarchies", *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:953–987.

-
148. H. S. Lu, S. F. Li, D. C. Simkins, W. K. Liu, J. Cao, “Reproducing kernel element method Part III: Generalized enrichment and applications”, *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:989–1011.
 149. D. C. Simkins, S. F. Li, H. S. Lu, W. K. Liu, “Reproducing kernel element method. Part IV: Globally compatible C^n ($n \geq 1$) triangular hierarchy”, *Computer Methods in Applied Mechanics and Engineering*, 2004; **193**:1013–1034.
 150. J. Dolbow, T. Belytschko, “An Introduction to Programming the Meshless Element-free Galerkin Method”, *Archives of Computational Mechanics in Engineering*, 1998; **5**:207–241.
 151. S. P. Timoshenko, J. N. Goodier, “Theory of Elasticity”, 3rd Edn, McGraw-Hill, New York, 1970.
 152. B. M. Irons, “Numerical integration applied to finite element methods”, *Proceedings of the conference on Use of Digital Computers in Structural Engineering*, University of Newcastle, 1966.
 153. B. M. Irons, A. Razzaque, “Experience with the patch test for convergence of finite elements”, In “The Mathematical Foundations of Finite Element with Application to Partial Differential Equations”, A. K. Aziz (editor), Academic Press, New York, 1972; 557–587.
 154. O. C. Zienkiewicz, R. L. Taylor, “The Finite Element Method”, 4th edn., McGraw-Hill, London, New York, 1989.
 155. R. D. Cook, D. S. Malkus, M. E. Plesha, “Concepts and Applications of Finite Element Analysis”, 3rd edn., John Wiley & Sons Inc, New York, 1989.

-
156. K. Y. Sze, “On immunizing five-beta hybrid-stress element models from trapezoidal locking in practical analyses”, *International Journal for Numerical Methods in Engineering*, 2000; **47**:907-920.
 157. R. H. Macneal, “A theorem regarding the locking of tapered four-noded membrane elements”, *International Journal for Numerical Methods in Engineering*, 1987; **24**:1793–1799.
 158. A. E. Ouatouati, D. A. Johnson, “A new approach for numerical modal analysis using the element-free method”, *International Journal for Numerical Methods in Engineering*, 1999; **46**:1–27.
 159. J. X. Zhou, H. Y. Zhang, L. Zhang, “Reproducing kernel particle method for free and forced vibration analysis”, *Journal of Sound and Vibration*, 2005; **279**:389-402.
 160. K. M. Liew, T. Y. Ng, X. Zhao, J. N. Reddy, “Harmonic reproducing kernel particle method for free vibration analysis of rotating cylindrical shells”, *Computer methods in applied mechanics and engineering*, 2002; **191**:4141–4157.
 161. H. Li, Q. X. Wang, K. Y. Lam, “Development of a novel meshless Local Kriging (LoKriging) method for structural dynamic analysis”, *Computer methods in applied mechanics and engineering*, 2004; **193**:2599–2619.
 162. E. D. Bel, P. Villon, P. H. Bouillard, “Forced vibrations in the medium frequency range solved by a partition of unity method with local information”, *International Journal for Numerical Methods in Engineering*, 2005; **62**:1105–1126.
 163. T. Nagashima, “Node-by-node meshless approach and its application to structural analyses”, *International Journal for Numerical Methods in Engineering*, 1999; **46**:341–385.

-
164. C. A. Brebbia, J. C. F. Telles, L. C. Wrobel, “Boundary Element Techniques”, Springer, Berlin, 1984.
 165. H. Lamb, “On Waves in an Elastic Plate”. *Proceedings of the Royal Society of London (Series A)*, 1917; **93**:114-128.
 166. R. Mindlin, “Waves and Vibrations in Isotropic Elastic Plates”, *Proceedings of the First Symposium On Naval Structural Mechanics*, Pergamon Press, 1960; 199-232.
 167. P. C. Kohnke, “Large deflection analysis of frame structures by fictitious forces”, *International Journal for Numerical Methods in Engineering*, 1978; **12**:1279–1294.
 168. Y. Basar, M. Itskov, “Constitutive model and finite element formulation for large strain elasto-plastic analysis of shells”, *Computational Mechanics*, 1999; **23**:466–481.
 169. C. Stephane, Z. Thomas, “Object-oriented nonlinear finite element programming: a primer”, *Advances in Engineering software*, 2001; **32**:611-628.
 170. J. Y. Liu, J. Z. Hong, “Geometric nonlinear formulation and discretization method for a rectangular plate undergoing large overall motions”, *Mechanics Research Communications*, 2005; **32**:561–571.
 171. W. K. Liu, S. Jun, “Multiple-scale reproducing kernel particle methods for large deformation problems”, *International Journal for Numerical Methods in Engineering*, 1998; **41**:1339–1362.
 172. S. Jun, W. K. Liu, T. Belytschko, “Explicit Reproducing Kernel Particle Methods for large deformation problems”, *International Journal for Numerical Methods in Engineering*, 1998; **41**:137–166.

-
173. S. Li, W. Hao, W. K. Liu, “Numerical simulations of large deformation of thin shell structures using meshfree methods”, *Computational Mechanics*, 2000; **25**:102–116.
 174. J. S. Chen, C. H. Pan, C. T. Wu, W. K. Liu, “Reproducing kernel particle methods for large deformation analysis of non-linear structures”, *Computer Methods in Applied Mechanics and Engineering*, 1996; **139**:195–227.
 175. K. M. Liew, K. Y. Ng, Y. C. Wu, “Meshfree method for large deformation analysis - a reproducing kernel particle approach”, *Engineering Structures*, 2002; **24**:543–551.
 176. J. N. Reddy, “An introduction to nonlinear finite element analysis”, Oxford University Press, Oxford, UK, 2004.
 177. E. T. Ooi, “Unsymmetric Finite Elements for Mesh-Distortion-Tolerant Performance”, PhD thesis of Nanyang Technological University, 2006.
 178. R. H. Macneal, “Finite elements: their Design and performance”, Marcel Dekker, New York, 1994.
 179. S. Chandra, G. Prathap, “A field-consistent formulation for the eight-noded solid finite element”, *Computers and Structures*, 1988; **33**:345-355.
 180. I. Babuska, R. Narasimhan, “The Babuska–Brezzi condition and the patch test: an example”, *Computer Methods in Applied Mechanics and Engineering*, 1997; **140**:183–199.
 181. D. Chapelle, K. J. Bathe, “The inf–sup test”, *Computers and Structures*, 1993; **47**:537–545.
 182. K. J. Bathe, “Finite Element Procedures in Engineering Analysis”, 2nd edn., Prentice-Hall, Englewood Cliff, NJ, 1996.

-
- 183 . K. J. Bathe, D. Hendriana, F. Brezzi, G. Sangalli, “Inf–sup test of upwinding methods”, *International Journal for Numerical Methods in Engineering*, 2000; **48**:745–760.