Development of flat-top partition of unity based high-order discontinuous deformation analysis

Liu, Xiaoying

2018

Liu, X. (2018). Development of flat-top partition of unity based high-order discontinuous deformation analysis. Doctoral thesis, Nanyang Technological University, Singapore.

<http://hdl.handle.net/10356/73508>

<https://doi.org/10.32657/10356/73508>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

SINGAPORE

**DEVELOPMENT OF FLAT-TOP PARTITION OF
UNITY BASED HIGH-ORDER DISCONTINUOUS
DEFORMATION ANALYSIS**

LIU XIAOYING

SCHOOL OF CIVIL AND ENVIRONMENTAL ENGINEERING

2018

**DEVELOPMENT OF FLAT-TOP PARTITION OF
UNITY BASED HIGH-ORDER DISCONTINUOUS
DEFORMATION ANALYSIS**

LIU XIAOYING

School of Civil and Environmental Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfillment of the requirement for the degree of
Doctor of Philosophy

2018

ACKNOWLEDGEMENT

First of all, I would like to express my sincere gratitude to my supervisor, Associate Professor Zhao Zhiye. It is my great pleasure to accomplish the Ph.D study with his encouragement and guidance. As a professional and easy-going person, Prof. Zhao taught me not only the way to think about research problems, but also the way to communicate with others. I know I will benefit from this not only in the Ph.D period, but also in my lifetime. I am so grateful to enjoy the comfortable and supportive research environment he provided in the past four years.

I would like to extend my thanks to Dr. An Xinmei, who provided valuable suggestions in the initial stage of my research work.

I am thankful to Dr. Nie Wen, who has provided me the software of 2D-DDA to help my reserch work and helped me solving problems with great patient.

My thanks are also given to my colleagues Mr. Xiao Fei and Dr. Choo Lingqian for their help in my research life.

I would like to extend my thanks to all the current and former research fellows in our research group, who have helped me in my reasearch work.

I give my thanks to Dr. Zhang Yang and Professor Wu Aiqing, who provided me the initial code of high-order approximation.

I own my thanks to Dr. Shi Genhua, who gave me the original DDA code and valuable suggestions about DDA study.

I am thankful to Nanyang Technological University for provinding me the opportunity and finantial support to carry on with my research journey.

Last but not least, I own my thanks to my parents, my elder brother and my husband, for their unconditional support in my life.

TABLE OF CONTENTS

ACKNOWLEDGEMENT	i
TABLE OF CONTENTS	iii
SUMMARY	viii
LIST OF PUBLICATIONS	x
LIST OF TABLES	xi
LIST OF FIGURES	xiii
LIST OF SYMBOLS	xix
CHAPTER 1 INTRODUCTION	1
1.1 Background	1
1.2 Objective and scope	3
1.3 Organization of thesis	4
CHAPTER 2 LITERATURE REVIEW	6
2.1 Introduction	6
2.2 Development of numerical analysis method	6
2.2.1 Continuum-based numerical methods	7
2.2.2 Discontinuum-based numerical methods	12
2.2.3 Numerical Manifold Method	16
2.3 Linear dependence problem in high-order PU based method	18
2.4 Flat-top PU based high-order DDA	21
2.5 Relationship between flat-top PU based DDA, DDA and NMM	23
2.6 Advantages and limitations of flat-top PU based high-order DDA method	24
CHAPTER 3 FLAT-TOP PU BASED HIGH-ORDER APPROXIMATION FOR CONTINUUM PROBLEM	26
3.1 Introduction	26

3.2 Construction of flat-top PU mesh within continuum	26
3.2.1 One-dimensional flat-top PU mesh	26
3.2.2 Two-dimensional flat-top PU mesh	29
3.2.3 Three-dimensional flat-top PU mesh	34
3.3 High-order approximation within the framework of flat-top PU method	45
3.3.1 Global displacement approximation	46
3.3.2 Constitutive equations	49
3.3.3 Establishment of global equilibrium equations	50
3.4 Numerical analysis procedure	71
3.5 Numerical examples	72
3.5.1 Simply-supported beam	73
3.5.2 MacNeal and Harder's slender beam	77
3.5.3 Curved cantilever beam	79
3.5.4 Twisted cantilever beam	81
3.5.5 Beam-to-column connection	83
3.5.6 Cook's problem	86
3.5.7 Thin plate	91
3.5.8 Table-like structure	94
3.6 Conclusions	96
CHAPTER 4 LOCAL REFINEMENT WITH FLAT-TOP PU MESH	97
4.1 Introduction	97
4.2 Basic Theory for proof of linear independence	97
4.2.1 Equivalent p -version element	98
4.2.2 Rank deficiency counting approach	99
4.3 Local refinement and proof of linear independence for 1D case	100

4.3.1 Element types in regularly patterned flat-top PU mesh	101
4.3.2 First basic local refinement case	103
4.3.3 Second basic local refinement case.....	104
4.3.4 Third basic local refinement case	105
4.4 Local refinement and proof of linear independence for 2D case.....	105
4.4.1 Element types in regularly patterned flat-top PU mesh	106
4.4.2 First basic local refinement case	107
4.4.3 Second basic local refinement case.....	112
4.4.4 Third basic local refinement case	116
4.4.5 Local refinement for PU elements	117
4.5 numerical examples	119
4.5.1 Infinite plate with a circular hole	119
4.5.2 L-shaped structure.....	124
4.6 Conclusions.....	127
CHAPTER 5 FLAT-TOP PU BASED HIGH-ORDER DDA METHOD.....	129
5.1 Introduction.....	129
5.2 Construction of flat-top PU mesh with discontinuity	130
5.2.1 One-dimensional case	130
5.2.2 Two-dimensional case	133
5.3 Contact algorithm within the framework of flat-top PU method.....	136
5.3.1 Contact algorithm in original DDA method	136
5.3.2 Contact algorithm for one-dimensional flat-top PU mesh.....	138
5.3.3 Contact algorithm for two-dimensional flat-top PU mesh.....	142
5.4 Numerical examples.....	146
5.4.1 Free dropping	146

5.4.2 Swing of pendulum	150
5.4.3 Collision	153
5.4.4 Sliding of slope	157
5.5 Conclusions.....	161
CHAPTER 6 FLAT-TOP PU BASED HIGH-ORDER DDA FOR STRESS WAVE PROPAGATION	162
6.1 Introduction.....	162
6.2 Non-reflective boundary	162
6.3 Numerical examples.....	168
6.3.1 Long elastic bar	168
6.3.2 Infinite plate with a circular hole	184
6.3.3 Half infinite plate	186
6.4 Conclusions.....	190
CHAPTER 7 PRELIMINARY STUDY OF CRACK PROPAGATION WITH FLAT-TOP PU BASED HIGH-ORDER DDA	192
7.1 Introduction.....	192
7.2 Criteria for crack initiation.....	193
7.2.1 Crack propagation mode	193
7.2.2 Failure Criteria	193
7.2.3 Initiation of crack within flat-top PU mesh	194
7.3 Algorithm for crack propagation	197
7.3.1 Two basic cases for crack propagation within flat-top PU mesh..	198
7.3.2 Different combinations of two basic cases	199
7.4 Numerical analysis procedure.....	204
7.5 Numerical examples.....	204
7.5.1 Spalling of long elastic bar	205

7.5.2 Square plate with internal crack.....	208
7.5.3 Cantilever beam with edge crack	211
7.6 Conclusions.....	213
CHAPTER 8 CONCLUSIONS AND RECOMMENDATIONS	215
8.1 Conclusions.....	215
8.2 Recommendations for future work	217
REFERENCES.....	219

SUMMARY

In this thesis, the flat-top partition of unity (PU) based high-order Discontinuous Deformation Analysis (DDA) is developed and applied in simulating different problems. In the proposed method, the continuous blocks in the problem domain are discretized by the flat-top PU mesh, and the interactions between blocks is simulated with the improved contact algorithm. For each flat-top PU unit in the mesh, high-order polynomial is assigned as local approximation, where the coefficients of monomials are denoted as unknowns. The global approximation can be obtained by multiplying the local approximation with the corresponding shape functions. With the principle of virtual work and simplex integration, the global equilibrium equations are established and solved. Thus, the displacement field and stress/strain field can be obtained. In flat-top PU based high-order DDA method, since the flat-top PU mesh and high-order approximation are utilized instead of discretizing the problem domain into numbers of small blocks, only the main discontinuities need to be considered. Therefore, the time-consuming iteration of contact detection, adding and removing stiff springs can be reduced.

The construction of regularly patterned flat-top PU mesh is introduced based on the traditional PU mesh, which is easy to understand and straightforward to implement. In order to simulate the stress concentration/singularity and crack propagation problems with high cost-efficiency, the local refinement of flat-top PU mesh is introduced and its property of linear independence is proven from element level based on the equivalent p -version element and rank deficiency counting approach.

During the generation of numerical model for discontinuous problem domain, in order to avoid ill-conditioned stiffness matrix in global equilibrium equations, modification for the numerical model is implemented based on the concept of percentage of effective area. During the numerical simulation, the contact algorithm in original DDA method is improved by determining the exact contact position. Therefore, the energy consumption is more accurate, especially for the collision problem.

The improved formula for non-reflective boundary is coupled with flat-top PU based high-order DDA method to simulate stress wave propagation within infinite

problem domain, where the numerical accuracy can be improved by deriving the analytical velocity of dashpots, instead of calculating the velocity with finite difference method.

Based on the crack propagation modes, the tension failure criterion and the Mohr-Coulomb failure criterion are adopted to determine the crack initiation in flat-top PU based high-order DDA method. Within each time step, the criteria are applied at every node in the flat-top PU mesh. In order to avoid generating new unknowns and determining their initial values artificially, it is assumed that the crack propagates along the element sides. Correspondingly, the treatment of PU elements after crack propagation is introduced.

Numerical examples are presented and analyzed regarding to each development of the flat-top PU based high-order DDA method.

LIST OF PUBLICATIONS

Journal

1. Liu, X. Y., Zhao, Z. Y., An, X. M. and Jiao, Y. Y. (2018), “Local Refinement of flat-top partition of unity based high-order approximation”, International Journal for Numerical Methods in Engineering, (Under 2nd round revision).
2. Liu, X. Y., Zhao, Z. Y. and Ma, S. Q. (2018), “Simulating stress wave with flat-top partition of unity based high-order discontinuous deformation analysis”, Engineering Analysis with Boundary Elements, (Accepted).
3. Liu, X. Y. and Zhao, Z. Y. (2018), “Dynamic analysis with flat-top partition of unity-based discontinuous deformation analysis”, Computers and Geotechnics, Vol. 98, pp. 35-47.
4. He, L., An, X. M., Liu, X. Y., Zhao, Z. Y. and Yang, S. Q. (2015). “Augmented Numerical Manifold Method with implementation of flat-top partition of unity”, Engineering Analysis with Boundary Elements, Vol. 61, pp. 153-171.
5. An, X. M., Liu, X. Y., Zhao, Z. Y. and He, L. (2014), “Proof of linear independence of flat-top PU-based high-order approximation”, Engineering Analysis with Boundary Elements, Vol. 44, pp. 104-111.

Conference

1. Liu, X. Y., Zhao, Z. Y., An, X. M. and Wu, A. Q. (2015), “Preliminary study on three-dimensional flat-top partition of unity based high-order DDA”, Proceedings of 12th International Conference on Analysis of Discontinuous Deformation, Wuhan, China.
2. Liu, X. Y., An, X. M., He, L. and Zhao, Z. Y. (2014), “Development of three-dimensional high-order discontinuous deformation analysis method”, Proceedings of 8th Asian Rock Mechanics Symposium, Sapporo, Japan.

LIST OF TABLES

Table 3.1 Sub-matrices for flat-top PU based high-order approximation	66
Table 3.2 A brief summary of the examples	72
Table 3.3 Vertical displacement at the center of the simply-supported beam	77
Table 3.4 Normalized numerical results for beam tip deflection.....	79
Table 3.5 Normalized numerical results for displacement at free end	81
Table 3.6 Normalized numerical results for displacement at free end	83
Table 3.7 Horizontal displacement at top of the column (U_x) and vertical displacement at the free end of the beam (U_z)	86
Table 3.8 Normalized numerical results for displacement at point A from Model 187	
Table 3.9 Sub-matrices generated by each element in Model 2	90
Table 3.10 Normalized numerical results for displacement at point A from Model 2	91
Table 3.11 Error of minimum principle stress $\sigma_{\min}$ at point B (unit: %).....	91
Table 3.12 Maximum displacement ($U_{z\max}$) at the center of the plate	93
Table 3.13 Displacement U_z at the center of the thin plate.....	95
Table 4.1 Properties for three different types of elements in one-dimensional case	102
Table 4.2 Properties for different types of elements in two-dimensional case ...	108
Table 4.3 Error of σ_x at point A and number of DOFs for three different models with different local approximation degree p	122
Table 4.4 Displacement U and von Mises stress σ_v at Point A for different numerical models	126
Table 5.1 Four types of penetration in one-dimensional case	141
Table 5.2 Nine types of penetration in two-dimensional case	147

Table 5.3 Maximum and minimum percentages of effective area and period T for
different cases151

LIST OF FIGURES

Figure 2.1 Different mesh type for DEM.....	8
Figure 2.2 FEM mesh for problem domain with irregular boundary (Lo, 1985)	9
Figure 2.3 GFEM mesh independent of the geometry for problem domain (Strouboulis et al., 2001).....	10
Figure 2.4 Representation of contact in two-dimensional DEM (Jing, 2003)	13
Figure 2.5 A simple scheme for two-dimensional simplex integration (Shi, 1996)	15
Figure 2.6 The generation of manifold elements in NMM (Ma et al., 2009)	17
Figure 2.7 An element in traditional PU mesh	19
Figure 2.8 Construct flat-top PU mesh by adding flat-top regions into PU mesh (after An et al., 2014)	21
Figure 2.9 Generation of numerical model for flat-top PU based high-order DDA	22
Figure 2.10 Relationship between flat-top PU based DDA, DDA and NMM	24
Figure 3.1 Construction of flat-top PU mesh in one-dimensional space	28
Figure 3.2 Generation of one-dimensional numerical model based on flat-top PU mesh	29
Figure 3.3 Traditional 4-node PU mesh and the corresponding shape function.....	31
Figure 3.4 Construction of flat-top PU mesh and the shape function for a complete flat-top PU unit in two-dimensional space.....	32
Figure 3.5 Generation of two-dimensional numerical model based on flat-top PU mesh	34
Figure 3.6 Traditional hexahedral PU mesh and the shape function contour	36
Figure 3.7 Construction of flat-top PU mesh in three-dimensional space	39
Figure 3.8 The external and internal structure of a complete flat-top PU unit	39
Figure 3.9 A flat-top element and its shape function contour.....	40

Figure 3.10 Elemental supports for a blue-colored PU element and its shape function contour derived according to one elemental support.....	41
Figure 3.11 Elemental supports for an orange-colored PU element and its shape function contour derived according to one elemental support.....	42
Figure 3.12 Elemental supports for a white-colored PU element and its shape function contour derived according to one elemental support.....	44
Figure 3.13 Generation of three-dimensional numerical model based on flat-top PU mesh	45
Figure 3.14 Deriving global displacement approximation in three-dimensional case	48
Figure 3.15 A simply-supported beam subjected to uniformly distributed loading on the top surface	73
Figure 3.16 Numerical model and results generated by one-element mesh	75
Figure 3.17 Numerical model and results generated by three-element mesh	77
Figure 3.18 A slender cantilever beam and two numerical models	78
Figure 3.19 A curved cantilever beam and two numerical models.....	81
Figure 3.20 A pre-twisted cantilever beam and two numerical models	83
Figure 3.21 A beam-to-column connection and its numerical model.....	84
Figure 3.22 Numerical results for horizontal displacement U_x of the column and vertical displacement U_z of the beam.....	85
Figure 3.23 Configuration and numerical models for Cook's problem.....	87
Figure 3.24 A thin plate clamped at its four edges and subjected to a point loading	92
Figure 3.25 Vertical displacement along a central line and contour for whole plate	93
Figure 3.26 A table-like structure and its numerical models	95
Figure 4.1 White-colored PU element and its equivalent p -version element	98
Figure 4.2 Different element types in a regularly patterned flat-top PU mesh.....	101

Figure 4.3 Nodes distribution in equivalent p -version element for element e_2	102
Figure 4.4 Local refinement within Type I element	103
Figure 4.5 Local refinement within Type II element	104
Figure 4.6 Local refinement within Type III element	105
Figure 4.7 Assemblage sequence and element types in a regularly patterned flat-top PU mesh	107
Figure 4.9 Flat-top PU mesh for second basic local refinement case	113
Figure 4.10 Nodes distribution for two different Types of elements	115
Figure 4.11 Flat-top PU mesh for third basic local refinement case	117
Figure 4.12 Local refinement for PU elements	118
Figure 4.13 Simplified model for an infinite plate with a circular hole subjected to uniformly distributed tension force	119
Figure 4.14 Three different numerical models	121
Figure 4.15 σ_x along line AB for Models 2 and 3 with different degree p	123
Figure 4.16 Contours for σ_x of Model 3 ($p = 6$) and analytical solution	123
Figure 4.17 Basic information for an L-shaped domain	124
Figure 4.18 Four different numerical models	125
Figure 4.19 Relationship between von Mises stress σ_v and unit size at Point A ...	127
Figure 5.1 Generation of numerical model through intersecting flat-top element with discontinuity	131
Figure 5.2 Generation of numerical model through intersecting PU element with discontinuity	133
Figure 5.3 Intersecting flat-top PU mesh with boundary and discontinuity	134
Figure 5.4 Optimization of elements in Block 1 and Block 2	135
Figure 5.5 Numerical model for discontinuous problem domain	135
Figure 5.6 Three different contact types in original DDA method (Shi, 1988)	136

Figure 5.7 Contact between convex angle and concave angle (Shi, 1988).....	137
Figure 5.8 Contact algorithm in one-dimensional flat-top PU mesh	139
Figure 5.9 Contact algorithm in two-dimensional flat-top PU mesh.....	143
Figure 5.10 Numerical model for a free dropping block	148
Figure 5.11 Displacement U_y and velocity V_y for different length ratio l_r and time step Δt	150
Figure 5.12 Generation of numerical model for a pendulum.....	151
Figure 5.13 Displacement for the pendulum with different length ratio l_r and time step Δt	152
Figure 5.14 Problem domain for two collision blocks.....	153
Figure 5.15 Generation of numerical model for two collision blocks	154
Figure 5.16 Displacement U_x and velocity V_x for Block 1 and Block 2	156
Figure 5.17 Generation of numerical model for slope analysis	158
Figure 5.18 Displacement (U_x, U_y) and velocity (V_x, V_y) for Block 1.....	160
Figure 6.1 Non-reflective boundary in numerical model.....	163
Figure 6.2 Numerical model of stress wave propagation within an elastic bar	169
Figure 6.3 Stress waveforms at Point A and Point B in standard numerical model	169
Figure 6.4 Attenuation of peak pressure in standard numerical model	170
Figure 6.5 Peak pressure attenuation for different local approximation degree p and numbers of flat-top PU units.....	171
Figure 6.6 Peak pressure attenuation and waveform at Point A for different numbers of flat-top PU units and local approximation degree p	173
Figure 6.7 Peak pressure attenuation with different time step Δt	175
Figure 6.8 Waveform at Point A for different time step Δt	175
Figure 6.9 Peak pressure attenuation with different length ratio l_r	176

Figure 6.10 Waveform at Point A and Point B for different length ratio l_r	177
Figure 6.11 Relationship between Young's modulus E and P -wave velocity V_P ..	178
Figure 6.12 Error of peak pressure at Point B with different wave length l_w	180
Figure 6.13 Reflection and transmission of stress wave at the interface of two different materials	181
Figure 6.14 Reflection of stress wave under different boundary conditions at the right end	184
Figure 6.15 Numerical model for stress wave propagation within infinite plate...	185
Figure 6.16 Incident P-wave with peak pressure equal to $3MPa$	185
Figure 6.17 Peak pressure ratio of numerical results and analytical solution.....	186
Figure 6.18 Numerical model for stress wave propagation within half infinite plate	187
Figure 6.19 Waveform at Point B within first $0.003s$	189
Figure 6.20 Attenuation of second peak pressure along the observation line AC .	189
Figure 6.21 Distribution of stress σ_y along the observation line AC when $t = 0.003s$	190
Figure 7.1 Three basic crack propagation modes	193
Figure 7.2 Sketch for Mohr-Coulomb failure criterion	194
Figure 7.3 A flat-top element with four nodes.....	195
Figure 7.4 Global and local coordinate systems	195
Figure 7.5 The first basic case of crack propagation	198
Figure 7.6 The second basic case of crack propagation.....	199
Figure 7.7 Elemental supports for a white-colored PU element not linked to any crack	200
Figure 7.8 Four cases in Group 1	201
Figure 7.9 Three cases in Group 2	202

Figure 7.10 Four cases in Group 3	203
Figure 7.11 Four cases in Group 4	203
Figure 7.12 A long elastic bar subjected to compressive P -wave	205
Figure 7.13 Spalling of elastic bar with different frequencies of incident P -wave	206
Figure 7.14 Velocity at the right end of the elastic bar	207
Figure 7.15 Velocity at the right end of the left-side intact block	207
Figure 7.16 Basic information for square plate and flat-top PU mesh	209
Figure 7.17 Crack propagation path under uniaxial loading with different angle θ	210
Figure 7.18 Numerical model for a square plate with an internal crack subjected to biaxial uniformly distributed loading.....	210
Figure 7.19 Crack propagation path under biaxial uniformly distributed loading	211
Figure 7.20 Basic information for cantilever beam and numerical model.....	212
Figure 7.21 Crack propagation path.....	212

LIST OF SYMBOLS

Symbol	Description	unit
e_i	Element	-
u_i	Flat-top PU unit	-
w_i	Shape function	-
(u_x, u_y, u_z)	Local displacement	m
(U_x, U_y, U_z)	Global displacement	m
p	Local approximation degree	-
$[T]$	Matrix for monomials	-
$[T']$	Matrix for monomials multiplied by shape function	-
$[D]$	Vector for unknowns	-
σ	Stress	Pa
ε	Strain	-
$[E]$	Elastic stiffness matrix	Pa
Π	Potential energy	J
$[K]$	Stiffness matrix in global equilibrium equations	-
$[F]$	Force matrix in global equilibrium equations	-
t	Time	s
Δt	Time step	s
ρ	Density	kg/m^3

$[V]$	Velocity	m/s
RD	Rank deficiency	-
E	Young's modulus	Pa
ν	Poisson's ratio	-
R	Radius	m
(r, θ)	Polar coordinates	$(m, ^\circ)$
A_{pe}	Percentage of effective area	-
A_e	Effective area	m^2
d	Distance	m
l	Length	m
l_r	Length ratio	-
g	Gravitational acceleration	m/s^2
T	Period	s
(V_P, V_S)	P -wave and S -wave velocities	m/s
(c_n, c_t)	Normal and tangential damping coefficients	$kg/(m \cdot s)$
(c, φ)	Cohesion and internal friction angle	$(Pa, ^\circ)$

CHAPTER 1 INTRODUCTION

1.1 Background

In order to accurately assess the safety and stability of civil engineering projects before construction, it is very important for engineers to carry out the corresponding mechanical analysis, considering the given initial condition, boundary condition and material property. Therefore, it is indispensable for researchers to develop an efficient method, which can provide accurate and reliable prediction of the mechanical behaviors for problem domain concerned. Thus, mechanical assessment and construction design can be made, accordingly, for the civil engineering project.

Actually, it is very difficult to analyze the mechanical behaviors of the problem domain due to its complexity and uncertainty. For example, for the rock mass, despite the material inhomogeneity caused by the random distribution of different minerals, the appearance of discontinuities, such as joints, fractures and faults, also leads to more difficulties in obtaining an accurate solution efficiently.

In early days, there were mainly two ways to solve the mechanical analysis problems. The first one was to derive the analytical solutions through establishing and solving strict differential or partial differential equations, according to the initial condition, boundary condition and material property. Analytical solutions were obtained for a limited number of problems with simple geometry, boundary and material conditions, such as the deformation and stress distribution in a shaped problem domain (Timoshenko and Goodier, 1951) and stress wave propagation within a long elastic bar (Graff, 1975). However, in reality, the actual situations of most problems are complicated, such as irregularly shaped problem domain (Stead et al., 2006), complicated stress-strain relationship (Pietruszczak and Mróz, 1981), interaction between solid and fluid (Jing et al., 2001) and mechanical behavior of materials under various construction conditions (Kim et al., 1999; Wu et al., 2004; Ning et al., 2011a). It is almost impossible to derive the analytical solutions in these complicated situations. Another way was to carry out the experiments in laboratory, which can verify the relationship between given conditions and mechanical behaviors (Cook, 1970; Dieterich, 1979; Cai and Liu, 2009; Liu et al., 2013).

However, the expense was high in order to construct equipment and prepare samples.

Therefore, along with the development of computer science, various numerical analysis methods were proposed and developed to simulate a large number of problems with huge computation over the last 50 years, which can overcome the shortcomings associated with deriving analytical solutions and carrying out experiments.

Finite Difference Method (FDM) was firstly proposed by utilizing regular mesh to discretize the problem domain and substituting differences between adjacent points for partial derivatives (Courant et al., 1952; Liszka and Orkisz, 1980; Cui, 2009). This theory made FDM easy to understand and implement. After that, another two main continuum-based numerical methods, Finite Element Method (FEM) (Zienkiewicz and Taylor, 2000; Onate et al., 2008; Liu et al., 2009) and Boundary Element Method (BEM) (Li et al., 1985; Grilli et al., 1989; Portela et al., 1992), were proposed successively, and FEM has become one of the most popular methods currently. Furthermore, the Embedded Finite Element Method (EFEM) was developed to model discontinuities by allowing the jump in displacement field (Dvorkin et al., 1990; Rabczuk, 2012). And the eXtended Finite Element Method (XFEM) was proposed to simulate crack propagation through enriching the elements cut by the crack (Sukumar et al., 2000; Bergara et al., 2017).

However, researchers still hoped to propose a numerical method, which was capable of simulating the interactions between blocks. The development for discontinuum domain analysis includes mainly two methods, the Distinct Element Method (DEM) (Cundall, 1971, 1988; Cundall and Hart, 1985) and the Discontinuous Deformation Analysis (DDA) method (Shi and Goodman, 1985; Shi, 1988, 2002), which aimed at simulating large displacement and large deformation of discontinuous system, such as sliding, detaching and rejoining. Compared with the explicit DEM, the implicit DDA is more accurate, stable and efficient in satisfying the requirement of equilibrium. In the original DDA method, the displacement was approximated by the first-order polynomial functions. Therefore, in order to increase the accuracy of DDA method, researchers derived the formula

for high-order polynomial approximations (Beyabanaki et al., 2010; Liu, 2013; Wu et al., 2014) or discretized the blocks into smaller ones with artificial joints (Grayeli and Mortazavi, 2006; Beyabanaki et al., 2009; Ning et al., 2011b).

The Numerical Manifold Method (NMM) (Shi, 1995; Chen et al., 1998; Ma et al., 2009) was developed more recently. NMM can be considered as a generalized FEM and DDA method. It has two independent sets of meshes, the mathematical cover and the physical cover, which discretize the blocks into manifold elements. Although NMM is still within the framework of first-order polynomial approximation, it can be more accurate in describing stress distributions within a large block as compared with the first-order DDA. However, since NMM adopts the partition of unity (PU) method to formulate its shape functions, linear dependence problem appears when increasing the degree of displacement functions (Melenk and Babuška, 1996; Tian et al., 2006; An et al., 2011).

In this thesis, the flat-top PU based high-order DDA method is established and developed with the following advantages: 1) it can increase the numerical accuracy through improving the degree of displacement approximation function and discretizing the continuous problem domain with flat-top PU mesh; 2) it can simulate the interaction between blocks with the improved contact algorithm; 3) with the construction of flat-top regions in the PU mesh, it can avoid the linear dependence problem.

1.2 Objective and scope

The purpose of this research work is to develop the flat-top PU based high-order DDA method. So that it can discretize the continuous problem domain with flat-top PU mesh, approximate the displacement field with high-order polynomial; meanwhile, the interaction between blocks can be simulated with the improved contact algorithm, which is initially proposed in DDA method (Shi, 1988). Specifically, the research scope can be summarized as follows:

1. Construct the one- to three-dimensional regularly patterned flat-top PU mesh based on the traditional PU mesh, and incorporate the two- and three-dimensional regularly patterned flat-top PU mesh with high-order DDA method;

2. Construct the one- and two-dimensional locally refined flat-top PU mesh, prove the property of linear independence for each case, and incorporate the two-dimensional locally refined flat-top PU mesh with high-order DDA method;
3. Improve the accuracy of contact algorithm in the original DDA method by adding stiff spring at the exact contact position, so that the error of energy loss can be reduced, especially in the case of collision;
4. Extend the flat-top PU based high-order DDA method to simulate stress wave propagation within infinite problem domain, and carry out detailed parameter study to obtain an insight into stress wave propagation;
5. Establish the crack propagation algorithm for the flat-top PU mesh according to the tension failure criterion and the Mohr-Coulomb failure criterion;

1.3 Organization of thesis

The thesis is comprised of eight chapters in total. The main content for each chapter is given as follows:

Chapter 1: The introduction chapter includes the background, objective and work scope of this research study.

Chapter 2: A brief literature review is given about the development and application of several main numerical methods and the flat-top PU method. By comparing with other numerical methods, the advantages and disadvantages of flat-top PU based high-order DDA method is summarized.

Chapter 3: Based on the traditional PU mesh system, the construction of regularly patterned flat-top PU mesh in one- to three-dimensional space is introduced. High-order polynomial approximation within the framework of flat-top PU mesh is derived in detail. Moreover, the numerical analysis procedure is summarized, and several numerical examples concerning about displacement/stress field within continuous problem domain are analyzed.

Chapter 4: The theory of local refinement for flat-top PU mesh in one- and two-dimensional space is presented, and the property of linear independence is proven from element level for each case. The code is written for two-dimensional locally refined flat-top PU mesh, and numerical examples are illustrated to verify its efficiency and accuracy.

Chapter 5: The two-dimensional flat-top PU mesh is coupled with high-order DDA method in this chapter. After segmented by the discontinuities and boundary, the generated flat-top PU mesh is optimized with the concept of percentage of effective area to avoid ill-conditioned stiffness matrix. The improved contact algorithm is presented, which can significantly improve the accuracy of interaction between blocks, especially in the case of collision. After that, several numerical examples concerning the displacement/stress field within discontinuous problem domain are presented and analyzed.

Chapter 6: Flat-top PU based high-order DDA method is utilized to simulate stress wave propagation in this chapter. Non-reflective boundary is incorporated to simulate infinite problem domain. Three numerical examples, i.e. stress wave propagation within a long elastic bar, an infinite plate and a half infinite plate, are illustrated and analyzed in detail.

Chapter 7: Based on the two crack propagation modes in two-dimensional space, the tension failure criterion and the Mohr-Coulomb failure criterion are adopted to determine crack initiation. According to the two basic crack propagation cases in flat-top PU mesh, the crack propagation algorithm is established. Numerical examples are provided to verify the efficiency and accuracy of flat-top PU based high-order DDA method in simulating crack propagation.

Chapter 8: Conclusions and recommendations for future work are presented.

CHAPTER 2 LITERATURE REVIEW

2.1 Introduction

In this chapter, the development and application of several main numerical methods are reviewed in Section 2.2. The methods can be mainly classified into two groups, i.e. continuum-based group and discontinuum-based group. While the NMM method can be treated as a combination of continuum-based FEM and discontinuum-based DDA. The necessity of adopting flat-top PU mesh to discretize problem domain is analyzed in Section 2.3. According to the literature review in Sections 2.2 and 2.3, the flat-top PU based high-order DDA method is introduced in Section 2.4 to overcome the limitations in other numerical methods. After that, the relationship between flat-top PU based DDA, DDA method and NMM is analyzed in Section 2.5. The advantages of flat-top PU based high-order DDA method are summarized in Section 2.6. However, since the flat-top PU mesh is initially proposed in 1997 (Babuška and Melenk, 1997), limited development has been made in recent years, especially for the coupled flat-top PU based high-order DDA method. There are still some limitations in current code for this method, which are also stated in Section 2.6.

2.2 Development of numerical analysis method

In the process of searching for displacement and stress/strain solutions under different initial, boundary and material property conditions, researchers keep on exploring more accurate and efficient methods. Apparently, numerical methods have successfully attracted researchers' attention by taking good advantages of computer science to improve their accuracy and efficiency of the calculation.

Typically, the numerical methods can be classified into two main categories according to the scope of their application, i.e. continuum-based method and discontinuum-based method. For the continuum-based methods, three main methods will be reviewed in this section, which are Finite Difference Method (FDM), Finite Element Method (FEM) and Boundary Element Method (BEM). The review for the discontinuum-based methods includes Distinct Element Method (DEM) and Discontinuous Deformation Analysis (DDA) method. Moreover, the Numerical Manifold Method (NMM), which can be recognized as a coupling of

FEM and DDA, will also be reviewed in this section. Because NMM has a close relationship with the flat-top PU based DDA method, which will be further explained in Section 2.5.

2.2.1 Continuum-based numerical methods

There are three main continuum-based numerical methods, i.e. Finite Difference Method (FDM), Finite Element Method (FEM) and Boundary Element Method (BEM).

Finite Difference Method (FDM)

As the earliest numerical method, FDM utilizes regular mesh, such as regular quadrilateral mesh as shown in Figure 2.1(a), to discretize a problem domain and substitute differences between adjacent points for partial derivatives (Courant et al., 1952; Liszka and Orkisz, 1980; Cui, 2009). This theory makes FDM easy to understand and implement. However, due to the limitation of mesh generation, the original FDM is not capable of simulating the problems with irregular boundary or material inhomogeneity. In order to enhance the flexibility of mesh generation in FDM, Brighi et al. (1998) employed triangular grids as shown in Figure 2.1(b); Perrone and Kao (1975) introduced irregular quadrilateral mesh as illustrated in Figure 2.1(c); while Sukumar (2003) adopted Voronoi cell as demonstrated in Figure 2.1(d). The FDM with triangular and quadrilateral grids is developed into the well-known commercial software, FLAC. Moreover, FDM is also extended to three-dimensional analysis with the staggered-grid approach (Pitarka, 1999; Bohlen, 2002), as illustrated in Figure 2.1(e).

(a) Regular quadrilateral mesh

(b) Triangular mesh

(c) Irregular quadrilateral mesh
(Perrone and Kao, 1975)

(d) Voronoi cell (Sukumar, 2003)

(e) Staggered-grid approach (Bohlen, 2002)

Figure 2.1 Different mesh type for DEM

As the oldest numerical method, FDM has been utilized to simulate various engineering problems, such as the stress concentration (Chen, 1975), stress wave propagation (Virieux, 1984, 1986; Dai et al., 1995), nonlinear shell problem (Bushnell et al., 1971), fluid flow (Narasimhan and Witherspoon, 1976; Morinishi et al., 1998; Casulli, 1999) and heat conduction (Brian, 1961; Ong, 1974; Raynaud and Bransier, 1986).

Finite Element Method (FEM)

Over the past decades, FEM has become the most popular continuum-based numerical method. It has been fully developed from different aspects and widely applied in various fields.

In FEM, the problem domain is also firstly discretized with mesh (Cavendish, 1974; Lindholm, 1983; Caendish et al., 1985). However, compared with FDM, FEM is more flexible in mesh generation, which ensures its capability in discretizing problem domain with irregular boundary, as shown in Figure 2.2. For each element in the FEM mesh, PU based shape functions can be generated with the isoparametric mapping, in which the coefficients can be determined by the coordinates of the nodes distributed over the element, while the displacements for each node are unknowns. Meanwhile, the continuity between elements can be ensured with the shape functions. According to the Gaussian quadrature integration and the principle of virtual work, the global equilibrium equations can be established. Thus, the unknowns can be obtained by solving the equations, and the displacement field can be derived with the shape functions. In FEM, material inhomogeneity and anisotropy can be easily realized by assigning elements with different material properties or different constitutive models. Generally, FEM is more accurate and have a wider application scope.

Figure 2.2 FEM mesh for problem domain with irregular boundary (Lo, 1985)

However, in order to acquiring satisfactory result in FEM, the work of generating and refining mesh is demanding and time-consuming, especially for problems with complex boundary condition. Sometimes, for the purpose of high efficiency and accuracy, mesh generation is dependent on the experience of researchers/engineers. Trial-and-error is necessary for the beginners. A large amount of research work has been carried out to overcome this problem. Johnson and Hansbo (1992) employed the adaptive FEM, which includes the h-, p- and hp-versions of Finite Element Method. Based on the partition of unity (PU) method (Melenk and Babuška, 1996), the so-called Generalized Finite Element Method (GFEM) was developed (Strouboulis et al., 2000, 2001), in which the mesh generated is independent of the geometry of the problem domain, as illustrated in Figure 2.3.

Figure 2.3 GFEM mesh independent of the geometry for problem domain
(Strouboulis et al., 2001)

As the most popular continuum-based numerical method, FEM has been widely used in simulating various engineering problems. Griffiths and Lane (1999) utilized FEM to analyze the stability of slope and dam. FEM was also extended to the non-linear problems (Zienkiewicz et al., 1970) and elastic-plastic problems (Yamada and Yoshimura, 1968). Christian and Boehmer (1970) simulated plane-strain soil consolidation with FEM. Hughes et al. (1989) employed FEM to simulate dynamic fluid flow. The dynamic interaction between fluid and structure was analyzed by

Donea et al. (1982). Kuhlemeyer and Lysmer (1973) verified the accuracy of FEM in solving stress wave propagation problems. Duarte et al. (2000) analyzed the mechanical behavior of three-dimensional structures with the GFEM. Goodman et al. (1968) incorporated “Goodman joint element” into FEM to simulate the discontinuous rock system in geological engineering. Belytschko and Black (1999) and Moës et al. (1999) analyzed the process of crack growth with FEM. Dvorkin et al. (1990) simulated the strong discontinuities using EFEM. Bergara et al. (2017) studied the fatigue crack propagation with XFEM.

Boundary Element Method (BEM)

BEM is another frequently used continuum-based element, which tries to determine a weak solution at the global level based on the local approximation at the boundary. In BEM, only the boundary of the problem domain is discretized for constructing global equilibrium equations. After discretization, based on the shape function, Gaussian quadrature integration and an integral statement, the global equilibrium equations can be established and solved. After obtaining the displacement and stress at the boundary, the corresponding values for the points inside problem domain can be derived.

Since the boundary is a two-dimensional area for a three-dimensional body and a one-dimensional line for a two-dimensional domain, the dimension for computation in BEM is one less than the FDM and FEM. Therefore, it is easier for BEM to generate mesh and solve equations. However, due to the lack of discretization and integration within problem domain, BEM is not efficient in simulating material non-homogeneity and non-linearity, which limits the application scope of BEM. Therefore, Sutradhar and Paulino (2004) simulated non-homogeneous media with Galerkin BEM by transforming non-homogeneous problem into homogeneous problem according to some special governing equations; Nakayama and Washizu (1981) utilized incremental method to solve non-linear problem with BEM; Liao (1997) analyzed non-linear problem through incorporating BEM with the homotopy analysis method.

As a continuum-based numerical method, BEM has also been applied in solving various engineering problems. Crouch and Tian (1988) employed BEM to simulate

stress wave propagation and reflection in infinite plate. Lafhaj and Shahrour (2000) studied the shape factor to determine the soil permeability with BEM. The incompressible viscous fluid flow is simulated by Grigoriev and Dargush (1999) through adopting poly-region integration in BEM. Blandford et al. (1981) calculated the stress intensity factor with BEM. Portela et al. (1993) simulated the process of crack growth under mixed-mode with dual BEM according to the criterion of maximum principal stress.

Summary

A situation is confronted by all three continuum-based numerical methods, i.e. their intrinsic limitations of accurately describing the interaction mechanism between blocks and effectively simulating large deformation and large displacement for the discontinuous system. In order to overcome this disadvantage, some improvements have been made by researchers. For instance, Coates and Schoenberg (1995) introduced the equivalent medium theory into FDM; Goodman et al. (1968) incorporated the joint element with FEM; Portela (1992) generated the dual equations with the displacement and traction boundary integral equations in BEM, which is also called as dual BEM. Despite these achievements, the numerical models in these methods are still not fundamentally discontinuous with strict contact algorithm.

2.2.2 Discontinuum-based numerical methods

Two main discontinuum-based numerical methods will be introduced in this section, which are Distinct Element Method (DEM) and Discontinuous Deformation Analysis (DDA) method.

Distinct Element Method (DEM)

In order to simulate the discontinuous system, DEM was proposed by Cundall (1971). In DEM, the discontinuous system is composed of numbers of independent rigid/deformable blocks. So that, the requirement of compatibility for displacement and deformation between blocks is not necessary, which allows relative displacement between blocks and makes DEM suitable for simulating large displacement problems. Within each time step, the contact between blocks is

detected and updated. Stiff spring is added between two contact blocks to simulate the interaction force, and the spring will be removed as long as the blocks are separated. The contact algorithm in two-dimensional case can be represented in Figure 2.4. Mechanically, the contact force between two blocks can be divided into two components, i.e. normal force and tangential force. So that, a spring with stiff K_n is added in the normal direction, and a spring with stiff K_t and a friction angle Φ is added in the tangential direction, as shown in Figure 2.4. In DEM, the displacement, velocity and acceleration of current time step are calculated according to the corresponding values obtained in the last time step in an explicit way. Then, the force and stress for the numerical model can be derived based on the constitutive equations. Based on the basic theory of DEM and triangular discretization, the widely used commercial software, Universal Distinct Element Code (UDEC), was developed (Cundall, 1980). Subsequently, its three-dimensional version, 3DEC, was developed with the tetrahedral discretization (Cundall, 1988; Hart et al., 1988).

Figure 2.4 Representation of contact in two-dimensional DEM (Jing, 2003)

Owing to the intrinsic characteristics of DEM, the main objective of this method is discontinuous system. Chrysanthakis et al. (1997) employed DEM to study the performance of fiber reinforced shotcrete in underground tunnel excavation. Cai and Zhao (2000) modelled the one-dimensional wave propagation in rock masses including multiple parallel fractures with UDEC. Harper and Last (1990) analyzed the influence of fluid injection on the hydraulic conductivity in blocky system with DEM. Papantonopoulos (2002) utilized DEM to predict the response of columns under earthquake. Zhang et al. (1997) applied DEM to the simulation of slope and

blocky structure under dynamic loading. Meguro and Hakuno (1989) simulated the mechanical behavior of fracture in concrete structure with DEM.

Discontinuous Deformation Analysis (DDA) method

DDA method was proposed by Shi (1988) as an implicit discontinuum-based numerical method, and was extended to three-dimensional version (Shi, 2001; Jiang and Yeung, 2004; Yeung et al., 2007).

Similar with DEM, in DDA method, the numerical model is composed of independent blocks; the requirement of compatibility for displacement and deformation between blocks is not necessary; stiff springs will be added between blocks when contact is detected and removed when blocks are separated. Therefore, DDA is also capable of simulating large displacement and large deformation of blocky system, including the opening, closing and sliding between blocks.

However, the establishment of global equilibrium equations in DDA method is similar with FEM rather than DEM. The equations are constructed according to the principle of virtual work. Therefore, the displacement, velocity and stress of the blocks are obtained in an implicit way. Compared with DEM, DDA has the following advantages: 1) it can satisfy the requirement of equilibrium automatically and strictly; 2) a larger time step can be adopted without the concerning of numerical instability; 3) the combination of DDA and FEM is straightforward and easy to implement.

Moreover, in the process of deriving global equilibrium equations in DDA method, simplex integration (Shi, 1996) rather than the traditional Gaussian quadrature integration is employed. The simplex integration is able to offer analytical solution for integration through dividing polygon into triangles and polyhedron into tetrahedrons. Take the two-dimensional simplex integration as an example. As illustrated in Figure 2.5, five triangles are constructed by an arbitrary point P_0 and a pentagon $P_1P_2P_3P_4P_5P_1$, which are $P_0P_1P_2$, $P_0P_2P_3$, $P_0P_3P_4$, $P_0P_4P_5$ and $P_0P_5P_1$, respectively. Simplex integration can calculate the analytical solution for each triangle. Moreover, in simplex integration, the integration is orientational, which can be determined by the right-hand rule. Therefore, the integration over triangle $P_0P_1P_2$ is negative, while others are positive, as illustrated in Figure 2.5. Thus, the

sum of the integration over these five triangles is equal to the integration over pentagon $P_1P_2P_3P_4P_5P_1$. The simplex integration in three-dimensional space can be implemented in the same way. This improvement can reduce the numerical error caused by integration to some degree.

Figure 2.5 A simple scheme for two-dimensional simplex integration (Shi, 1996)

In the original DDA method, the displacement for each block is approximated by the first-order polynomial function, which means the stress/strain is constant within a block. This is far away from the situations in most engineering project in reality. Therefore, researchers keep on seeking ways to improve the accuracy of describing the stress field. For instance, Ke (1996) divided a large block into several small sub-blocks and glued them together by artificial joints; Shyu (1993) and Chang (1994) incorporated FEM into DDA by discretizing blocks with triangular or quadrilateral meshes; and Jiang et al. (2006) coupled FEM and DDA in three-dimensional space; Hsiung (2001) and Zhang (2009) introduced high-order polynomial function into DDA to approximate the displacement function of each block.

Due to the particular advantages offered by DDA method, it is rapidly developed and applied in various fields, especially in rock mechanics. Yeung and Leong (1997) analyzed the stability problem of tunnel under different distribution conditions of joints with the two-dimensional DDA method. Wu et al. (2006) utilized DDA

method to study the initiation, development and failure of the Qianjiangping landslide in Three Gorges Reservoir area. Kim et al. (1999) extended the DDA method to study the interaction between water and rock, the influence of loading/unloading sequence and the effect of rock reinforcement. Gu and Zhao (2009) investigated the factors affecting the transmission and reflection of stress wave, such as block size, boundary conditions and wave frequency. Ning et al. (2011a) simulated the whole failure process of rock mass induced by blasting with DDA method. Cai et al. (2013) simulated the process of fracture propagation through coupling FEM with DDA method.

2.2.3 Numerical Manifold Method

Numerical Manifold Method (NMM) (Shi, 1995; Jiang et al., 2009; Tal et al., 2014) is treated as a combination of DDA method and FEM. It takes the main advantages of DDA and FEM through defining two sets of covers, i.e. physical covers and mathematical covers. The physical covers are generated by the intersection of the problem domain and mathematical covers. Then, the manifold elements can be generated through the intersection of different physical covers.

For example, in Figure 2.6 (a), the problem domain is surrounded by the thicker lines, which contains a triangle and a trapezoid separated by a discontinuity, while two mathematical covers M_1 and M_2 are represented by the rectangle and circle, respectively. As shown in Figure 2.6 (b), after the intersection of the problem domain and M_1 , two physical covers, i.e. P_1 and P_2 , are generated; it is the same for the physical covers P_3 and P_4 . Apparently, different physical covers overlap each other, as shown by the dashed line in Figure 2.6 (b). After the intersection of different physical covers, five manifold elements can be generated, as illustrated in Figure 2.6 (c).

PU based shape function is derived for every mathematical cover, while local approximation function is assigned to each physical cover. Then, the global approximation can be determined by the shape functions and local approximation functions. Similar with FEM and DDA, the global equilibrium equations are established according to the principle of virtual work. The displacement and

velocity can be obtained by solving the equations, and the stress can be derived according to the constitutive equations.

(a) Problem domain and mathematical covers

(b) Physical covers

(c) Manifold elements

Figure 2.6 The generation of manifold elements in NMM (Ma et al., 2009)

NMM is capable of discretizing blocks with manifold elements as in FEM and modeling discontinuities with contact algorithm as in DDA. So that, the accuracy of describing stress/strain field can be improved in simulating discontinuous system.

Since NMM possesses these advantages, it has attracted numbers of researchers. In the last two decades, large amount of research work has been carried out to improve its accuracy in two ways, namely modifying the meshes and increasing the degree of local approximation functions. Cheng et al. (2002) adopted Wilson non-

confirming element in quadrilateral covers and verified this theory by simulating a cantilever slab. Chen et al. (1998) developed NMM with high-order displacement functions based on the triangular meshes and verified its accuracy with a beam subjected to point loading. However, similar to other numerical methods utilizing PU theory to generate shape function, linear dependence problem is arising and the global stiffness matrix is ill-conditioned for high-order local approximation (An et al., 2011).

In recent years, NMM has been widely applied in various engineering problems. Ma et al. (2009) simulated the mechanical behavior of multiple branched cracks and evaluated the stress intensity factors under mixed-mode with NMM. Chiou et al. (2002) incorporated the virtual crack extension method into NMM to investigate the fracture propagation under mixed-mode according to the maximum tangential stress criterion. Fan et al. (2013) utilized NMM to simulate the process of stress wave propagation in rock mass with numerous inherent fractures and carried out the safety assessment for a tunnel under blasting. Jiang et al. (2010) modeled the unconfined seepage flow by the three-dimensional NMM with tetrahedral mesh. Zhang and Zhou (2006) analyzed the dynamic behavior of saturated porous media with NMM. Ning et al. (2011) studied the factors influencing the stability of footwall slope and simulated its failure process by incorporating the Mohr-Coulomb criterion into NMM.

2.3 Linear dependence problem in high-order PU based method

The concept of partition of unity has been adopted in various numerical methods, such as the partition of unity method (Babuška and Melenk, 1997), the PU-based FEM (Melenk and Babuška, 1996; Dolbow et al., 2000), the hp-cloud method (Duarte and Oden, 1996), the cloud-based hp FEM (Oden et al., 1998), the PU-based meshfree method (Rabczuk and Zi, 2007), the element-free Galerkin method (Belytschko and Fleming, 1999), the GFEM (Strouboulis et al., 2001), the PU-based local maximum-entropy (Amiri et al., 2014) and the NMM (Shi, 1995). In these numerical methods, partition of unit requires that: 1) the value for all the shape functions is within the interval $[0, 1]$; 2) the sum of all the shape functions is equal

to constant one for the whole problem domain. Therefore, for each element, the requirement of partition of unity can be expressed as follows:

$$\begin{aligned} 0 \leq w_i(X) \leq 1 & \quad (X \in e_i) \\ w_i(X) = 0 & \quad (X \notin e_i) \\ \sum_i w_i(X) = 1 & \quad (X \in e_i) \end{aligned} \quad (2.1)$$

where w_i is the shape function for i th element e_i ; X represents the coordinate.

However, for all these PU based numerical methods, linear dependence problem arises when high-order polynomial is employed as local approximation (Babuška and Melenk, 1997; Tian et al., 2006). Moreover, the property of linear dependence for these methods has been proven theoretically by An et al. (2011, 2012) with the rank deficiency counting approach. Take the element in one-dimensional PU mesh in Figure 2.7 as an example. The two red nodes are added into the mesh when constructing the element. For each node, a polynomial with degree p is assigned as local approximation. Thus, there are $p + 1$ unknowns for every node, and the total unknowns in the global equilibrium equations are $2 \times (p + 1)$. In PU mesh, the space of the shape function for a one-dimensional element is $\{1, x\}$. Therefore, after multiplying the local approximation with the shape function, the space of the global approximation is the same as the space of a polynomial with degree $p + 1$. Hence, the independent DOFs in the global approximation are $p + 2$. Therefore, the rank deficiency RD for this element is calculated as:

$$RD = 2 \times (p + 1) - (p + 2) = p \quad (2.2)$$

From Eq. (2.2), it is observed that the rank deficiency RD is greater than 0 when $p > 0$. As a result, linear dependence problem occurs when $p > 0$.

Figure 2.7 An element in traditional PU mesh

In order to solve the linear dependence problem, Strouboulis et al. (2000) implemented a simple modification within the direct linear solver; Tian and Yagawa (2005) introduced the high-performance elements, i.e. 4-node quadratic

tetrahedral element and 3-node quadratic triangular element, into the framework of GFEM; Cai et al. (2010) developed a triangular element with incomplete quadratic polynomial which did not include the terms x^2 and y^2 ; Zheng and Xu (2014) imposed a constraint on the gradients-based DOFs over the traction-free boundary part; while Ghorashi et al. (2015) adopted analysis-suitable T-splines to ensure the property of linear independence, in which the T-mesh satisfied some topological conditions.

However, the most commonly used method to resolve this problem is adding flat-top regions into the original PU based mesh system (Hong and Lee, 2013a, 2013b; An et al., 2014; He et al., 2015). The construction of flat-top PU mesh in one-dimensional case is illustrated in Figure 2.8 as an example. Firstly, the original PU mesh is shown in Figure 2.8 (a), where the red-colored problem domain is discretized into several line elements by the black-colored nodes x_i ($i = 1, 2, \dots, n$). The shape functions for these line elements are depicted over the problem domain, which are denoted as w_i ($i = 1, 2, \dots, n$), respectively. Then, the flat-top PU mesh can be constructed by adding the two adjacent shape functions together according to:

$$\bar{w}_i = w_{(2i-1)} + w_{2i} \quad (2.3)$$

After summation, several flat-tops appear as illustrated in Figure 2.8 (b). These flat-tops are called flat-top regions. The shape functions for these flat-top regions are constant. The shape functions for non-flat-top regions keep the same as in Figure 2.8 (a), which are called as PU regions. The corresponding elements under these shape functions are denoted as flat-top elements and PU elements, respectively. An et al. (2014) and He et al. (2015) have proven in detail that it is linearly independent for the flat-top PU mesh in one- and two-dimensional spaces.

(a) Original PU mesh

(b) Flat-top PU mesh

Figure 2.8 Construct flat-top PU mesh by adding flat-top regions into PU mesh
(after An et al., 2014)

Therefore, in this research work, flat-top PU mesh is selected to be coupled with the high-order DDA method to improve the accuracy of numerical simulation and avoid linear dependence problem in high-order PU based method. Moreover, in this thesis, the flat-top PU mesh is constructed based on the extension of nodal supports in traditional PU mesh, which is easier to understand and more straightforward to implement compared with other construction methods (Riker and Holzer, 2009; Hong and Lee, 2013), especially for the two- and three-dimensional cases.

2.4 Flat-top PU based high-order DDA

In high-order DDA method, even though high-order approximation can improve the numerical accuracy to some degree, it is still difficult to describe a complex stress field, such as the stress wave within a block. Therefore, it is necessary to discretize the blocks firstly. Then, high-order polynomial can be assigned locally to approximate the stress field piecewise. Some researchers divided the blocks into sub-blocks and connected them together with springs (Ke, 1996; Zhang, 2009; Liu, 2013). However, it is usually experience dependent to determine the number, stiffness and distribution of the springs. Therefore, flat-top PU mesh is incorporated with the high-order DDA method in this thesis. With the shape functions in the flat-top PU mesh, the continuity between elements can be ensured, automatically.

In flat-top PU based high-order DDA method, the flat-top elements and PU elements are generated by the intersection of discontinuities and flat-top PU mesh. Take the one-dimensional case as an example. As illustrated in Figure 2.9 (a), the red-colored problem domain is divided by a point discontinuity into two independent parts. Then, a regularly patterned flat-top PU mesh is covered over the problem domain as shown in Figure 2.9 (b). After it is intersected by the

discontinuity, the numerical model for flat-top PU based high-order DDA is generated in Figure 2.9 (c), where the flat-top region containing the discontinuity in Figure 2.9 (b) is divided into two independent flat-top regions. The interaction between them can be simulated with the contact algorithm in DDA method. In the numerical model, each flat-top element together with its adjacent PU elements forms a flat-top PU unit. The shape function for a flat-top PU unit is demonstrated as the blue-colored line in Figure 2.9 (c).

Figure 2.9 Generation of numerical model for flat-top PU based high-order DDA

In flat-top PU based high-order DDA method, high-order polynomial is assigned as local displacement approximation for every flat-top PU unit. In one-dimensional space, it can be expressed as:

$$u_i = \sum_{j=1}^m a_{ij} f_{ij}(x) \quad (2.4)$$

where u_i represents the local displacement approximation for i th flat-top PU unit; m is the number of monomials included in the high-order polynomial; a_{ij} is the coefficient, which is recognized as unknowns; $f_{ij}(x)$ represents the monomial, in which x is the coordinate variable. The global displacement approximation can be derived by multiplying the local approximation with the corresponding shape function. Then, the global equilibrium equations can be established according to the constitutive equations and the principle of virtual work from element level. Finally, the displacement, velocity, strain and stress can be obtained after solving the equations.

Therefore, flat-top PU-based high-order DDA method is a promising numerical method in simulating continuous and discontinuous system. It is capable of: 1) utilizing flat-top PU mesh and high-order approximation to calculate complicated stress/strain field distributed in a block; 2) simulating the interaction between blocks with contact algorithm; 3) avoiding linear dependence problem when high-order polynomial is assigned as local approximation.

2.5 Relationship between flat-top PU based DDA, DDA and NMM

Actually, the flat-top PU based DDA method can be transferred into DDA or NMM easily by changing the length of PU elements or flat-top elements, respectively.

Take the one-dimensional case as an example. Firstly, the numerical model for flat-top PU based DDA is illustrated in Figure 2.10 (a), in which the flat-top elements are connected by the PU elements. If the length for PU elements shrinks as in Figure 2.10 (b), the DDA method can be generated as a special case of this situation, as demonstrated in Figure 2.10 (c). At this time, flat-top elements are transferred into independent blocks, and the continuity between them is no longer satisfied with the vanishing of PU elements. Instead, the interaction between blocks is determined by the contact algorithm in DDA. On the contrary, if the length for flat-top elements shrinks as in Figure 2.10 (d), NMM can be constructed as a special case under this condition, as illustrated in Figure 2.10 (e). At this time, the block is discretized with traditional PU mesh, and the continuity is still automatically satisfied. However, since the flat-top elements vanish, linear dependence occurs when high-order local approximation is employed.

Therefore, DDA method and NMM can be treated as two special cases of flat-top PU based DDA method.

(a) Flat-top PU based DDA

(b) Shrinkage of PU elements

Figure 2.10 Relationship between flat-top PU based DDA, DDA and NMM

2.6 Advantages and limitations of flat-top PU based high-order DDA method

According to the literature review and the introduction of flat-top PU based high-order DDA, the advantages for this method can be concluded as follows:

1. It is an implicit numerical method. Compared with the explicit methods, a bigger time step can be selected without concerning about the numerical stability problem.
2. It is capable of modelling both continuous and discontinuous systems with the coupling of mesh and contact algorithm.
3. Compared with original DDA method, the work for contact detection and calculation can be reduced with only significant joints are considered, rather than constructing the problem domain with numbers of individual blocks.
4. Compared with adding springs to glue blocks together, the continuity between blocks can be automatically ensured without concerning about the number, stiffness and distribution of springs.
5. The construction of flat-top PU mesh is flexible without concerning about the compatibility with boundary.

6. Since simplex integration is adopted, closed-form solutions can be provided when establishing the global equilibrium equations, which can improve the numerical accuracy compared with the traditional Gaussian quadratic integration.
7. The numerical accuracy can be improved with the mesh discretization and the high-order polynomial approximation.
8. Compared with traditional PU based numerical methods, when high-order polynomial is employed as local approximation, linear dependence problem can be avoided with the construction of flat-top regions.

However, since the coupling of flat-top PU mesh and high-order DDA is still at a primary stage of development, limitations are confronted with the current code developed for this method. Further study is necessary in the future to improve the accuracy of crack propagation and extend the application scope of this method, which can be concluded as follows:

1. In order to establish a strict local refinement algorithm, quadrilateral flat-top PU mesh is adopted, and due to the limitation of time for further research work, the crack propagates along the boundaries of the elements in current code. Therefore, the path for crack propagation is approximated with a series of serrations. It will be more accurate if the cracks are allowed to propagate through elements. Correspondingly, new flat-top PU units will be generated by splitting the original elements with crack. Meanwhile, the determination of the unknowns for newly generated flat-top PU units is challenging, which is necessary for the calculation in next time step.
2. Currently, the flat-top PU based high-order DDA is applied in static/dynamic analysis of structures, stress wave propagation in continuous media and crack propagation. However, considering its advantages, this method is applicable to various other engineering problems, such as the stress wave propagation within fractured media, rock bolting for underground excavation, fluid flow within porous media, structure deformation under high-heat condition, material nonlinearity and elastic-plastic problems and so on.

CHAPTER 3 FLAT-TOP PU BASED HIGH-ORDER APPROXIMATION FOR CONTINUUM PROBLEM

3.1 Introduction

In this chapter, the flat-top PU mesh, which has been proven linearly independent (An et al., 2014; He et al., 2015), is utilized for discretizing continuous problem domain. Firstly, in Section 3.2, the construction of flat-top PU mesh based on the traditional PU mesh is introduced for one- to three-dimensional spaces, respectively. Then, high-order polynomial approximation within the framework of flat-top PU mesh is presented in Section 3.3, in which the local approximation, global approximation, constitutive function and the establishment of global equilibrium equations are included. After summarizing the numerical analysis procedure in Section 3.4, several numerical examples are presented in Section 3.5. Finally, conclusions regarding to the flat-top PU based high-order approximation within continuous problem domain are addressed in Section 3.6.

3.2 Construction of flat-top PU mesh within continuum

Based on the traditional PU mesh, the construction of flat-top PU mesh is easy and straightforward. The main idea for adding flat-top regions into the original PU mesh is to extend the original nodal supports into elemental supports. Details will be provided for one- to three-dimensional spaces in this section.

3.2.1 One-dimensional flat-top PU mesh

In one-dimensional space, a traditional PU mesh is firstly constructed as shown in Figure 3.1 (a), in which only PU elements, e_{pui} ($i = 1, 2, \dots, n$), are generated with nodal supports x_j ($j = 0, 1, 2, \dots, n$), and w_{pui} is the shape function corresponding to the PU element e_{pui} , which satisfying the requirement of partition of unity as follows:

$$\begin{aligned} 0 \leq w_{puij}(x) \leq 1 & \quad (x \in e_{pui}) \\ w_{puij}(x) = 0 & \quad (x \notin e_{pui}) \\ w_{pui} = \sum_j w_{puij}(x) = 1 & \quad (x \in e_{pui}) \end{aligned} \tag{3.1}$$

where $w_{puij}(x)$ is the shape function for PU element e_{pui} generated by the j th nodal support x_j ; x represents the coordinate variable. Therefore, according to the requirement in Eq. (3.1), the shape function w_{puij} for PU element e_{pui} in one-dimensional space can be expressed as:

$$w_{puij}(x) = \begin{cases} \frac{x_{j+1} - x}{x_{j+1} - x_j} & (j = i - 1) \\ \frac{x - x_{j-1}}{x_j - x_{j-1}} & (j = i) \\ 0 & (j \neq i - 1 \& j \neq i) \end{cases} \quad x \in [x_{i-1}, x_i] \quad (3.2)$$

From Eq. (3.2), it can be noted that shape function w_{puij} is equal to one at the corresponding nodal support x_j , and decreases linearly to zero at the adjacent nodal support x_{j-1} or x_{j+1} , as illustrated in Figure 3.1 (a).

Then, each nodal support x_j is extended into an elemental support, which is denoted as flat-top element e_{fij} , as illustrated in Figure 3.1 (b). The value of the shape function w_{fij} for every flat-top element e_{fij} inherits from the original nodal support, which is equal to a constant and expressed as:

$$w_{fij}(x) = \begin{cases} 1 & (x \in e_{fij}) \\ 0 & (x \notin e_{fij}) \end{cases} \quad (3.3)$$

Each flat-top element e_{fij} together with its adjacent PU elements form a flat-top PU unit u_{fpj} , as depicted in Figure 3.1 (b). The shape function w_{fpj} for a flat-top PU unit u_{fpj} is composed of the shape functions derived according to its elemental support, i.e. flat-top element e_{fij} . Therefore, the shape function w_{fpj} can be expressed as:

$$w_{fpj} = w_{puij} + w_{fij} + w_{pu(j+1)j} \quad (3.4)$$

Take the flat-top PU unit u_{fp2} as an example. As shown in Figure 3.1 (b), as a complete unit, it includes one flat-top element e_{f2} and two PU elements, e_{pu2} and e_{pu3} . According to Eq. (3.4), the shape function w_{fp2} can be depicted by the blue-colored part.

Obviously, the adjacent flat-top PU units overlap each other on the PU element they share. For example, as demonstrated in Figure 3.1 (b), the units u_{fp1} and u_{fp2} overlap on the element e_{pu2} . So that, the continuity between adjacent units can be ensured automatically by the shape function of PU element they share.

(a) Traditional PU mesh

(b) Flat-top PU mesh

Figure 3.1 Construction of flat-top PU mesh in one-dimensional space

Then, the flat-top PU mesh is fixed to cover the whole red-colored problem domain, as shown in Figure 3.2 (a). After intersected by the boundary, the numerical model can be generated as in Figure 3.2 (b), where incomplete flat-top PU units, namely u_{fp0} , u_{fp2} and u_{fp3} , are generated at the two sides of the model. For each flat-top PU unit, whether it is complete or incomplete, independent high-order polynomial will be assigned as local displacement approximation.

(a) Flat-top PU mesh fixed over problem domain

(b) Numerical model

Figure 3.2 Generation of one-dimensional numerical model based on flat-top PU mesh

3.2.2 Two-dimensional flat-top PU mesh

In two-dimensional case, for the convenience of implementing local refinement in Chapter 4, the traditional quadrilateral PU mesh is utilized for constructing flat-top PU mesh.

Firstly, the traditional 4-node PU mesh is constructed as in Figure 3.3 (a), where the PU elements are numbered as e_{pui} ($i = 1, 2, \dots, n$) in sequence. The four nodes for each PU element e_{pui} are numbered from one to four in counter-clockwise direction, which can be represented as e_{puij} ($j = 1, 2, 3, 4$). In Figure 3.3 (a), the four nodes for element e_{pu5} are illustrated as examples, which can be referred as e_{pu51} , e_{pu52} , e_{pu53} and e_{pu54} , respectively. Apparently, each node is shared by four PU elements, except those distributed on the boundary. Therefore, the node e_{pu51} can also be referred as e_{pu42} , e_{pu73} and e_{pu84} .

Similar with the one-dimensional case, the shape functions for each PU element e_{pui} satisfy the requirement of partition of unity as follows:

$$\begin{aligned} 0 \leq w_{puij}(x, y) \leq 1 & \quad ((x, y) \in e_{pui}) \\ w_{puij}(x, y) = 0 & \quad ((x, y) \notin e_{pui}) \\ \sum_j w_{puij}(x, y) = 1 & \quad ((x, y) \in e_{pui}) \end{aligned} \quad (3.5)$$

where $w_{puij}(x, y)$ represents the shape function generated by the j th node in i th PU element, and (x, y) are the coordinate variables. Therefore, according to the

requirement in Eq. (3.5), the shape function w_{puij} for element e_{pui} can be expressed as:

$$w_{puij} = S_{puij} / S_{pui} \quad (3.6)$$

where S_{pui} is a determinant which can be calculated according to the four nodes of element e_{pui} as:

$$S_{pui} = \begin{vmatrix} 1 & x_{pui1} & y_{pui1} & x_{pui1}y_{pui1} \\ 1 & x_{pui2} & y_{pui2} & x_{pui2}y_{pui2} \\ 1 & x_{pui3} & y_{pui3} & x_{pui3}y_{pui3} \\ 1 & x_{pui4} & y_{pui4} & x_{pui4}y_{pui4} \end{vmatrix} \quad (3.7)$$

where (x_{puij}, y_{puij}) are the coordinates for j th node in element e_{pui} ; while S_{puij} is the determinant when the elements in j th row of S_{pui} are replaced by 1, x , y and xy , respectively. For example, S_{pui1} is expressed as:

$$S_{pui1} = \begin{vmatrix} 1 & x & y & xy \\ 1 & x_{pui2} & y_{pui2} & x_{pui2}y_{pui2} \\ 1 & x_{pui3} & y_{pui3} & x_{pui3}y_{pui3} \\ 1 & x_{pui4} & y_{pui4} & x_{pui4}y_{pui4} \end{vmatrix} \quad (3.8)$$

From Eq. (3.6) to Eq. (3.8), it can be shown that, similar with one-dimensional case, each shape function w_{puij} is equal to one at the corresponding nodal support (x_{puij}, y_{puij}) and decreases to zero at the adjacent nodal supports. However, in the two-dimensional case, the shape function is bilinear rather than linear. It is obvious that, for each internal node, four bilinear shape functions are generated for four different PU elements, which can be depicted as in Figure 3.3 (b).

(a) Traditional quadrilateral PU mesh

(b) Shape function generated by an internal nodal support

Figure 3.3 Traditional 4-node PU mesh and the corresponding shape function

Next, the nodal supports in traditional quadrilateral PU mesh are extended into elemental supports to construct flat-top elements, which are marked with grey color as illustrated in Figure 3.4 (a). The flat-top elements are numbered as e_{fik} ($k = 1, 2, \dots, n$) in sequence. The value of the shape function w_{fik} for each flat-top element e_{fik} inherits from the original nodal support, which is equal to 1 and expressed as:

$$w_{fik}(x, y) = \begin{cases} 1 & ((x, y) \in e_{fik}) \\ 0 & ((x, y) \notin e_{fik}) \end{cases} \quad (3.9)$$

However, in two-dimensional case, besides the flat-top elements, new PU elements are also generated in order to horizontally and vertically connect the flat-top elements together. These newly generated PU elements are marked with blue color as shown in Figure 3.4 (a). The derivation of shape functions for these blue-colored PU elements is exactly same with the original PU elements, which is expressed in Eqs. (3.6), (3.7) and (3.8). The white-colored PU elements in Figure 3.4 (a) represent the original PU elements in Figure 3.3 (a). All the blue- and white-colored PU elements are numbered as e_{pui} in new sequence, as shown in Figure 3.4 (a).

Each flat-top element e_{fik} together with its adjacent PU elements forms a flat-top PU unit u_{fpk} . The unit u_{fp6} in Figure 3.4 (a), which is surrounded by the red-colored quadrilateral, is illustrated as an example. As a complete flat-top PU unit, it is

composed of one flat-top element e_{ft6} , four blue-colored PU elements and four white-colored PU elements.

(a) Flat-top PU mesh

(b) Shape function for a complete flat-top PU unit

Figure 3.4 Construction of flat-top PU mesh and the shape function for a complete flat-top PU unit in two-dimensional space

From Figure 3.4 (a), it can be observed that, for each blue-colored PU element e_{pui} , it shares its four nodes with two different flat-top elements, which means that the four shape functions, w_{puij} ($j = 1, 2, 3, 4$), are derived according to two elemental supports. Therefore, the four shape functions pertain to two different flat-top PU units. Take the blue-colored element e_{pu6} as an example. It shares the nodes e_{pu61} and e_{pu62} with flat-top element e_{ft6} , and e_{pu63} and e_{pu64} with e_{ft2} . Therefore, the

corresponding shape functions w_{pu61} and w_{pu62} belong to unit u_{fp6} , while w_{pu63} and w_{pu64} belong to u_{fp2} . While for every white-colored PU element e_{pui} , it shares its four nodes with four different flat-top elements, which means that the four shape functions, w_{puij} ($j = 1, 2, 3, 4$), are derived according to four elemental supports. Therefore, the four shape functions pertain to four different flat-top PU units, respectively. Here, the white-colored element $e_{pu(17)}$ is taken as an example. It shares the nodes $e_{pu(17)1}$, $e_{pu(17)2}$, $e_{pu(17)3}$ and $e_{pu(17)4}$ with flat-top elements $e_{ft(10)}$, $e_{ft(11)}$, e_{ft7} and e_{ft6} , respectively. Therefore, the corresponding shape functions $w_{pu(17)1}$, $w_{pu(17)2}$, $w_{pu(17)3}$ and $w_{pu(17)4}$ pertain to unit $u_{ft(10)}$, $u_{ft(11)}$, u_{ft7} and u_{ft6} , respectively.

Therefore, the shape function for a complete flat-top PU unit can be depicted as in Figure 3.4 (b), from which it can be concluded that the shape functions for the flat-top element, four blue-colored PU elements and four white-colored PU elements are constant one, linear functions and bilinear functions, respectively.

From Figure 3.4 (a), it can be observed that the adjacent flat-top PU units overlap each other on the PU elements they share. So that the continuity between adjacent units can be automatically ensured through the shape functions for PU elements.

Then, the flat-top PU mesh is fixed to cover the whole problem domain, whose boundary is marked with red color as illustrated in Figure 3.5 (a). After the mesh is intersected by the physical boundary, the numerical model can be generated as shown in Figure 3.5 (b).

(a) Flat-top PU mesh fixed over problem domain

(b) Numerical model

Figure 3.5 Generation of two-dimensional numerical model based on flat-top PU mesh

Same with one-dimensional case, each flat-top PU unit in the numerical model, whether it contains all nine elements or not, is assigned with independent high-order polynomial as local displacement approximation.

3.2.3 Three-dimensional flat-top PU mesh

In three-dimensional space, the traditional hexahedral PU mesh is utilized for constructing flat-top PU mesh.

Firstly, the traditional 8-node hexahedral PU mesh is constructed as shown in Figure 3.6 (a), where the PU elements are numbered as e_{pui} ($i = 1, 2, \dots, n$) in sequence. However, in three-dimensional PU mesh, some of the elements, such as $e_{pu(13)}$, $e_{pu(14)}$ and $e_{pu(16)}$ and so on, are not visible, as they are blocked by other elements. The eight nodes for each PU element e_{pui} are numbered from one to eight in the sequence as illustrated in Figure 3.6 (b), which can be represented as e_{puij} ($j = 1, 2, \dots, 8$). From Figure 3.6 (a), it can be noticed that each node is shared by eight PU elements, except those distributed on the boundary. Meanwhile, each internal node, as a nodal support, generates eight shape functions for eight different elements. From the viewpoint of elements, as illustrated in Figure 3.6 (b), the shape functions for element e_{pui} are composed of eight shape functions derived based on eight nodal supports. As the PU mesh, the shape functions for each element e_{pui} satisfy the requirement of partition of unity as follows:

$$\begin{aligned}
 0 \leq w_{puij}(x, y, z) \leq 1 & \quad ((x, y, z) \in e_{pui}) \\
 w_{puij}(x, y, z) = 0 & \quad ((x, y, z) \notin e_{pui}) \\
 \sum_j w_{puij}(x, y, z) = 1 & \quad ((x, y, z) \in e_{pui})
 \end{aligned} \tag{3.10}$$

where $w_{puij}(x, y, z)$ represents the shape function generated by the j th node in i th PU element, and (x, y, z) are the coordinate variables. Therefore, according to the requirement in Eq. (3.10), the shape function w_{puij} for element e_{pui} can be expressed as:

$$w_{puij} = S_{puij} / S_{pui} \tag{3.11}$$

where S_{pui} is a determinant of 8×8 matrix which can be calculated according to the eight nodes of element e_{pui} as:

$$S_{pui} = \begin{vmatrix} 1 & x_{pui1} & y_{pui1} & z_{pui1} & x_{pui1}y_{pui1} & y_{pui1}z_{pui1} & z_{pui1}x_{pui1} & x_{pui1}y_{pui1}z_{pui1} \\ 1 & x_{pui2} & y_{pui2} & z_{pui2} & x_{pui2}y_{pui2} & y_{pui2}z_{pui2} & z_{pui2}x_{pui2} & x_{pui2}y_{pui2}z_{pui2} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & x_{pui8} & y_{pui8} & z_{pui8} & x_{pui8}y_{pui8} & y_{pui8}z_{pui8} & z_{pui8}x_{pui8} & x_{pui8}y_{pui8}z_{pui8} \end{vmatrix} \tag{3.12}$$

where $(x_{puij}, y_{puij}, z_{puij})$ are the coordinates for j th node in element e_{pui} ; while S_{puij} is a determinant of 8×8 matrix when the elements in j th row of S_{pui} are replaced by 1, x , y , z , xy , yz , zx and xyz , respectively. For example, S_{pui1} is expressed as:

$$S_{pui1} = \begin{vmatrix} 1 & x & y & z & xy & yz & zx & xyz \\ 1 & x_{pui2} & y_{pui2} & z_{pui2} & x_{pui2}y_{pui2} & y_{pui2}z_{pui2} & z_{pui2}x_{pui2} & x_{pui2}y_{pui2}z_{pui2} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & x_{pui8} & y_{pui8} & z_{pui8} & x_{pui8}y_{pui8} & y_{pui8}z_{pui8} & z_{pui8}x_{pui8} & x_{pui8}y_{pui8}z_{pui8} \end{vmatrix} \tag{3.13}$$

From Eq. (3.11) to Eq. (3.13), it can be shown that each shape function w_{puij} is equal to one at the corresponding nodal support (x_{puij}, y_{puij}) and decreases to zero at the adjacent nodal supports. In one- and two-dimensional spaces, the interpolations for the shape functions are linear and bilinear, respectively, while it is trilinear in three-dimensional case. For the consideration of visibility, the shape function w_{pui1} , which is derived based on the red-colored node e_{pui1} in Figure 3.6 (b), is depicted in the form of contour, as shown in Figure 3.6 (c). The contours on surfaces 1-2-3-4 and

1-4-8-5 are exactly same with that on 1-5-6-2; and the contour on surface 4-8-7-3 is the same with those on 5-6-7-8 and 2-3-7-6, which is equal to 0 everywhere.

(a) Traditional 8-node hexahedral PU mesh

(b) Numbering sequence for eight nodes in a hexahedral element

(c) Contour for shape function w_{pui1} generated by node e_{pui1}

Figure 3.6 Traditional hexahedral PU mesh and the shape function contour

Then, the nodal supports in traditional hexahedral PU mesh are extended into elemental supports to construct flat-top elements, which are marked with grey color as shown in Figure 3.7 (a). The flat-top elements are numbered as e_{fik} ($k = 1, 2, \dots, n$) in sequence. The value of the shape function w_{fik} for a flat-top element e_{fik} inherits from the original nodal support, which is equal to 1 and expressed as:

$$w_{fik}(x, y, z) = \begin{cases} 1 & ((x, y, z) \in e_{fik}) \\ 0 & ((x, y, z) \notin e_{fik}) \end{cases} \quad (3.14)$$

However, in three-dimensional case, besides the flat-top elements, two new types of PU elements are also generated in order to ensure the continuity between flat-top elements. These two new types of PU elements are marked with blue and orange colors, respectively, as shown in Figure 3.7 (a). Similar with two-dimensional case, all the blue-, orange- and white-colored PU elements are numbered as e_{pui} in new sequence. The derivation of shape functions for the blue- and orange-colored PU elements is exactly same with the original white-colored PU elements in Figure 3.6 (a), which is expressed in Eqs. (3.11), (3.12) and (3.13). The difference between these two new type elements will be introduced later in this section.

In order to have an insight for the internal structure of three-dimensional flat-top PU mesh, the first layer counting from the front view is removed, as demonstrated in Figure 3.7 (b). Thus, the remaining part of the three-dimensional flat-top PU mesh is shown in Figure 3.7 (c). At this time, nine white-colored PU elements are visible, which are the original e_{pu1} , e_{pu2} , ..., e_{pu8} and e_{pu9} in Figure 3.6 (a), respectively. Counting from the front view, the odd-numbered layers have the same distribution structure for different types of elements, and it is same with the even-numbered layers.

Each flat-top element e_{fik} together with its adjacent PU elements forms a flat-top PU unit u_{fpk} , as illustrated in Figure 3.8 (a). In order to have an insight of the flat-top PU unit, it is decomposed into layers, which are perpendicular to the direction as illustrated by the red-colored arrow in Figure 3.8 (a). Then, three layers are generated as shown in Figure 3.8 (b). It can be observed that, as a complete unit, it includes 27 elements in total, i.e. one flat-top elements, six blue-colored PU

elements, twelve orange-colored PU elements and eight white-colored PU elements. Moreover, the numbers of blue-, orange- and white-colored PU elements in a complete unit correspond to the numbers of surfaces, sides and nodes of the flat-top element, respectively.

In three-dimensional flat-top PU mesh, for the consideration of visibility, the shape function derived according to a specific elemental support, is depicted at element level instead of unit level in two-dimensional case.

(a) Flat-top PU mesh

(b) Removing the first layer from front view

(c) The remaining part of flat-top PU mesh

Figure 3.7 Construction of flat-top PU mesh in three-dimensional space

(a) A complete flat-top PU unit

(b) Decomposing a complete flat-top PU unit into layers

Figure 3.8 The external and internal structure of a complete flat-top PU unit

Firstly, for a flat-top element, it is an elemental support, which is not adjacent to any other flat-top elements, as shown by Figure 3.9 (a). From Eq. (3.14), it can be observed that the shape function is equal to one everywhere within a flat-top element, as illustrated in Figure 3.9 (b).

(a) A flat-top element not adjacent to any other flat-top elements

(b) Shape function contour for a flat-top element

Figure 3.9 A flat-top element and its shape function contour

As for the blue-colored PU elements, each of them connects two flat-top elements by sharing a surface with each one, as illustrated in Figure 3.10 (a). Therefore, based on the derivation of shape functions in Eqs. (3.11), (3.12) and (3.13), it can be observed that the eight shape functions w_{puij} ($j = 1, 2, \dots, 8$) for a blue-colored PU element are derived according to eight nodes pertaining to two elemental supports.

Take the blue-colored PU element e_{pui} in Figure 3.10 (a) as an example. Based on the numbering sequence for nodes in Figure 3.6 (b), the shape functions w_{pui1} , w_{pui4} , w_{pui5} and w_{pui8} are derived according to elemental support $e_{ft(k1)}$, while the rest four shape functions, i.e. w_{pui2} , w_{pui3} , w_{pui6} and w_{pui7} , are derived according to the elemental support $e_{ft(k2)}$. Therefore, the shape function $w_{fp(k1)}$ for the flat-top PU unit $u_{fp(k1)}$ regarding to the blue-colored PU element e_{pui} can be calculated as:

$$w_{fp(k1)}(x, y, z) = w_{pui1}(x, y, z) + w_{pui4}(x, y, z) + w_{pui5}(x, y, z) + w_{pui8}(x, y, z) \quad ((x, y, z) \in e_{pui}) \quad (3.15)$$

which is illustrated in Figure 3.10 (b) in the form of contour. It can be observed that the shape function is equal to one on the surface 1-4-8-5, which is shared with element $e_{ft(k1)}$; and it decreases linearly to zero on the surface 2-3-7-6, which is shared with element $e_{ft(k2)}$. It is similar with the shape function $w_{fp(k2)}$ for flat-top PU unit $u_{fp(k2)}$ regarding to element e_{pui} .

(a) A blue-colored PU element and its two elemental supports

(b) Shape function contour for the blue-colored PU element e_{pui} derived according to elemental support $e_{ft(k1)}$

Figure 3.10 Elemental supports for a blue-colored PU element and its shape function contour derived according to one elemental support

For the orange-colored PU elements, each of them connects four flat-top elements by sharing a side with each one, as illustrated in Figure 3.11 (a). Therefore, based on the derivation of shape functions in Eqs. (3.11), (3.12) and (3.13), it can be observed that the eight shape functions w_{puij} ($j = 1, 2, \dots, 8$) for an orange-colored PU element are derived according to eight nodes pertaining to four elemental supports.

Take the orange-colored PU element e_{pui} in Figure 3.11 (a) as an example. Based on the numbering sequence for nodes in Figure 3.6 (b), the shape functions w_{pui5} and w_{pui8} are derived according to the elemental support $e_{ft(k1)}$; w_{pui6} and w_{pui7} according to $e_{ft(k2)}$; w_{pui1} and w_{pui4} according to $e_{ft(k3)}$; while w_{pui2} and w_{pui3} according to $e_{ft(k4)}$. Therefore, the shape function $w_{fp(k1)}$ for the flat-top PU unit $u_{fp(k1)}$ regarding to the orange-colored PU element e_{pui} can be calculated as:

$$w_{fp(k1)}(x, y, z) = w_{pui5}(x, y, z) + w_{pui8}(x, y, z) \quad ((x, y, z) \in e_{pui}) \quad (3.16)$$

which is illustrated in Figure 3.11 (b) in the form of contour.

(a) An orange-colored PU element and its four elemental supports

(b) Shape function contour for the orange-colored PU element e_{pui} derived according to elemental support $e_{ft(k1)}$

Figure 3.11 Elemental supports for an orange-colored PU element and its shape function contour derived according to one elemental support

From Figure 3.11 (b), it can be observed that: 1) the shape function is equal to one on the side 8-5, which is shared with element $e_{ft(k1)}$; 2) the shape function decreases

linearly to zero on the sides 6-7 and 4-1, which are shared with elements $e_{ft(k2)}$ and $e_{ft(k3)}$, respectively; 3) the shape function decreases bilinearly to zero on the side 2-3, which is shared with element $e_{ft(k4)}$. It is similar with the shape functions $w_{fp(k2)}$, $w_{fp(k3)}$ and $w_{fp(k4)}$ for flat-top PU units $u_{fp(k2)}$, $u_{fp(k3)}$ and $u_{fp(k4)}$, respectively, regarding to element e_{pui} .

As for the white-colored PU elements, each of them connects eight flat-top elements by sharing a node with each one, as illustrated in Figure 3.12 (a). Therefore, based on the derivation of shape functions in Eqs. (3.11), (3.12) and (3.13), it can be observed that the eight shape functions w_{puij} ($j = 1, 2, \dots, 8$) for an white-colored PU element are derived according to eight nodes pertaining to eight different elemental supports, respectively.

Take the white-colored PU element e_{pui} in Figure 3.11 (a) as an example. Based on the numbering sequence for nodes in Figure 3.6 (b), the eight shape functions w_{pui1} , w_{pui2} , $\dots$, w_{pui7} and w_{pui8} are derived according to the elemental supports $e_{ft(k3)}$, $e_{ft(k4)}$, $e_{ft(k8)}$, $e_{ft(k7)}$, $e_{ft(k1)}$, $e_{ft(k2)}$, $e_{ft(k6)}$ and $e_{ft(k5)}$, respectively. Actually, in this case, the shape function $w_{fp(kl)}$ ($l = 1, 2, \dots, 8$) for the flat-top PU unit $u_{fp(kl)}$ regarding to PU element e_{pui} is exactly same with the shape function derived according to the corresponding original nodal support. For example, the original nodal support e_{pui1} in Figure 3.6 (b) is extended into elemental support $e_{ft(k3)}$ in Figure 3.12 (a). Therefore, the shape function $w_{fp(k3)}$ for the flat-top PU unit $u_{fp(k3)}$ regarding to the white-colored PU element e_{pui} is expressed as:

$$w_{fp(k3)}(x, y, z) = w_{pui1}(x, y, z) \quad ((x, y, z) \in e_{pui}) \quad (3.17)$$

which is illustrated in Figure 3.11 (b) in the form of contour.

It can be observed that: 1) the shape function is equal to one on the node 1, which is shared with element $e_{ft(k3)}$; 2) the shape function decreases linearly to zero on the nodes 2, 4 and 5, which are shared with elements $e_{ft(k4)}$, $e_{ft(k7)}$ and $e_{ft(k1)}$, respectively; 3) the shape function decreases bilinearly to zero on the nodes 3, 6 and 8, which are shared with elements $e_{ft(k8)}$, $e_{ft(k2)}$ and $e_{ft(k5)}$, respectively; 4) the shape function decreases trilinearly to zero on the node 7, which is shared element $e_{ft(k6)}$. It is similar with the other shape functions for the rest seven flat-top PU units regarding to element e_{pui} .

(a) A white-colored PU element and its eight elemental supports

(b) Shape function contour for the white-colored PU element e_{pui} derived according to elemental support $e_{ft(k3)}$

Figure 3.12 Elemental supports for a white-colored PU element and its shape function contour derived according to one elemental support

With the introduction of flat-top PU unit and four different types of elements in three-dimensional space, it is easy to observe that the adjacent flat-top PU units overlap each other on the PU elements they share. So that the continuity between adjacent units can be automatically ensured through the shape functions of PU elements.

Then, the flat-top PU mesh in Figure 3.7 (a) is fixed to cover the whole problem domain of the U-shaped structure in Figure 3.13 (a). After the mesh is intersected by the physical boundary, the numerical model can be generated as shown in Figure 3.13 (b). Same with one- and two-dimensional cases, each flat-top PU unit in the

numerical model, whether it contains all 27 elements or not, is assigned with independent high-order polynomial as local displacement approximation.

(a) A U-shaped structure

(b) Numerical model

Figure 3.13 Generation of three-dimensional numerical model based on flat-top PU mesh

3.3 High-order approximation within the framework of flat-top PU method

According to the Weierstrass approximation theorem, any continuous function, which is defined on a closed interval, can be uniformly approximated by polynomial as closely as desired. Therefore, in order to improve the accuracy of numerical results, high-order polynomial is utilized as local approximation for the displacement field of each flat-top PU unit, in which the coefficients of monomials are unknowns. The global displacement approximation can be obtained by multiplying the local high-order polynomial with the shape function corresponding

to the flat-top PU unit. Then, the stress/strain field can be derived according to the constitutive equations. Based on the displacement, stress, strain, forces and constraints, the global equilibrium equations are established according to the principle of virtual work. Then, the displacement, stress and strain can be obtained by solving the equations.

The high-order approximation for three-dimensional flat-top PU mesh is introduced as an example in the remaining part of this section. For two- and one-dimensional cases, the approximation can be easily obtained by reducing one and two dimension(s), respectively.

3.3.1 Global displacement approximation

The local displacement field (u_{xi}, u_{yi}, u_{zi}) for the flat-top PU unit u_{fpi} ($i = 1, 2, \dots, n$) is approximated by high-order polynomial as follows:

$$\begin{cases} u_{xi} = \sum_{j=1}^m a_{ij} f_j(x, y, z) \\ u_{yi} = \sum_{j=1}^m b_{ij} f_j(x, y, z) \\ u_{zi} = \sum_{j=1}^m c_{ij} f_j(x, y, z) \end{cases} \quad (3.18)$$

where (a_{ij}, b_{ij}, c_{ij}) represent coefficients, which are unknowns in global equilibrium equations; m is the number of monomials, which can be determined by the degree p of polynomial as:

$$m = \frac{(p+1)(p+2)(p+3)}{3!} \quad (3.19)$$

$f_j(x, y, z) = x^{\alpha_j} y^{\beta_j} z^{\gamma_j}$ represents the monomial, α_j, β_j and γ_j are the exponents of the bases x, y and z , respectively, which include all the non-negative integers satisfying the following requirements:

$$\begin{cases} 0 \leq \alpha_j, \beta_j, \gamma_j \leq p \\ 0 \leq \alpha_j + \beta_j + \gamma_j \leq p \end{cases} \quad (3.20)$$

For the convenience of reference in the rest part of this thesis, Eq. (3.18) is written in the matrix form as:

$$[u_i] = [T][D_i] \quad (3.21)$$

where $[u_i] = (u_{xi}, u_{yi}, u_{zi})^T$ is the local displacement matrix for flat-top PU unit u_{fpi} ; $[T]$ is the $3 \times 3m$ matrix for monomials, which can be expressed as:

$$[T] = \begin{pmatrix} f_1 & 0 & 0 & f_2 & 0 & 0 & \cdots & f_m & 0 & 0 \\ 0 & f_1 & 0 & 0 & f_2 & 0 & \cdots & 0 & f_m & 0 \\ 0 & 0 & f_1 & 0 & 0 & f_2 & \cdots & 0 & 0 & f_m \end{pmatrix} \quad (3.22)$$

where $f_j = f_j(x, y, z)$ for simplification; $[D_i]$ is the $3m \times 1$ matrix for unknowns in flat-top PU unit u_{fpi} , which can be expressed as:

$$[D_i] = (a_{i1} \ b_{i1} \ c_{i1} \ a_{i2} \ b_{i2} \ c_{i2} \ \cdots \ a_{im} \ b_{im} \ c_{im})^T \\ = (d_{i1} \ d_{i2} \ d_{i3} \ d_{i4} \ d_{i5} \ d_{i6} \ \cdots \ d_{i(3m-2)} \ d_{i(3m-1)} \ d_{i(3m)})^T \quad (3.23)$$

Based on the local approximation for flat-top PU units and the corresponding shape functions, the global displacement approximation is derived from element level. For an element e_k , which is either a flat-top element or a PU element, its global displacement approximation $[U_k] = (U_{xk}, U_{yk}, U_{zk})^T$ can be expressed as:

$$[U_k] = \sum_i w_{fpi}(x, y, z)[u_i] = \sum_i w_{fpi}(x, y, z)[T(x, y, z)][D_i] = \sum_i [T'_i(x, y, z)][D_i] \\ ((x, y, z) \in e_k) \quad (3.24)$$

where $w_{fpi}(x, y, z) ((x, y, z) \in e_k)$ represents the shape function for element e_k derived according to the elemental support in flat-top PU unit u_{fpi} ; $[T'_i(x, y, z)] = w_{fpi}(x, y, z)[T(x, y, z)]$.

Specifically, for the flat-top element e_k as shown in Figure 3.14 (a), it pertains to only one flat-top PU unit u_{fpi} , and its global displacement approximation can be expressed as:

$$[U_k] = [T'_i(x, y, z)][D_i] \quad ((x, y, z) \in e_k) \quad (3.25)$$

For the blue-colored PU element e_k as shown in Figure 3.14 (b), it pertains to two flat-top PU units, i.e. $u_{fpi(i1)}$ and $u_{fpi(i2)}$, and its global displacement approximation can be expressed as:

$$[U_k] = [T'_{i1}(x, y, z)][D_{i1}] + [T'_{i2}(x, y, z)][D_{i2}] \quad ((x, y, z) \in e_k) \quad (3.26)$$

For the orange-colored PU element e_k as shown in Figure 3.14 (c), it pertains to four flat-top PU units, i.e. $u_{fp(i1)}$, $u_{fp(i2)}$, $u_{fp(i3)}$ and $u_{fp(i4)}$, and its global displacement approximation can be expressed as:

$$[U_k] = [T'_{i1}(x, y, z)][D_{i1}] + [T'_{i2}(x, y, z)][D_{i2}] + [T'_{i3}(x, y, z)][D_{i3}] + [T'_{i4}(x, y, z)][D_{i4}] \quad ((x, y, z) \in e_k) \quad (3.27)$$

For the white-colored PU element e_k as shown in Figure 3.14 (d), it pertains to eight flat-top PU units, i.e. from $u_{fp(i1)}$ to $u_{fp(i8)}$, and its global displacement approximation can be expressed as:

$$[U_k] = [T'_{i1}(x, y, z)][D_{i1}] + [T'_{i2}(x, y, z)][D_{i2}] + [T'_{i3}(x, y, z)][D_{i3}] + [T'_{i4}(x, y, z)][D_{i4}] + [T'_{i5}(x, y, z)][D_{i5}] + [T'_{i6}(x, y, z)][D_{i6}] + [T'_{i7}(x, y, z)][D_{i7}] + [T'_{i8}(x, y, z)][D_{i8}] \quad ((x, y, z) \in e_k) \quad (3.28)$$

Figure 3.14 Deriving global displacement approximation in three-dimensional case

3.3.2 Constitutive equations

In high-order approximation with flat-top PU mesh, linear elasticity is assumed to establish the relationship between stress and strain, in which the strain field is derived according to the global displacement approximation in Section 3.3.1.

Firstly, for an element e_k , according to Eq. (3.24), the strain vector $(\varepsilon_{xk}, \varepsilon_{yk}, \varepsilon_{zk}, \varepsilon_{xyk}, \varepsilon_{yzk}, \varepsilon_{zrk})^T$ can be expressed by the displacement vector $(U_{xk}, U_{yk}, U_{zk})^T$ as follows:

$$\begin{pmatrix} \varepsilon_{xk} \\ \varepsilon_{yk} \\ \varepsilon_{zk} \\ \varepsilon_{xyk} \\ \varepsilon_{yzk} \\ \varepsilon_{zrk} \end{pmatrix} = \begin{pmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{pmatrix} \begin{pmatrix} U_{xk} \\ U_{yk} \\ U_{zk} \end{pmatrix} = [A][U_k] = [A] \sum_i [T_i][D_i] = \sum_i [A][T_i][D_i] \quad (3.29)$$

where $[A]$ represents the differentiation operator. It can be observed that the differentiation operator $[A]$ is only valid for matrix $[T_i]$ which contains coordinate variables x, y and z , and not valid for matrix $[D_i]$ which only includes coefficients for monomials. Therefore, Eq. (3.29) can be written as:

$$\begin{pmatrix} \varepsilon_{xk} & \varepsilon_{yk} & \varepsilon_{zk} & \varepsilon_{xyk} & \varepsilon_{yzk} & \varepsilon_{zrk} \end{pmatrix}^T = \sum_i [A][T_i][D_i] = \sum_i [B_i][D_i] \quad (3.30)$$

where $[B_i] = [A][T_i]$.

Then, based on the theory of linear elasticity, the stress vector $(\sigma_{xk}, \sigma_{yk}, \sigma_{zk}, \sigma_{xyk}, \sigma_{yzk}, \sigma_{zrk})^T$ for element e_k can be derived according to the strain vector $(\varepsilon_{xk}, \varepsilon_{yk}, \varepsilon_{zk}, \varepsilon_{xyk}, \varepsilon_{yzk}, \varepsilon_{zrk})^T$ as follows:

$$\begin{pmatrix} \sigma_{xk} \\ \sigma_{yk} \\ \sigma_{zk} \\ \sigma_{xyk} \\ \sigma_{yzk} \\ \sigma_{zrk} \end{pmatrix} = [E_k] \begin{pmatrix} \varepsilon_{xk} \\ \varepsilon_{yk} \\ \varepsilon_{zk} \\ \varepsilon_{xyk} \\ \varepsilon_{yzk} \\ \varepsilon_{zrk} \end{pmatrix} = [E_k] \sum_i [B_i][D_i] \quad (3.31)$$

where $[E_k]$ represents the elastic stiffness matrix for element e_k , which can be expressed by the Young's modulus E_k and Poisson's ratio ν_k as:

$$[E_k] = \frac{E_k}{2(1+\nu_k)(1-2\nu_k)} \begin{pmatrix} 2(1-\nu_k) & 2\nu_k & 2\nu_k & 0 & 0 & 0 \\ 2\nu_k & 2(1-\nu_k) & 2\nu_k & 0 & 0 & 0 \\ 2\nu_k & 2\nu_k & 2(1-\nu_k) & 0 & 0 & 0 \\ 0 & 0 & 0 & (1-2\nu_k) & 0 & 0 \\ 0 & 0 & 0 & 0 & (1-2\nu_k) & 0 \\ 0 & 0 & 0 & 0 & 0 & (1-2\nu_k) \end{pmatrix} \quad (3.32)$$

From Eq. (3.32), it can be observed that, with the construction of flat-top PU mesh, the Young's modulus E_k and Poisson's ratio ν_k can be assigned at element level to realize material inhomogeneity.

3.3.3 Establishment of global equilibrium equations

Based on the formula of global displacement in Section 3.3.1 and strain/stress in Section 3.3.2, the global equilibrium equations can be established with given boundary conditions according to the principle of virtual work. For a continuous problem domain, in the process of assembling global equilibrium equations, the sub-matrices induced by ten mechanical conditions are considered, which are elastic stiffness, initial stress, point loading, line loading, plane loading, volume loading, fixed point, fixed line, fixed plane and inertia force, respectively. The derivation for these sub-matrices is introduced one by one in the remaining part of this section.

Elastic stiffness

For an element e_k , the potential energy Π_{ek} contributed by elastic stiffness can be expressed as:

$$\begin{aligned} \Pi_{ek} &= \iiint_{V_k} \frac{1}{2} (\varepsilon_{xk} \sigma_{xk} + \varepsilon_{yk} \sigma_{yk} + \varepsilon_{zk} \sigma_{zk} + \varepsilon_{xyk} \sigma_{xyk} + \varepsilon_{yzk} \sigma_{yzk} + \varepsilon_{zjk} \sigma_{zjk}) dx dy dz \\ &= \frac{1}{2} \iiint_{V_k} \begin{pmatrix} \varepsilon_{xk} & \varepsilon_{yk} & \varepsilon_{zk} & \varepsilon_{xyk} & \varepsilon_{yzk} & \varepsilon_{zjk} \end{pmatrix} \begin{pmatrix} \sigma_{xk} \\ \sigma_{yk} \\ \sigma_{zk} \\ \sigma_{xyk} \\ \sigma_{yzk} \\ \sigma_{zjk} \end{pmatrix} dx dy dz \end{aligned} \quad (3.33)$$

where V_k represents the volume for element e_k . Substituting Eqs. (3.30) and (3.31) into Eq. (3.33), the potential energy Π_{ek} can be expressed in matrix form as:

$$\begin{aligned}\Pi_{ek} &= \frac{1}{2} \iiint_{V_k} \left(\sum_i [B_i] [D_i] \right)^T \left([E_k] \sum_i [B_i] [D_i] \right) dx dy dz \\ &= \frac{1}{2} \sum_j \sum_i \iiint_{V_k} [D_i]^T [B_i]^T [E_k] [B_j] [D_j] dx dy dz \\ &= \frac{1}{2} \sum_j \sum_i \left([D_i]^T \iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz [D_j] \right)\end{aligned}\quad (3.34)$$

According to the principle of virtual work, the derivative of Π_{ek} with respect to the unknowns d_{ir} and d_{js} ($r, s = 1, 2, \dots, 3m$) can be obtained as:

$$k_{ijrs} = \frac{\partial^2 \Pi_{ek}}{\partial d_{ir} \partial d_{js}} = \frac{1}{2} \frac{\partial^2}{\partial d_{ir} \partial d_{js}} \left\{ \sum_j \sum_i \left([D_i]^T \iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz [D_j] \right) \right\} \quad (3.35)$$

where k_{ijrs} is added into the element in r th row and s th column of matrix $[K_{ij}]$, and $[K_{ij}]$ is a $3m \times 3m$ sub-matrix, which is positioned at the intersection of the row for flat-top PU unit u_{fpi} and the column for flat-top PU unit u_{fpi} in global stiffness matrix.

From Eqs. (3.25)-(3.28) and Eq. (3.35), it can be observed that, by considering the elastic stiffness, 1) for each flat-top element as shown in Figure 3.14 (a), $3m \times 3m$ elements are generated and added into the matrix $[K_{ii}]$; 2) for each blue-colored PU element as shown in Figure 3.14 (b), $4 \times 3m \times 3m$ elements are generated and added into matrices $[K_{ij}]$ ($i, j = i1, i2$), correspondingly; 3) for each orange-colored PU element as shown in Figure 3.14 (c), $16 \times 3m \times 3m$ elements are generated and added into matrices $[K_{ij}]$ ($i, j = i1, i2, i3, i4$), correspondingly; 4) for each white-colored PU element as shown in Figure 3.14 (d), $64 \times 3m \times 3m$ elements are generated and added into matrices $[K_{ij}]$ ($i, j = i1, i2, \dots, i8$), correspondingly. In summary, the generated elements are added into the corresponding matrices as:

$$\begin{aligned}\iiint_{V_k} [B_i]^T [E_k] [B_i] dx dy dz &\rightarrow [K_{ii}] \\ \iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz &\rightarrow [K_{ij}] \\ \iiint_{V_k} [B_j]^T [E_k] [B_i] dx dy dz &\rightarrow [K_{ji}] \\ \iiint_{V_k} [B_j]^T [E_k] [B_j] dx dy dz &\rightarrow [K_{jj}]\end{aligned}\quad (3.36)$$

Initial stress

As for the initial stress, two conditions are considered for the reason of high-order approximation: 1) the initial stress is provided in input file, which is a set of constants for each element, this constant initial stress is only used in the first time step; 2) the initial stress is calculated at the end of last time step, which is expressed in the form of polynomial, this polynomial form initial stress is used in all the time steps but the first one.

Firstly, the constant initial stress is considered.

In the input file, the initial stress is defined as a set of constants for each element e_k .

Under this condition, the potential energy $\Pi_{\sigma_k^0}$ for element e_k contributed by initial stress $[\sigma_k^0] = (\sigma_{xk}^0 \quad \sigma_{yk}^0 \quad \sigma_{zk}^0 \quad \sigma_{xyk}^0 \quad \sigma_{yzk}^0 \quad \sigma_{zsk}^0)^T$ can be expressed as:

$$\begin{aligned} \Pi_{\sigma_k^0} &= -\iiint_{V_k} (\varepsilon_{xk} \sigma_{xk}^0 + \varepsilon_{yk} \sigma_{yk}^0 + \varepsilon_{zk} \sigma_{zk}^0 + \varepsilon_{xyk} \sigma_{xyk}^0 + \varepsilon_{yzk} \sigma_{yzk}^0 + \varepsilon_{zsk} \sigma_{zsk}^0) dx dy dz \\ &= -\iiint_{V_k} (\varepsilon_{xk} \quad \varepsilon_{yk} \quad \varepsilon_{zk} \quad \varepsilon_{xyk} \quad \varepsilon_{yzk} \quad \varepsilon_{zsk}) [\sigma_k^0] dx dy dz \end{aligned} \quad (3.37)$$

Substituting Eq. (3.30) into Eq. (3.37), the potential energy $\Pi_{\sigma_k^0}$ can be expressed in matrix form as:

$$\begin{aligned} \Pi_{\sigma_k^0} &= -\iiint_{V_k} \left(\sum_i [B_i] [D_i] \right)^T [\sigma_k^0] dx dy dz \\ &= -\sum_i \iiint_{V_k} [D_i]^T [B_i]^T [\sigma_k^0] dx dy dz \\ &= -\sum_i \left([D_i]^T \iiint_{V_k} [B_i]^T dx dy dz [\sigma_k^0] \right) \end{aligned} \quad (3.38)$$

According to the principle of virtual work, the derivative of $\Pi_{\sigma_k^0}$ with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = -\frac{\partial \Pi_{\sigma_k^0}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i \left([D_i]^T \iiint_{V_k} [B_i]^T dx dy dz [\sigma_k^0] \right) \right\} \quad (3.39)$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$, and $[F_i]$ represents the $3m \times 1$ sub-matrix corresponding to flat-top PU unit u_{fpi} in global force matrix.

From Eqs. (3.25)-(3.28) and Eq. (3.39), it can be observed that, by considering the constant initial stress, 1) for each flat-top element as shown in Figure 3.14 (a), $3m \times 1$ elements are generated and added into the matrix $[F_i]$; 2) for each blue-colored PU element as shown in Figure 3.14 (b), $2 \times 3m \times 1$ elements are generated and added into matrices $[F_i]$ ($i = i1, i2$), correspondingly; 3) for each orange-colored PU element as shown in Figure 3.14 (c), $4 \times 3m \times 1$ elements are generated and added into matrices $[F_i]$ ($i = i1, i2, i3, i4$), correspondingly; 4) for each white-colored PU element as shown in Figure 3.14 (d), $8 \times 3m \times 1$ elements are generated and added into matrices $[F_i]$ ($i = i1, i2, \dots, i8$), correspondingly. In summary, the generated elements are added into the corresponding matrices as:

$$\iiint_{V_k} [B_i]^T dx dy dz [\sigma_k^0] \rightarrow [F_i] \quad (3.40)$$

Then, the sub-matrix derived according to polynomial form initial stress is presented.

At the end of t th ($t = 1, 2, \dots, n$) time step, the stress for each element e_k can be obtained by substituting the solved unknowns $[D_i]_t$ into Eq. (3.31), which is the initial stress at the beginning of $(t+1)$ th time step. Therefore, the initial stress for the $(t+1)$ th time step is expressed in the form of polynomial. Under this condition, according to Eqs. (3.30) and (3.37), the potential energy $\Pi_{\sigma_k^0}$ for each element e_k , which is contributed by initial stress $[\sigma_k^0] = [E_k] \sum_i [B_i] [D_i]_t$, can be expressed as follows:

$$\begin{aligned} \Pi_{\sigma_k^0} &= - \iiint_{V_k} \left(\sum_i [B_i] [D_i] \right)^T \left([E_k] \sum_i [B_i] [D_i]_t \right) dx dy dz \\ &= - \sum_i \sum_j \left(\iiint_{V_k} [D_i]^T [B_i]^T [E_k] [B_j] [D_j]_t dx dy dz \right) \\ &= - \sum_i [D_i]^T \sum_j \left(\iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz [D_j]_t \right) \end{aligned} \quad (3.41)$$

According to the principle of virtual work, the derivative of $\Pi_{\sigma_k^0}$ in Eq. (3.41) with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = - \frac{\partial \Pi_{\sigma_k^0}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i [D_i]^T \sum_j \left(\iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz [D_j]_t \right) \right\} \quad (3.42)$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$.

Similar with the constant initial stress condition, the elements generated by Eq. (3.42) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, under this condition with polynomial form initial stress, the elements generated by Eq. (3.42) are added into the corresponding matrices as:

$$\sum_j \left(\iiint_{V_k} [B_i]^T [E_k] [B_j] dx dy dz [D_j]_t \right) \rightarrow [F_i] \quad (3.43)$$

Point loading

It is assumed that the loading $[F_{pk}] = (f_{px}, f_{py}, f_{pz})^T$ is applied at a given point (x_0, y_0, z_0) , which is located within the element e_k . According to Eq. (3.24), the potential energy Π_{Fpk} contributed by point loading can be expressed as:

$$\begin{aligned} \Pi_{F_{pk}} &= - \left(f_{px} \cdot U_{xk}(x_0, y_0, z_0) + f_{py} \cdot U_{yk}(x_0, y_0, z_0) + f_{pz} \cdot U_{zk}(x_0, y_0, z_0) \right) \\ &= - \begin{pmatrix} U_{xk}(x_0, y_0, z_0) & U_{yk}(x_0, y_0, z_0) & U_{zk}(x_0, y_0, z_0) \end{pmatrix} \begin{pmatrix} f_{px} \\ f_{py} \\ f_{pz} \end{pmatrix} \\ &= - [U_k(x_0, y_0, z_0)]^T [F_{pk}] \\ &= - \sum_i [D_i]^T [T'_i(x_0, y_0, z_0)]^T [F_{pk}] \end{aligned} \quad (3.44)$$

According to the principle of virtual work, the derivative of Π_{Fpk} with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = - \frac{\partial \Pi_{F_{pk}}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i [D_i]^T [T'_i(x_0, y_0, z_0)]^T [F_{pk}] \right\} \quad (3.45)$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$.

Similar with the situation for initial stress, the elements generated by Eq. (3.45) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering point loading are added into the corresponding matrices as:

$$[T'_i(x_0, y_0, z_0)]^T [F_{pk}] \rightarrow [F_i] \quad (3.46)$$

Line loading

It is assumed that the loading $[F_{lk}] = (f_{lx}, f_{ly}, f_{lz})^T$ is applied on the line L_k , which is located within the element e_k . According to Eq. (3.24), the potential energy $\Pi_{F_{lk}}$ contributed by line loading can be expressed as:

$$\begin{aligned} \Pi_{F_{lk}} &= -\iiint_{L_k} (f_{lx} \cdot U_{xk} + f_{ly} \cdot U_{yk} + f_{lz} \cdot U_{zk}) dx dy dz \\ &= -\iiint_{L_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{lx} \\ f_{ly} \\ f_{lz} \end{pmatrix} dx dy dz \\ &= -\iiint_{L_k} [U_k]^T [F_{lk}] dx dy dz \\ &= -\iiint_{L_k} \sum_i [D_i]^T [T'_i]^T [F_{lk}] dx dy dz \\ &= -\sum_i [D_i]^T \iiint_{L_k} [T'_i]^T dx dy dz [F_{lk}] \end{aligned} \quad (3.47)$$

According to the principle of virtual work, the derivative of $\Pi_{F_{lk}}$ with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = -\frac{\partial \Pi_{F_{lk}}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i [D_i]^T \iiint_{L_k} [T'_i]^T dx dy dz [F_{lk}] \right\} \quad (3.48)$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$.

Similar with the situation for initial stress, the elements generated by Eq. (3.48) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering line loading are added into the corresponding matrices as:

$$\iiint_{L_k} [T'_i]^T dx dy dz [F_{lk}] \rightarrow [F_i] \quad (3.49)$$

Plane loading

It is assumed that the loading $[F_{sk}] = (f_{sx}, f_{sy}, f_{sz})^T$ is applied on the plane S_k , which is located within the element e_k . According to Eq. (3.24), the potential energy $\Pi_{F_{sk}}$ contributed by plane loading can be expressed as:

$$\begin{aligned}
 \Pi_{F_{sk}} &= -\iiint_{S_k} (f_{sx} \cdot U_{xk} + f_{sy} \cdot U_{yk} + f_{sz} \cdot U_{zk}) dx dy dz \\
 &= -\iiint_{S_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{sx} \\ f_{sy} \\ f_{sz} \end{pmatrix} dx dy dz \\
 &= -\iiint_{S_k} [U_k]^T [F_{sk}] dx dy dz \\
 &= -\iiint_{S_k} \sum_i [D_i]^T [T'_i]^T [F_{sk}] dx dy dz \\
 &= -\sum_i [D_i]^T \iiint_{S_k} [T'_i]^T dx dy dz [F_{sk}]
 \end{aligned} \tag{3.50}$$

According to the principle of virtual work, the derivative of $\Pi_{F_{sk}}$ with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = -\frac{\partial \Pi_{F_{sk}}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i [D_i]^T \iiint_{S_k} [T'_i]^T dx dy dz [F_{sk}] \right\} \tag{3.51}$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$.

Similar with the situation for initial stress, the elements generated by Eq. (3.51) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering plane loading are added into the corresponding matrices as:

$$\iiint_{S_k} [T'_i]^T dx dy dz [F_{sk}] \rightarrow [F_i] \tag{3.52}$$

Volume loading

It is assumed that the loading $[F_{vk}] = (f_{vx}, f_{vy}, f_{vz})^T$ is applied on the volume V_k , which is located within the element e_k . According to Eq. (3.24), the potential energy $\Pi_{F_{vk}}$ contributed by volume loading can be expressed as:

$$\begin{aligned}
 \Pi_{F_{vk}} &= -\iiint_{V_k} (f_{vx} \cdot U_{xk} + f_{vy} \cdot U_{yk} + f_{vz} \cdot U_{zk}) dx dy dz \\
 &= -\iiint_{V_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{vx} \\ f_{vy} \\ f_{vz} \end{pmatrix} dx dy dz \\
 &= -\iiint_{V_k} [U_k]^T [F_{vk}] dx dy dz \\
 &= -\iiint_{V_k} \sum_i [D_i]^T [T'_i]^T [F_{vk}] dx dy dz \\
 &= -\sum_i [D_i]^T \iiint_{V_k} [T'_i]^T dx dy dz [F_{vk}]
 \end{aligned} \tag{3.53}$$

According to the principle of virtual work, the derivative of Π_{Fvk} with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$f_{ir} = -\frac{\partial \Pi_{Fvk}}{\partial d_{ir}} = \frac{\partial}{\partial d_{ir}} \left\{ \sum_i [D_i]^T \iiint_{V_k} [T_i']^T dx dy dz [F_{vk}] \right\} \quad (3.54)$$

where f_{ir} is added into the element in r th row of matrix $[F_i]$.

Similar with the situation for initial stress, the elements generated by Eq. (3.54) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering volume loading are added into the corresponding matrices as:

$$\iiint_{V_k} [T_i']^T dx dy dz [F_{vk}] \rightarrow [F_i] \quad (3.55)$$

Fixed point

It is assumed that the point (x_0, y_0, z_0) located within element e_k is fixed in x , y and z directions. In flat-top PU based high-order approximation, penalty method is utilized to implement this kind of constraint by adding a stiff spring at the fixed point in each direction. Assuming the stiffness for each spring is k_p , according to Eq. (3.24), the constraining force $[F_{cpk}] = (f_{cpx}, f_{cpy}, f_{cpz})^T$ at point (x_0, y_0, z_0) can be expressed as:

$$\begin{pmatrix} f_{cpx} \\ f_{cpy} \\ f_{cpz} \end{pmatrix} = \begin{pmatrix} -k_p U_{xk}(x_0, y_0, z_0) \\ -k_p U_{yk}(x_0, y_0, z_0) \\ -k_p U_{zk}(x_0, y_0, z_0) \end{pmatrix} = -k_p [U_k(x_0, y_0, z_0)] = -k_p \sum_i [T_i'(x_0, y_0, z_0)] [D_i] \quad (3.56)$$

Based on Eqs. (3.24) and (3.56), the potential energy Π_{Fcpk} contributed by fixed point can be expressed as:

$$\begin{aligned} \Pi_{Fcpk} &= -\frac{1}{2} (f_{cpx} \cdot U_{xk}(x_0, y_0, z_0) + f_{cpy} \cdot U_{yk}(x_0, y_0, z_0) + f_{cpz} \cdot U_{zk}(x_0, y_0, z_0)) \\ &= -\frac{1}{2} \begin{pmatrix} U_{xk}(x_0, y_0, z_0) & U_{yk}(x_0, y_0, z_0) & U_{zk}(x_0, y_0, z_0) \end{pmatrix} \begin{pmatrix} f_{cpx} \\ f_{cpy} \\ f_{cpz} \end{pmatrix} \\ &= \frac{1}{2} \left(\sum_i [T_i'(x_0, y_0, z_0)] [D_i] \right)^T \left(k_p \sum_i [T_i'(x_0, y_0, z_0)] [D_i] \right) \\ &= \frac{1}{2} k_p \sum_i \sum_j [D_i]^T [T_i'(x_0, y_0, z_0)]^T [T_j'(x_0, y_0, z_0)] [D_j] \end{aligned} \quad (3.57)$$

According to the principle of virtual work, the derivative of Π_{Fcpk} with respect to the unknowns d_{ir} and d_{js} ($r, s = 1, 2, \dots, 3m$) can be obtained as:

$$k_{ijrs} = \frac{\partial^2 \Pi_{Fcpk}}{\partial d_{ir} \partial d_{js}} = \frac{1}{2} \frac{\partial^2}{\partial d_{ir} \partial d_{js}} \left\{ k_p \sum_i \sum_j [D_i]^T [T'_i(x_0, y_0, z_0)]^T [T'_j(x_0, y_0, z_0)] [D_j] \right\} \quad (3.58)$$

where k_{ijrs} is added into the element in r th row and s th column of matrix $[K_{ij}]$.

Similar with the situation for elastic stiffness, the elements generated by Eq. (3.58) are added into one, four, sixteen and sixty-four matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering fixed point are added into the corresponding matrices as follows:

$$\begin{aligned} k_p [T'_i(x_0, y_0, z_0)]^T [T'_i(x_0, y_0, z_0)] &\rightarrow [K_{ii}] \\ k_p [T'_i(x_0, y_0, z_0)]^T [T'_j(x_0, y_0, z_0)] &\rightarrow [K_{ij}] \\ k_p [T'_j(x_0, y_0, z_0)]^T [T'_i(x_0, y_0, z_0)] &\rightarrow [K_{ji}] \\ k_p [T'_j(x_0, y_0, z_0)]^T [T'_j(x_0, y_0, z_0)] &\rightarrow [K_{jj}] \end{aligned} \quad (3.59)$$

For the case that point (x_0, y_0, z_0) is not fixed in one or two directions, the spring stiffness for the corresponding directions can be assigned as zero instead of k_p to allow displacement.

Fixed line

It is assumed that the line L_k located within element e_k is fixed in x , y and z directions. Similarly, spring with stiffness k_l is added in each direction. According to Eq. (3.24), the constraining force $[F_{clk}] = (f_{clx}, f_{cly}, f_{clz})^T$ on line L_k can be expressed as:

$$\begin{pmatrix} f_{clx} \\ f_{cly} \\ f_{clz} \end{pmatrix} = \begin{pmatrix} -k_l U_{xk} \\ -k_l U_{yk} \\ -k_l U_{zk} \end{pmatrix} = -k_l [U_k] = -k_l \sum_i [T_i] [D_i] \quad (3.60)$$

Based on Eqs. (3.24) and (3.60), the potential energy Π_{Fclk} contributed by fixed line can be expressed as:

$$\begin{aligned}
 \Pi_{F_{clk}} &= -\frac{1}{2} \iiint_{L_k} (f_{clx} \cdot U_{xk} + f_{cly} \cdot U_{yk} + f_{clz} \cdot U_{zk}) dx dy dz \\
 &= -\frac{1}{2} \iiint_{L_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{clx} \\ f_{cly} \\ f_{clz} \end{pmatrix} dx dy dz \\
 &= \frac{1}{2} \iiint_{L_k} \left(\sum_i [T_i] [D_i] \right)^T \left(k_l \sum_i [T_i] [D_i] \right) dx dy dz \\
 &= \frac{1}{2} k_l \iiint_{L_k} \sum_i \sum_j [D_i]^T [T_i]^T [T_j] [D_j] dx dy dz \\
 &= \frac{1}{2} k_l \sum_i \sum_j [D_i]^T \iiint_{L_k} [T_i]^T [T_j] dx dy dz [D_j]
 \end{aligned} \tag{3.61}$$

According to the principle of virtual work, the derivative of $\Pi_{F_{clk}}$ with respect to the unknowns d_{ir} and d_{js} ($r, s = 1, 2, \dots, 3m$) can be obtained as:

$$k_{ijrs} = \frac{\partial^2 \Pi_{F_{clk}}}{\partial d_{ir} \partial d_{js}} = \frac{1}{2} \frac{\partial^2}{\partial d_{ir} \partial d_{js}} \left\{ k_l \sum_i \sum_j [D_i]^T \iiint_{L_k} [T_i]^T [T_j] dx dy dz [D_j] \right\} \tag{3.62}$$

where k_{ijrs} is added into the element in r th row and s th column of matrix $[K_{ij}]$.

Similar with the situation for elastic stiffness, the elements generated by Eq. (3.62) are added into one, four, sixteen and sixty-four matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering fixed line are added into the corresponding matrices as follows:

$$\begin{aligned}
 k_l \iiint_{L_k} [T_i]^T [T_i] dx dy dz &\rightarrow [K_{ii}] \\
 k_l \iiint_{L_k} [T_i]^T [T_j] dx dy dz &\rightarrow [K_{ij}] \\
 k_l \iiint_{L_k} [T_j]^T [T_i] dx dy dz &\rightarrow [K_{ji}] \\
 k_l \iiint_{L_k} [T_j]^T [T_j] dx dy dz &\rightarrow [K_{jj}]
 \end{aligned} \tag{3.63}$$

For the case that the line L_k is not fixed in one or two directions, the spring stiffness for the corresponding directions can be assigned as zero instead of k_l to allow displacement.

Fixed plane

It is assumed that the plane S_k located within element e_k is fixed in x , y and z directions. Similarly, spring with stiffness k_s is added in each direction. According to Eq. (3.24), the constraining force $[F_{csk}] = (f_{csx}, f_{csy}, f_{csz})^T$ on plane S_k can be expressed as:

$$\begin{pmatrix} f_{csx} \\ f_{csy} \\ f_{csz} \end{pmatrix} = \begin{pmatrix} -k_s U_{xk} \\ -k_s U_{yk} \\ -k_s U_{zk} \end{pmatrix} = -k_s [U_k] = -k_s \sum_i [T_i^T] [D_i] \quad (3.64)$$

Based on Eqs. (3.24) and (3.64), the potential energy Π_{Fcsk} contributed by fixed plane can be expressed as:

$$\begin{aligned} \Pi_{Fcsk} &= -\frac{1}{2} \iiint_{S_k} (f_{csx} \cdot U_{xk} + f_{csy} \cdot U_{yk} + f_{csz} \cdot U_{zk}) dx dy dz \\ &= -\frac{1}{2} \iiint_{S_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{csx} \\ f_{csy} \\ f_{csz} \end{pmatrix} dx dy dz \\ &= \frac{1}{2} \iiint_{S_k} \left(\sum_i [T_i^T] [D_i] \right)^T \left(k_s \sum_i [T_i^T] [D_i] \right) dx dy dz \\ &= \frac{1}{2} k_s \iiint_{S_k} \sum_i \sum_j [D_i]^T [T_i]^T [T_j'] [D_j] dx dy dz \\ &= \frac{1}{2} k_s \sum_i \sum_j [D_i]^T \iiint_{S_k} [T_i]^T [T_j'] dx dy dz [D_j] \end{aligned} \quad (3.65)$$

According to the principle of virtual work, the derivative of Π_{Fcsk} with respect to the unknowns d_{ir} and d_{js} ($r, s = 1, 2, \dots, 3m$) can be obtained as:

$$k_{ijrs} = \frac{\partial^2 \Pi_{Fcsk}}{\partial d_{ir} \partial d_{js}} = \frac{1}{2} \frac{\partial^2}{\partial d_{ir} \partial d_{js}} \left\{ k_s \sum_i \sum_j [D_i]^T \iiint_{S_k} [T_i]^T [T_j'] dx dy dz [D_j] \right\} \quad (3.66)$$

where k_{ijrs} is added into the element in r th row and s th column of matrix $[K_{ij}]$.

Similar with the situation for elastic stiffness, the elements generated by Eq. (3.66) are added into one, four, sixteen and sixty-four matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by considering fixed plane are added into the corresponding matrices as follows:

$$\begin{aligned}
 k_s \iiint_{S_k} [T_i']^T [T_i'] dx dy dz &\rightarrow [K_{ii}] \\
 k_s \iiint_{S_k} [T_i']^T [T_j'] dx dy dz &\rightarrow [K_{ij}] \\
 k_s \iiint_{S_k} [T_j']^T [T_i'] dx dy dz &\rightarrow [K_{ji}] \\
 k_s \iiint_{S_k} [T_j']^T [T_j'] dx dy dz &\rightarrow [K_{jj}]
 \end{aligned} \tag{3.67}$$

For the case that the plane S_k is not fixed in one or two directions, the spring stiffness for the corresponding directions can be assigned as zero instead of k_s to allow displacement.

Inertia force

When carrying out dynamic analysis with flat-top PU based high-order approximation, inertia force is considered for every element. The time interval for each time step is assigned as Δt . According to Eq. (3.24), at the beginning of current time step, the displacement $[U_k(0)]$ for element e_k can be expressed as:

$$[U_k(0)] = \sum_i [T_i'] [D_i(0)] = 0 \tag{3.68}$$

where $[D_i(0)] = 0$ is the value for unknowns at the beginning of current time step. Correspondingly, the displacement $[U_k(\Delta t)]$ for element e_k at the end of current time step can be expressed as:

$$[U_k(\Delta t)] = \sum_i [T_i'] [D_i(\Delta t)] \tag{3.69}$$

where $[D_i(\Delta t)]$ is the value for unknowns at the end of current time step.

From Eqs. (3.68) and (3.69), it can be observed that $[D_i]$ is a function of Δt . While for the matrix $[T_i']$, Δt only influences the shape function w_{fpi} as indicated by Eq. (3.24), which is negligible since small displacement and small deformation are satisfied in each time step. Therefore, Eq. (3.69) can be approximated by its second-order Taylor series as:

$$\sum_i [T_i'] [D_i(\Delta t)] = \sum_i [T_i'] [D_i(0)] + (\Delta t) \sum_i [T_i'] \frac{\partial [D_i(0)]}{\partial (\Delta t)} + \frac{(\Delta t)^2}{2} \sum_i [T_i'] \frac{\partial^2 [D_i(0)]}{\partial (\Delta t)^2} \tag{3.70}$$

According to Eq. (3.70) and the given condition $[D_i(0)] = 0$, the second-order partial derivative of displacement $[U_k(0)]$, i.e. the acceleration of element e_k at the beginning of current time step, can be derived as:

$$\sum_i [T'_i] \frac{\partial^2 [D_i(0)]}{\partial (\Delta t)^2} = \frac{2}{(\Delta t)^2} \sum_i [T'_i] [D_i(\Delta t)] - \frac{2}{\Delta t} \sum_i [T'_i] \frac{\partial [D_i(0)]}{\partial (\Delta t)} \quad (3.71)$$

Meanwhile, the inertia force $[F_{ink}] = (f_{inx}, f_{iny}, f_{inz})^T$ for element e_k at the beginning of current time step is expressed as:

$$\begin{pmatrix} f_{inx} \\ f_{iny} \\ f_{inz} \end{pmatrix} = \begin{pmatrix} -\rho_k \frac{\partial^2 U_{xk}(0)}{\partial (\Delta t)^2} \\ -\rho_k \frac{\partial^2 U_{yk}(0)}{\partial (\Delta t)^2} \\ -\rho_k \frac{\partial^2 U_{zk}(0)}{\partial (\Delta t)^2} \end{pmatrix} = -\rho_k \frac{\partial^2 U_k(0)}{\partial (\Delta t)^2} = -\rho_k \sum_i [T'_i] \frac{\partial^2 [D_i(0)]}{\partial (\Delta t)^2} \quad (3.72)$$

where ρ_k represents the density for element e_k . Substituting Eq. (3.71) into Eq. (3.72), the inertia force $[F_{ink}]$ can be expressed as:

$$\begin{aligned} [F_{ink}] &= -\rho_k \left(\frac{2}{(\Delta t)^2} \sum_i [T'_i] [D_i(\Delta t)] - \frac{2}{\Delta t} \sum_i [T'_i] \frac{\partial [D_i(0)]}{\partial (\Delta t)} \right) \\ &= -\rho_k \left(\frac{2}{(\Delta t)^2} \sum_i [T'_i] [D_i(\Delta t)] - \frac{2}{\Delta t} \sum_i [T'_i] [D'_i(0)] \right) \end{aligned} \quad (3.73)$$

where $[D'_i(0)] = \partial [D_i(0)] / \partial (\Delta t)$ represents the coefficients of velocity polynomial for flat-top PU unit u_{fpi} at the beginning of current time step.

In flat-top PU based high-order approximation, for element e_k , the relationship between the velocity at the beginning of last time step $[V_k(0)]_{-1}$, the velocity at the end of last time step $[V_k(\Delta t)]_{-1}$ and the displacement at the end of last time step $[U_k(\Delta t)]_{-1}$ can be expressed as:

$$\begin{aligned}
 [U_k(\Delta t)]_{-1} &= \frac{\Delta t}{2} ([V_k(0)]_{-1} + [V_k(\Delta t)]_{-1}) \\
 &= \frac{\Delta t}{2} ([V_k(0)]_{-1} + [V_k(0)]) \\
 &= \frac{\Delta t}{2} \left(\frac{\partial [U_k(0)]_{-1}}{\partial(\Delta t)} + \frac{\partial [U_k(0)]}{\partial(\Delta t)} \right) \\
 &= \frac{\Delta t}{2} \left(\sum_i [T'_i] \frac{\partial [D_i(0)]_{-1}}{\partial(\Delta t)} + \sum_i [T'_i] \frac{\partial [D_i(0)]}{\partial(\Delta t)} \right) \\
 &= \frac{\Delta t}{2} \left(\sum_i [T'_i] [D'_i(0)]_{-1} + \sum_i [T'_i] [D'_i(0)] \right)
 \end{aligned} \tag{3.74}$$

where the velocity at the beginning of current time step $[V_k(0)]$ is equal to the velocity at the end of last time step $[V_k(\Delta t)]_{-1}$; the subscript -1 represents the corresponding matrix in last time step. Meanwhile, for element e_k , according to Eq. (3.69), the displacement at the end of last time step $[U_k(\Delta t)]_{-1}$ can be expressed as

$$[U_k(\Delta t)]_{-1} = \sum_i [T'_i] [D_i(\Delta t)]_{-1} \tag{3.75}$$

Combining Eqs. (3.74) and (3.75), it can be derived that:

$$\sum_i [T'_i] [D'_i(0)] = \frac{2}{\Delta t} \sum_i [T'_i] [D_i(\Delta t)]_{-1} - \sum_i [T'_i] [D'_i(0)]_{-1} \tag{3.76}$$

which means that:

$$[D'_i(0)] = \frac{2}{\Delta t} [D_i(\Delta t)]_{-1} - [D'_i(0)]_{-1} \tag{3.77}$$

Therefore, the coefficients of velocity polynomial $[D'_i(0)]$ in Eq. (3.73) can be obtained by the numerical results in last time step, and the coefficients of velocity polynomial at the beginning of first time step is provided as initial velocity in input file. Therefore, according to Eqs. (3.24) and (3.73), the potential energy Π_{Fink} for element e_k , which is contributed by inertia force, can be expressed as:

$$\begin{aligned}
 \Pi_{F_{ink}} &= -\iiint_{V_k} (f_{inx} \cdot U_{xk} + f_{iny} \cdot U_{yk} + f_{inz} \cdot U_{zk}) dx dy dz \\
 &= -\iiint_{V_k} \begin{pmatrix} U_{xk} & U_{yk} & U_{zk} \end{pmatrix} \begin{pmatrix} f_{inx} \\ f_{iny} \\ f_{inz} \end{pmatrix} dx dy dz \\
 &= \iiint_{V_k} \left(\sum_i [T_i] [D_i] \right)^T \left(\rho_k \begin{pmatrix} \frac{2}{(\Delta t)^2} \sum_i [T_i'] [D_i(\Delta t)] - \\ \frac{2}{\Delta t} \sum_i [T_i'] [D_i'(0)] \end{pmatrix} \right) dx dy dz \\
 &= \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} \left(\sum_i [T_i] [D_i] \right)^T \left(\sum_i [T_i'] [D_i] \right) dx dy dz - \\
 &\quad \frac{2\rho_k}{\Delta t} \iiint_{V_k} \left(\sum_i [T_i] [D_i] \right)^T \left(\sum_i [T_i'] [D_i'(0)] \right) dx dy dz \\
 &= \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} \sum_i \sum_j [D_i]^T [T_i]^T [T_j'] [D_j] dx dy dz - \\
 &\quad \frac{2\rho_k}{\Delta t} \iiint_{V_k} \sum_i \sum_j [D_i]^T [T_i]^T [T_j'] [D_j'(0)] dx dy dz \\
 &= \frac{2\rho_k}{(\Delta t)^2} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T_j'] dx dy dz [D_j] - \\
 &\quad \frac{2\rho_k}{\Delta t} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T_j'] dx dy dz [D_j'(0)]
 \end{aligned} \tag{3.78}$$

where $[D_i] = [D_i(\Delta t)]$ are unknowns in current time step when assembling the global equilibrium equations. According to the principle of virtual work, the derivative of Π_{Fink} with respect to the unknown d_{ir} ($r = 1, 2, \dots, 3m$) can be obtained as:

$$\frac{\partial \Pi_{Fink}}{\partial d_{ir}} = k_{ir} - f_{ir} = \frac{\partial}{\partial d_{ir}} \left\{ \begin{aligned} &\frac{2\rho_k}{(\Delta t)^2} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T_j'] dx dy dz [D_j] - \\ &\frac{2\rho_k}{\Delta t} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T_j'] dx dy dz [D_j'(0)] \end{aligned} \right\} \tag{3.79}$$

where

$$k_{ir} = \frac{\partial}{\partial d_{ir}} \left\{ \frac{2\rho_k}{(\Delta t)^2} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T_j'] dx dy dz [D_j] \right\} \tag{3.80}$$

$$f_{ir} = \frac{\partial}{\partial d_{ir}} \left\{ \frac{2\rho_k}{\Delta t} \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T'_j] dx dy dz [D'_j(0)] \right\} \quad (3.81)$$

It can be observed from Eq. (3.80) that the unknown $[D_j]$ is still included after the first-order derivative. Therefore, further derivative of k_{ir} with respect to the unknown d_{js} ($s = 1, 2, \dots, 3m$) is carried out, and the following result can be obtained:

$$k_{ijrs} = \frac{\partial k_{ir}}{\partial d_{js}} = \frac{2\rho_k}{(\Delta t)^2} \frac{\partial^2}{\partial d_{ir} \partial d_{js}} \left\{ \sum_i \sum_j [D_i]^T \iiint_{V_k} [T_i]^T [T'_j] dx dy dz [D_j] \right\} \quad (3.82)$$

where k_{ijrs} is added into the element in r th row and s th column of matrix $[K_{ij}]$. Similar with the situation for elastic stiffness, the elements generated by Eq. (3.82) are added into one, four, sixteen and sixty-four matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by Eq. (3.82) are added into the corresponding matrices as follows:

$$\begin{aligned} \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T_i]^T [T'_i] dx dy dz &\rightarrow [K_{ii}] \\ \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T_i]^T [T'_j] dx dy dz &\rightarrow [K_{ij}] \\ \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_j]^T [T'_i] dx dy dz &\rightarrow [K_{ji}] \\ \frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_j]^T [T'_j] dx dy dz &\rightarrow [K_{jj}] \end{aligned} \quad (3.83)$$

Moreover, the f_{ir} in Eq. (3.81) is added into the element in r th row of matrix $[F_i]$. Similar with the situation for initial stress, the elements generated by Eq. (3.81) are added into one, two, four and eight matrices for the case of flat-top element, blue-colored PU element, orange-colored PU element and white-colored PU element, respectively. In summary, the elements generated by Eq. (3.81) are added into the corresponding matrices as:

$$\frac{2\rho_k}{\Delta t} \sum_j \iiint_{V_k} [T_i]^T [T'_j] dx dy dz [D'_j(0)] \rightarrow [F_i] \quad (3.84)$$

Summary

Based on the derivation of sub-matrices in this section, a brief summary can be presented as listed in Table 3.1, which includes the formula of sub-matrices derived according to ten mechanical conditions, i.e. elastic stiffness, initial stress, point loading, line loading, plane loading, volume loading, fixed point, fixed line, fixed plane and inertia force, respectively.

Table 3.1 Sub-matrices for flat-top PU based high-order approximation

Mechanical condition		Sub-matrix
Elastic stiffness		$\iiint_{V_k} [B_i]^T [E_k] [B_i] dxdydz \rightarrow [K_{ii}]$
		$\iiint_{V_k} [B_i]^T [E_k] [B_j] dxdydz \rightarrow [K_{ij}]$
		$\iiint_{V_k} [B_j]^T [E_k] [B_i] dxdydz \rightarrow [K_{ji}]$
		$\iiint_{V_k} [B_j]^T [E_k] [B_j] dxdydz \rightarrow [K_{jj}]$
Initial stress	Constant form	$\iiint_{V_k} [B_i]^T dxdydz [\sigma_k^0] \rightarrow [F_i]$
	Polynomial form	$\sum_j \left(\iiint_{V_k} [B_i]^T [E_k] [B_j] dxdydz [D_j]_t \right) \rightarrow [F_i]$
Point loading		$[T'_i(x_0, y_0, z_0)]^T [F_{pk}] \rightarrow [F_i]$
Line loading		$\iiint_{L_k} [T'_i]^T dxdydz [F_{lk}] \rightarrow [F_i]$
Plane loading		$\iiint_{S_k} [T'_i]^T dxdydz [F_{sk}] \rightarrow [F_i]$
Volume loading		$\iiint_{V_k} [T'_i]^T dxdydz [F_{vk}] \rightarrow [F_i]$

Fixed point	$k_p [T'_i(x_0, y_0, z_0)]^T [T'_i(x_0, y_0, z_0)] \rightarrow [K_{ii}]$ $k_p [T'_i(x_0, y_0, z_0)]^T [T'_j(x_0, y_0, z_0)] \rightarrow [K_{ij}]$ $k_p [T'_j(x_0, y_0, z_0)]^T [T'_i(x_0, y_0, z_0)] \rightarrow [K_{ji}]$ $k_p [T'_j(x_0, y_0, z_0)]^T [T'_j(x_0, y_0, z_0)] \rightarrow [K_{jj}]$
Fixed line	$k_l \iiint_{L_k} [T'_i]^T [T'_i] dx dy dz \rightarrow [K_{ii}]$ $k_l \iiint_{L_k} [T'_i]^T [T'_j] dx dy dz \rightarrow [K_{ij}]$ $k_l \iiint_{L_k} [T'_j]^T [T'_i] dx dy dz \rightarrow [K_{ji}]$ $k_l \iiint_{L_k} [T'_j]^T [T'_j] dx dy dz \rightarrow [K_{jj}]$
Fixed plane	$k_s \iiint_{S_k} [T'_i]^T [T'_i] dx dy dz \rightarrow [K_{ii}]$ $k_s \iiint_{S_k} [T'_i]^T [T'_j] dx dy dz \rightarrow [K_{ij}]$ $k_s \iiint_{S_k} [T'_j]^T [T'_i] dx dy dz \rightarrow [K_{ji}]$ $k_s \iiint_{S_k} [T'_j]^T [T'_j] dx dy dz \rightarrow [K_{jj}]$
Inertia force	$\frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_i]^T [T'_i] dx dy dz \rightarrow [K_{ii}]$ $\frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_i]^T [T'_j] dx dy dz \rightarrow [K_{ij}]$ $\frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_j]^T [T'_i] dx dy dz \rightarrow [K_{ji}]$ $\frac{2\rho_k}{(\Delta t)^2} \iiint_{V_k} [T'_j]^T [T'_j] dx dy dz \rightarrow [K_{jj}]$
	$\frac{2\rho_k}{\Delta t} \sum_j \iiint_{V_k} [T'_i]^T [T'_j] dx dy dz [D'_j(0)] \rightarrow [F_i]$

Assuming there are totally n flat-top PU units in the numerical model, the submatrices listed in Table 3.1 are assembled into the global equilibrium equations for each unit, which can be expressed as follows:

$$\begin{pmatrix} [K_{11}] & [K_{12}] & \cdots & [K_{1n}] \\ [K_{21}] & [K_{22}] & \cdots & [K_{2n}] \\ \vdots & \vdots & \ddots & \vdots \\ [K_{n1}] & [K_{n2}] & \cdots & [K_{nn}] \end{pmatrix} \begin{pmatrix} [D_1] \\ [D_2] \\ \vdots \\ [D_n] \end{pmatrix} = \begin{pmatrix} [F_1] \\ [F_2] \\ \vdots \\ [F_n] \end{pmatrix} \quad (3.85)$$

After solving Eq. (3.85), the unknowns $[D_i]$ for each flat-top PU unit u_{fpi} can be obtained. Therefore, the global displacement, stress and strain field for unit u_{fpi} can be derived by substituting the solved unknowns into Eqs. (3.24), (3.30) and (3.31), respectively.

From Table 3.1, it can be observed that, in the process of assembling global equilibrium equations, integration of high-order polynomial on line, plane and volume in three-dimensional space is required. In flat-top PU based high-order approximation, for the purpose of improving accuracy, reducing space for data storage and saving time, the recursive formula for simplex integration (Lin, 2005) is adopted, where the integration of higher order monomial can take advantage of the results calculated for the lower order. For completeness, the recursive formula for simplex integration is briefly presented here.

In three-dimensional space, for the integration of monomial over volume, each element e_k is discretized into several simplexes, i.e. tetrahedrons. Then, the integration is carried out on these simplexes one by one. The recursive formula for simplex integration of monomial $x^{\alpha j} y^{\beta j} z^{\gamma j}$ over the volume V_s of a tetrahedron can be expressed as:

$$\iiint_{V_s} x^{\alpha j} y^{\beta j} z^{\gamma j} dx dy dz = \frac{(n_v - 1)! vol}{(\alpha j + \beta j + \gamma j + n_v - 1)!} S_{\alpha j, \beta j, \gamma j} \quad (3.86)$$

where $n_v = 4$ is the number of vertices for simplex; vol is the volume of simplex, which can be calculated as:

$$vol = \frac{1}{6} \left| \det \begin{pmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{pmatrix} \right| \quad (3.87)$$

where (x_i, y_i, z_i) ($i = 1, 2, 3, 4$) are the coordinates for four vertices of the simplex; $S_{\alpha j, \beta j, \gamma j}$ can be calculated as follows:

$$\begin{aligned} S_{i,0,0} &= \sum_{l=1}^i C_{i-1}^{l-1} e_{l,0,0} S_{i-l,0,0} \quad (i=1 \sim \alpha j) \\ S_{i,j,0} &= \sum_{m=1}^j C_{j-1}^{m-1} \sum_{l=0}^i C_i^l e_{l,m,0} S_{i-l,j-m,0} \quad (i=0 \sim \alpha j, j=1 \sim \beta j) \\ S_{i,j,k} &= \sum_{n=1}^k C_{k-1}^{n-1} \sum_{m=0}^j C_j^m \sum_{l=0}^i C_i^l e_{l,m,n} S_{i-l,j-m,k-n} \quad (i=0 \sim \alpha j, j=0 \sim \beta j, k=1 \sim \gamma j) \end{aligned} \quad (3.88)$$

where

$$S_{0,0,0} = 1 \quad (3.89)$$

$$C_i^l = \frac{i!}{l!(i-l)!} \quad (i \geq l) \quad (3.90)$$

$$e_{l,m,n} = (l+m+n-1)! [E]^T \left([X]^{(l)} \circ [Y]^{(m)} \circ [Z]^{(n)} \right) \quad (3.91)$$

where

$$[X] = \begin{pmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \end{pmatrix} \quad [Y] = \begin{pmatrix} y_1 \\ y_2 \\ y_3 \\ y_4 \end{pmatrix} \quad [Z] = \begin{pmatrix} z_1 \\ z_2 \\ z_3 \\ z_4 \end{pmatrix} \quad [E] = \begin{pmatrix} 1 \\ 1 \\ 1 \\ 1 \end{pmatrix} \quad (3.92)$$

A brief introduction for Hadamard power and Hadamard product is presented here in order to calculate $[X]^{(l)} \circ [Y]^{(m)} \circ [Z]^{(n)}$ in Eq. (3.91). It is assumed that $[A]$ and $[B]$ are matrices with the same dimension as follows:

$$[A] = \begin{pmatrix} a_{11} & a_{12} & \cdots & a_{1n} \\ a_{21} & a_{22} & \cdots & a_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ a_{m1} & a_{m2} & \cdots & a_{mn} \end{pmatrix} \quad [B] = \begin{pmatrix} b_{11} & b_{12} & \cdots & b_{1n} \\ b_{21} & b_{22} & \cdots & b_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ b_{m1} & b_{m2} & \cdots & b_{mn} \end{pmatrix} \quad (3.93)$$

Therefore, the l th Hadamard power of matrix $[A]$, i.e. $[A]^{(l)}$, can be calculated as:

$$[A]^{(l)} = \begin{pmatrix} a_{11}^l & a_{12}^l & \cdots & a_{1n}^l \\ a_{21}^l & a_{22}^l & \cdots & a_{2n}^l \\ \vdots & \vdots & \ddots & \vdots \\ a_{m1}^l & a_{m2}^l & \cdots & a_{mn}^l \end{pmatrix} \quad (3.94)$$

And the Hadamard product of matrices $[A]$ and $[B]$, i.e. $[A] \circ [B]$, can be calculated as:

$$[A] \circ [B] = \begin{pmatrix} a_{11}b_{11} & a_{12}b_{12} & \cdots & a_{1n}b_{1n} \\ a_{21}b_{21} & a_{22}b_{22} & \cdots & a_{2n}b_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ a_{m1}b_{m1} & a_{m2}b_{m2} & \cdots & a_{mn}b_{mn} \end{pmatrix} \quad (3.95)$$

However, it can be observed from Table 3.1 that the integration of monomial over three-dimensional plane and line is also required, which is essentially not simplex integration.

For the case of integration over a three-dimensional plane, the plane is firstly discretized into triangles. Then the integration over the domain S of a triangle can be calculated by modifying Eq. (3.86) as:

$$\iiint_S x^{\alpha j} y^{\beta j} z^{\gamma j} dx dy dz = \frac{(n_v - 1)! A}{(\alpha j + \beta j + \gamma j + n_v - 1)!} S_{\alpha j, \beta j, \gamma j} \quad (3.96)$$

where $n_v = 3$ is the number of vertices for triangle; A is the area of triangle, which can be calculated as:

$$A = \frac{1}{2} \left| \det \begin{pmatrix} \vec{i} & \vec{j} & \vec{k} \\ (x_2 - x_1) & (y_2 - y_1) & (z_2 - z_1) \\ (x_3 - x_1) & (y_3 - y_1) & (z_3 - z_1) \end{pmatrix} \right| \quad (3.97)$$

where (x_i, y_i, z_i) ($i = 1, 2, 3$) are the coordinates for three vertices of the triangle; $(\vec{i}, \vec{j}, \vec{k})$ represent the unit basis vectors. $S_{\alpha j, \beta j, \gamma j}$ in Eq. (3.96) is still calculated by Eqs. (3.88) to (3.91), but the matrices $[X]$, $[Y]$, $[Z]$ and $[E]$ in Eq. (3.91) becomes:

$$[X] = \begin{pmatrix} x_1 \\ x_2 \\ x_3 \end{pmatrix} \quad [Y] = \begin{pmatrix} y_1 \\ y_2 \\ y_3 \end{pmatrix} \quad [Z] = \begin{pmatrix} z_1 \\ z_2 \\ z_3 \end{pmatrix} \quad [E] = \begin{pmatrix} 1 \\ 1 \\ 1 \end{pmatrix} \quad (3.98)$$

For the case of integration over a three-dimensional line L , the result can be obtained by modifying Eq. (3.86) as:

$$\iiint_L x^{\alpha j} y^{\beta j} z^{\gamma j} dx dy dz = \frac{(n_v - 1)! l}{(\alpha j + \beta j + \gamma j + n_v - 1)!} S_{\alpha j, \beta j, \gamma j} \quad (3.99)$$

where $n_v = 2$ is the number of vertices for line; l is the length of line L , which can be calculated as:

$$l = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2 + (z_2 - z_1)^2} \quad (3.100)$$

where (x_i, y_i, z_i) ($i = 1, 2$) are the coordinates for two vertices of line L . $S_{\alpha j, \beta j, \gamma j}$ in Eq. (3.99) is still calculated by Eqs. (3.88) to (3.91), but the matrices $[X]$, $[Y]$, $[Z]$ and $[E]$ in Eq. (3.91) becomes:

$$[X] = \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} \quad [Y] = \begin{pmatrix} y_1 \\ y_2 \end{pmatrix} \quad [Z] = \begin{pmatrix} z_1 \\ z_2 \end{pmatrix} \quad [E] = \begin{pmatrix} 1 \\ 1 \end{pmatrix} \quad (3.101)$$

3.4 Numerical analysis procedure

Based on the introduction in Sections 3.2 and 3.3, the numerical analysis procedure for flat-top PU based high-order approximation can be summarized as follows:

Step 1: The numerical model is generated through intersecting the flat-top PU mesh with physical boundary of the problem domain, as introduced in Section 3.2.

Step 2: Within each time step, the material properties, loading conditions and boundary conditions are assigned for the numerical model.

Step 3: High-order polynomial is assigned as local displacement approximation for each flat-top PU unit in the numerical model, and the coefficients for monomials are denoted as unknowns. The shape function is derived for every flat-top PU unit as introduced in Section 3.2. Then, the global displacement approximation can be obtained by multiplying the local approximation with the corresponding shape function, as introduced in Section 3.3.1. Based on the global displacement approximation, the stress/strain field can be derived according to the formula in Section 3.3.2.

Step 4: Based on the principle of virtual work and simplex integration in Section 3.3.3, the global equilibrium equations are established by assembling the sub-matrices listed in Table 3.1 for every flat-top PU unit.

Step 5: After solving the global equilibrium equations, the displacement and stress/strain fields can be obtained by substituting the solved unknowns into the corresponding polynomial functions.

Step 6: All the coordinates for the element nodes, loading control points and boundary control points are updated by calculating the corresponding displacement within current time step. Meanwhile, the initial stress and the velocity are also updated.

Step 7: The numerical analysis will be stopped, if anyone of the following two conditions is satisfied: 1) the displacement within current time step is negligible compared with the corresponding total displacement; 2) the required number of time steps is reached. Otherwise, the simulation goes to Step 2.

3.5 Numerical examples

In this section, several examples are simulated and analyzed to verify the various matrices listed in Table 3.1 and to assess the accuracy of flat-top PU based high-order approximation. A brief summary of the examples is presented in Table 3.2.

Table 3.2 A brief summary of the examples

Title	Dimension	Objective
Simply supported beam	1D	Verify the numerical accuracy and convergence for problem with even function as analytical solution
		Study the influence of relative size between flat-top elements and PU elements
MacNeal and Harder's slender beam	1D	Verify the numerical accuracy and convergence for problem with large aspect ratio
Curved cantilever beam	2D	Verify the numerical accuracy and convergence for concave structure
Twisted cantilever beam	3D	Verify the numerical accuracy and convergence for twisted structure
Beam-to-column connection	2D	Verify the numerical accuracy and convergence for the 2D combination of beam and column

Cook's problem	2D	Take degree $p = 2$ as an example to demonstrate the numerical analysis procedure
		Verify the numerical accuracy and convergence for a plate
		Study the influence of relative size between flat-top elements and PU elements
Thin plate	2D	Verify the numerical accuracy and convergence for a plate with non-polynomial analytical solution
Table-like structure	3D	Verify the numerical accuracy and convergence for the 3D combination of plate and columns

3.5.1 Simply-supported beam

As illustrated in Figure 3.15, a simply-supported beam subjected to uniformly distributed loading q on the top surface is simulated. The dimensions for this beam are $10m \times 1m \times 1m$. The loading q is pointing downward with magnitude as $10kPa$. The material is assumed to be linear elastic with Young's modulus E and Poisson's ratio ν as $20GPa$ and 0.2 , respectively. The central lines of the left and right side surfaces are fixed as shown in Figure 3.15. So that the beam is fixed in x , y and z directions, while rotation around the fixed central line is permissible. The stiffness for penalty springs is taken as $10E$. The coordinate system is defined as follows: the origin is located at the center of left side surface, the x and z axes are pointing rightward and downward, respectively.

Figure 3.15 A simply-supported beam subjected to uniformly distributed loading on the top surface

Since both the geometry and loading do not change along with the y -axis, the analytical solution for displacement (U_x , U_y , U_z) can be derived based on a two-dimensional vertical section ($0 \leq x \leq 10$, $y = 0$, $-0.5 \leq z \leq 0.5$) as follows (Timoshenko and Goodier, 1951):

$$\begin{aligned}
 U_x &= \frac{q}{2EI} \left\{ \left[\left(\frac{l^2}{4} - \frac{h^2}{10} \right) \left(x - \frac{l}{2} \right) - \frac{1}{3} \left(x - \frac{l}{2} \right)^3 \right] z + \right. \\
 &\quad \left. \frac{2}{3} \left(x - \frac{l}{2} \right) z^3 + \nu \left(x - \frac{l}{2} \right) \left(\frac{z^3}{3} - \frac{h^2 z}{4} + \frac{h^3}{12} \right) \right\} \\
 U_y &= 0 \\
 U_z &= -\frac{q}{2EI} \left\{ \frac{z^4}{12} - \frac{h^2 z^2}{8} + \frac{h^3 z}{12} + \nu \left[\left(\frac{l^2}{4} - \left(x - \frac{l}{2} \right)^2 \right) \frac{z^2}{2} + \frac{z^4}{6} - \frac{h^2 z^2}{20} \right] \right\} - \\
 &\quad \frac{q}{2EI} \left[\frac{l^2}{8} \left(x - \frac{l}{2} \right)^2 - \frac{1}{12} \left(x - \frac{l}{2} \right)^4 + \left(\frac{1}{5} + \frac{\nu}{8} \right) h^2 \left(x - \frac{l}{2} \right)^2 \right] + \\
 &\quad \frac{5ql^4}{384EI} \left[1 + \frac{12h^2}{5l^2} \left(\frac{4}{5} + \frac{\nu}{2} \right) \right]
 \end{aligned} \tag{3.102}$$

where $l = 10m$ and $h = 1m$ are the length and height of the beam, respectively; $I = bh^3/12$ is the moment of inertia of the cross section, in which $b = 1m$ is the width of the beam.

The deflection of the beam along the longitudinal axis, i.e. x -axis, can be deduced by substituting the value for each parameter into U_z in Eq. (3.102), which is expressed as:

$$U_z|_{y=0, z=0} = -3 \times 10^{-6} \left[\frac{509}{40} (x-5)^2 - \frac{1}{12} (x-5)^4 \right] + \frac{1277}{1.6 \times 10^6} \tag{3.103}$$

As for the numerical simulation, the simply-supported beam is firstly discretized by a one-element mesh, as illustrated in Figure 3.16 (a), which results in one flat-top PU unit with one flat-top element. The degree p for local approximation polynomial ranges from 1 to 6. The deflection of the beam along its longitudinal axis is numerically obtained and compared with the analytical solution provided by Eq. (3.103) as shown in Figure 3.16 (b). The numerical results for the deflection at the center of simply-supported beam, i.e. point (5, 0, 0), are also compared with the analytical solution in Table 3.3. Convergence of the numerical results is recognized

with the increase of degree p for local approximation polynomial. From Eq. (3.103), it can be observed that the analytical solution is an even function of $(x-5)$, which is well reflected by the numerical results symmetric about $x = 5$. It is also noted that the numerical result is almost equal to zero for $p = 1$, the same value for $p = 2$ and 3, and identical for $p = 4$ and above. That is because the real behavior of the beam is governed by polynomial basis of even degrees, inclusion of any odd degrees will not improve the approximation accuracy. Thus, the odd degrees can be excluded from the local approximations to further improve the computational efficiency. Since the analytical solution is a polynomial of degree 4, the one-element model is able to exactly reproduce the analytical solution when the degree of local approximation polynomial is equal to 4 or above.

(a) One-element model

(b) Numerical results with degree $p = 1 \sim 6$ and analytical solution

Figure 3.16 Numerical model and results generated by one-element mesh

In order to investigate the influence of the mesh configuration on the simulation results, the simply-supported beam is further represented by a three-element model as illustrated in Figure 3.17(a), which includes two flat-top PU units with two flat-

top elements and one PU element. The length for each flat-top element is $l_{ft} = 4.5m$, and for PU element is $l_{pu} = 1m$. The numerical results for three-element model are summarized in Figure 3.17 (b) and Table 3.3. For this case, convergence of the numerical results is also recognized when increasing the local approximation degree p . Further comparison between numerical results from the one- and three-element models shows that the increase of number of flat-top PU units can improve the approximation accuracy.

Lastly, the local approximation degree p is fixed as 4, while the length of PU element l_{pu} takes different values as $1m$, $2m$, $4m$, $6m$ and $8m$, respectively. The results obtained from the five models are almost identical and all of them are consistent with the analytical solution, as illustrated in Figure 3.17(c). It can be concluded that the numerical results are not sensitive to the relative length of flat-top elements and PU elements. Therefore, the construction of flat-top PU mesh is quite flexible.

(a) Three-element model

(b) Numerical results with degree $p = 1 \sim 6$ and analytical solution

(c) Numerical results with different values of l_{pu} ($p = 4$)

Figure 3.17 Numerical model and results generated by three-element mesh

Table 3.3 Vertical displacement at the center of the simply-supported beam

Numerical results							Analytical solution (mm)
p	1	2	3	4	5	6	
Model 1 (mm)	0.000	0.637	0.637	0.798	0.798	0.798	0.798
Error (%)	100	20.2	20.2	0	0	0	
Model 2 (mm)	0.126	0.659	0.798	0.798	0.798	0.798	
Error (%)	84.2	17.4	0	0	0	0	

Note: Model 1: one-element model; Model 2: three-element model with $l_{pu} = 1m$

3.5.2 MacNeal and Harder's slender beam

A slender cantilever beam subjected to a unit in-plane force $P_0 = 1kN$ and a unit out-of-plane force $P_1 = 1kN$ at the free end is studied. As shown in Figure 3.18 (a), the dimensions for this slender beam are $6m \times 0.2m \times 0.1m$. The Young's modulus E and Poisson's ratio ν are $10GPa$ and 0.3 , respectively.

The slender beam is firstly discretized by a one-element mesh, which results in one flat-top PU unit including one flat-top element, as shown in Figure 3.18 (b). The degree p for local approximation polynomial ranges from 1 to 6. The beam tip deflection in the direction of applied loads are numerically obtained and normalized with respect to the reference solutions provided by MacNeal and Harder (1985). As listed in Table 3.4, the convergence is evident with the increase of degree p .

The slender beam is then discretized by a two-element mesh, which results in two flat-top PU units including one flat-top element and one PU element, as shown in Figure 3.18 (c). The length for flat-top element l_{ft} and PU element l_{pu} are both $3m$. At this time, the left side surface of PU element e_{pu1} is designed as fixed plane. The numerical results for this two-element model are also presented in Table 3.4. Convergence with the increase of degree p can also be observed in this case. With further comparison between the numerical results of one- and two-element models, it can be concluded that the approximation accuracy can be improved by increasing the number of flat-top PU units.

The numerical results for slender beam also demonstrate that good approximation accuracy can be achieved even for elements with large aspect ratio, which are 60 and 30 for two numerical models, respectively, in this example.

(a) A slender cantilever beam subjected to unit in- and out-of-plane forces

(b) Model 1: one-element model

(c) Model 2: two-element model

Figure 3.18 A slender cantilever beam and two numerical models

Table 3.4 Normalized numerical results for beam tip deflection

p		1	2	3	4	5	6
In-plane loading	Model 1	0.002	0.737	0.934	0.952	0.965	0.975
	Model 2	0.627	0.916	0.956	0.976	0.985	0.990
Out-of-plane loading	Model 1	0.001	0.713	0.931	0.960	0.971	0.978
	Model 2	0.627	0.862	0.961	0.977	0.985	0.989

Note: results are normalized by the reference solutions: 0.108082 for in-plane loading and 0.4321 for out-of-plane loading (MacNeal and Harder, 1985; Tian et al., 2006)

3.5.3 Curved cantilever beam

A curved cantilever beam subjected to a unit in-plane force $P_0 = 1kPa$ and a unit out-of-plane force $P_1 = 1kPa$ at the free end is considered. As illustrated in Figure 3.19 (a), its inner and outer radii are $4.12m$ and $4.32m$, respectively. The thickness in z direction is equal to $0.1m$ and the arc is designed as 90° . The Young's modulus E and Poisson's ratio ν is $10GPa$ and 0.25 , respectively.

In numerical model, the curved beam is represented by six segments for geometric description purpose. As shown in Figure 3.19 (b), the beam is firstly discretized by a one-element mesh, which results in one flat-top PU unit with one flat-top element. The degree p for local approximation polynomial ranges from 1 to 6. The displacements at the free end in the directions of the applied loads are numerically obtained and normalized with respect to the reference solutions provided by MacNeal and Harder (1985). The results are summarized in Table 3.5. Again, the numerical results are convergent to the reference solutions with the increase of degree p . The example demonstrates that one high-order element is sufficient even for curved and concave structural component.

Then, a two-element mesh, which includes two flat-top PU units with one flat-top element and one PU element, is utilized to discretize this curved beam, as illustrated in Figure 3.19 (c). After intersected by the boundary, the numerical model is generated with point loading applied on the PU element e_{pu1} . The results from this

two-element model are also listed in Table 3.5. Convergence with the increase of degree p is still evident in this case. Through comparing numerical results from the one- and two-element models, it can be observed that the approximation accuracy has been improved by increasing the number of flat-top PU units, especially when degree p is no more than 4.

(a) A curved cantilever beam subjected to unit in- and out-of-plane forces

(b) Model 1: one-element model

(c) Model 2: two-element model

Figure 3.19 A curved cantilever beam and two numerical models

Table 3.5 Normalized numerical results for displacement at free end

p		1	2	3	4	5	6
In-plane loading	Model 1	0.001	0.013	0.478	0.775	0.962	0.974
	Model 2	0.021	0.284	0.802	0.952	0.991	1.001
Out-of-plane loading	Model 1	0.001	0.403	0.626	0.881	0.932	0.943
	Model 2	0.061	0.380	0.840	0.919	0.941	0.943

Note: results are normalized by the reference solutions: 0.08734 for in-plane loading and 0.5022 for out-of-plane loading (MacNeal and Harder, 1985; Tian et al., 2006)

3.5.4 Twisted cantilever beam

A pre-twisted cantilever beam subjected to two unit transverse forces $P_0 = 1kPa$ and $P_1 = 1kPa$ at the free end is examined. As shown in Figure 3.20 (a), the dimensions for twisted beam are $12m \times 1.1m \times 0.32m$ and the twist angle from fixed end to free end is equal to 90° . The Young's modulus E and Poisson's ratio ν are $29GPa$ and 0.22 , respectively.

In numerical model, the twisted beam is represented by several segments for geometric description purpose. As illustrated in Figure 3.20 (b), this beam is firstly discretized by a one-element mesh, which results in one flat-top PU unit with one flat-top element. The degree for local approximation polynomial ranges from 1 to 6. The displacements at the free end in the directions of applied loads are numerically obtained and normalized by the reference solutions provided by MacNeal and Harder (1985), as listed in Table 3.6. The convergence of numerical results is still obvious in this case. This example demonstrates the approximation accuracy and efficiency of the high-order element for twisted structural component.

This twisted beam is further analyzed with a three-element mesh, as shown in Figure 3.20 (c). After intersected by the boundary, a numerical model is generated, which includes three flat-top PU units with one flat-top element and two PU elements. The length for each element is equal to $4m$. In this model, both point loadings and fixed end are applied on PU elements. The numerical results are presented in Table 3.6, where convergence can be observed with the increase of degree p . Through further comparison between the numerical results from two different models, it can be recognized that the approximation accuracy has been improved after increasing the number of flat-top PU units, especially when degree p is no more than 4.

(a) A pre-twisted cantilever beam subjected to two unit transverse forces

(b) Model 1: one-element model

(c) Model 2: three-element model

Figure 3.20 A pre-twisted cantilever beam and two numerical models

Table 3.6 Normalized numerical results for displacement at free end

p		1	2	3	4	5	6
In-plane loading	Model 1	0.018	0.204	0.733	0.918	0.935	0.942
	Model 2	0.541	0.849	0.933	0.942	0.944	0.945
Out-of-plane loading	Model 1	0.005	0.602	0.790	0.877	0.951	0.954
	Model 2	0.368	0.874	0.950	0.954	0.956	0.957

Note: results are normalized by the reference solutions: 0.005424 for in-plane loading and 0.001754 for out-of-plane loading (MacNeal and Harder, 1985; Tian et al., 2006)

3.5.5 Beam-to-column connection

A beam-to-column connection is examined. The geometry shape and point loading $P = 0.5kN$ are illustrated in Figure 3.21 (a). The Young's modulus E and Poisson's ratio ν are $20GPa$ and 0.3 , respectively. The coordinate system is defined as follows: the origin is located at the center of the upper left side, which is marked as red line; the x and z axes are pointing rightward and downward, respectively. The analytical solution is not available for this problem. Thus, the convergence test is carried out using the FEM and it is then used as the reference solution to calibrate the numerical results obtained from flat-top PU based high-order approximation. The reference FEM model discretizes the structure into 658 10-node (second-order) tetrahedral elements with the total number of nodes as 1489 and the total number of DOFs as $1489 \times 3 = 4467$.

(a) A beam-to column connection subjected to point loading

(b) Numerical model for flat-top PU based high-order approximation

Figure 3.21 A beam-to-column connection and its numerical model

The numerical model for flat-top PU based high-order approximation is illustrated in Figure 3.21 (b), where the structure is discretized into two flat-top PU units with two flat-top elements and one PU element. The degree p for local approximation polynomial ranges from 1 to 6. The numerical results for horizontal displacement U_x of the column and vertical displacement U_z of the beam are plotted in Figure 3.22 (a) and (b), respectively, where convergence is observed with the increase of degree p . The numerical results for horizontal displacement U_x at the top of the column and vertical displacement U_z at the free end of the beam are obtained and further compared with the reference FEM solution in Table 3.7, in which the numerical results quickly converge to the reference solution with the increase of

degree p . The flat-top PU based high-order approximation employs two flat-top PU units and thus $84 \times 3 \times 2 = 504$ total number of DOFs ($p = 6$), which is only 11.3% of the FEM model to achieve a comparative accuracy.

(a) Numerical results for horizontal displacement U_x of the column

(b) Numerical results for vertical displacement U_z of the beam

Figure 3.22 Numerical results for horizontal displacement U_x of the column and vertical displacement U_z of the beam

Table 3.7 Horizontal displacement at top of the column (U_x) and vertical displacement at the free end of the beam (U_z)

Flat-top PU based high-order approximation							FEM
p	1	2	3	4	5	6	(mm)
U_x (mm)	0.00	1.01	1.33	1.42	1.45	1.46	1.48
Error (%)	100	31.7	10.1	4.1	2.0	1.3	
U_z (mm)	0.13	2.36	2.79	3.06	3.16	3.17	3.33
Error (%)	96.1	29.1	16.2	8.1	5.1	4.8	

3.5.6 Cook's problem

The Cook's problem (Cook et al., 1989) is studied here with its configuration shown in Figure 3.23 (a). The thickness for problem domain is $1mm$. The left side is fixed and the uniformly distributed shear force $F = 1kN$ is applied on the right side. The Young's modulus E and Poisson's ratio ν is $1kPa$ and $1/3$, respectively.

The problem domain is firstly represented by a one-element model, as illustrated in Figure 3.23 (b), which results in one flat-top PU unit with one flat-top element. The degree p of local approximation polynomial ranges from 1 to 6. The numerical results of displacement at Point A in the direction of applied force are normalized with respect to the reference solution $23.96mm$ (Zheng and Xu, 2014). As listed in Table 3.8, Convergence of the numerical results is recognized with the increase of degree p . The error for the one-element model is within 3.1% when the degree p takes 4 and above.

The problem domain is further discretized by a three-element mesh, as shown in Figure 3.23 (c). After the mesh intersected by the physical boundary, the numerical model is generated in Figure 3.23 (d), which including two flat-top PU units with two flat-top elements and one PU element. Specifically, element e_{f1} together with element e_{pu1} generates unit u_{fp1} , while e_{f2} with e_{pu1} generates u_{fp2} . Each flat-top PU unit is assigned with high-order polynomial as local displacement approximation.

(a) Basic information for Cook's problem

(b) Model 1

(c) three-element mesh

(d) Model 2

Figure 3.23 Configuration and numerical models for Cook's problem

Table 3.8 Normalized numerical results for displacement at point A from Model 1

p	1	2	3	4	5	6
Model 1	0.188	0.625	0.937	0.969	0.985	0.993

For example, if the polynomial is of degree 2, according to Eq. (3.21), the local displacements for flat-top PU units u_{fp1} and u_{fp2} are expressed as:

$$\begin{cases}
 u_{x1} = d_{11} + d_{13}x + d_{15}y + d_{17}x^2 + d_{19}xy + d_{1(11)}y^2 \\
 u_{y1} = d_{12} + d_{14}x + d_{16}y + d_{18}x^2 + d_{1(10)}xy + d_{1(12)}y^2 \\
 u_{x2} = d_{21} + d_{23}x + d_{25}y + d_{27}x^2 + d_{29}xy + d_{2(11)}y^2 \\
 u_{y2} = d_{22} + d_{24}x + d_{26}y + d_{28}x^2 + d_{2(10)}xy + d_{2(12)}y^2
 \end{cases} \quad (3.104)$$

where 24 unknowns, i.e. d_{ij} ($i = 1, 2; j = 1 \sim 12$), are generated in total. Since the shape function for flat-top elements is equal to one everywhere, according to Eq. (3.24), the global displacements (U_{xft1}, U_{yft1}) and (U_{xft2}, U_{yft2}) for elements e_{ft1} and e_{ft2} are approximated as:

$$\begin{cases} U_{xft1} = 1 \cdot u_{x1} = d_{11} + d_{13}x + d_{15}y + d_{17}x^2 + d_{19}xy + d_{1(11)}y^2 \\ U_{yft1} = 1 \cdot u_{y1} = d_{12} + d_{14}x + d_{16}y + d_{18}x^2 + d_{1(10)}xy + d_{1(12)}y^2 \\ U_{xft2} = 1 \cdot u_{x2} = d_{21} + d_{23}x + d_{25}y + d_{27}x^2 + d_{29}xy + d_{2(11)}y^2 \\ U_{yft2} = 1 \cdot u_{y2} = d_{22} + d_{24}x + d_{26}y + d_{28}x^2 + d_{2(10)}xy + d_{2(12)}y^2 \end{cases} \quad (3.105)$$

while according to the derivation of shape function for PU elements in Eq. (3.6), the global displacement (U_{xpu1}, U_{ypu1}) for element e_{pu1} is approximated as:

$$\begin{cases} U_{xpu1} = w_{pu11} \cdot u_{x1} + w_{pu12}u_{x2} + w_{pu13}u_{x2} + w_{pu14}u_{x1} \\ \quad = (w_{pu11} + w_{pu14}) \cdot u_{x1} + (w_{pu12} + w_{pu13})u_{x2} \\ \quad = (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{11} + d_{13}x + d_{15}y + d_{17}x^2 + d_{19}xy + d_{1(11)}y^2) + \\ \quad \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{21} + d_{23}x + d_{25}y + d_{27}x^2 + d_{29}xy + d_{2(11)}y^2) \\ U_{ypu1} = w_{pu11} \cdot u_{y1} + w_{pu12}u_{y2} + w_{pu13}u_{y2} + w_{pu14}u_{y1} \\ \quad = (w_{pu11} + w_{pu14}) \cdot u_{y1} + (w_{pu12} + w_{pu13})u_{y2} \\ \quad = (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{12} + d_{14}x + d_{16}y + d_{18}x^2 + d_{1(10)}xy + d_{1(12)}y^2) + \\ \quad \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{22} + d_{24}x + d_{26}y + d_{28}x^2 + d_{2(10)}xy + d_{2(12)}y^2) \end{cases} \quad (3.106)$$

where the coefficients c_i ($i = 1 \sim 8$) are determined by the vertices' coordinates of the PU element e_{pu1} , as illustrated in Eq. (3.6). According to the following equations:

$$\begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{pmatrix} = \begin{pmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{pmatrix}^T \begin{pmatrix} U_x \\ U_y \end{pmatrix} \quad (3.107)$$

the strain vectors for elements e_{ft1} , e_{ft2} and e_{pu1} can be derived as Eqs. (3.108), (3.109) and (3.110), respectively.

$$\begin{cases} \varepsilon_{xft1} = d_{13} + 2d_{17}x + d_{19}y \\ \varepsilon_{yft1} = d_{16} + d_{1(10)}x + 2d_{1(12)}y \\ \varepsilon_{xyft1} = d_{14} + d_{15} + 2d_{18}x + d_{19}x + d_{1(10)}y + 2d_{1(11)}y \end{cases} \quad (3.108)$$

$$\begin{cases} \varepsilon_{xft2} = d_{23} + 2d_{27}x + d_{29}y \\ \varepsilon_{yft2} = d_{26} + d_{2(10)}x + 2d_{2(12)}y \\ \varepsilon_{xyft2} = d_{24} + d_{25} + 2d_{28}x + d_{29}x + d_{2(10)}y + 2d_{2(11)}y \end{cases} \quad (3.109)$$

$$\begin{cases} \varepsilon_{xft2} = (c_2 + c_4y) \cdot (d_{11} + d_{13}x + d_{15}y + d_{17}x^2 + d_{19}xy + d_{1(11)}y^2) + \\ \quad (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{13} + 2d_{17}x + d_{19}y) + \\ \quad (c_6 + c_8y) \cdot (d_{21} + d_{23}x + d_{25}y + d_{27}x^2 + d_{29}xy + d_{2(11)}y^2) + \\ \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{23} + 2d_{27}x + d_{29}y) \\ \varepsilon_{yft2} = (c_3 + c_4x) \cdot (d_{12} + d_{14}x + d_{16}y + d_{18}x^2 + d_{1(10)}xy + d_{1(12)}y^2) + \\ \quad (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{16} + d_{1(10)}x + 2d_{1(12)}y) + \\ \quad (c_7 + c_8x) \cdot (d_{22} + d_{24}x + d_{26}y + d_{28}x^2 + d_{2(10)}xy + d_{2(12)}y^2) + \\ \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{26} + d_{2(10)}x + 2d_{2(12)}y) \\ \varepsilon_{xyft2} = (c_3 + c_4x) \cdot (d_{11} + d_{13}x + d_{15}y + d_{17}x^2 + d_{19}xy + d_{1(11)}y^2) + \\ \quad (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{15} + d_{19}x + 2d_{1(11)}y) + \\ \quad (c_7 + c_8x) \cdot (d_{21} + d_{23}x + d_{25}y + d_{27}x^2 + d_{29}xy + d_{2(11)}y^2) + \\ \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{25} + d_{29}x + 2d_{2(11)}y) + \\ \quad (c_2 + c_4y) \cdot (d_{12} + d_{14}x + d_{16}y + d_{18}x^2 + d_{1(10)}xy + d_{1(12)}y^2) + \\ \quad (c_1 + c_2x + c_3y + c_4xy) \cdot (d_{14} + 2d_{18}x + d_{1(10)}y) + \\ \quad (c_6 + c_8y) \cdot (d_{22} + d_{24}x + d_{26}y + d_{28}x^2 + d_{2(10)}xy + d_{2(12)}y^2) + \\ \quad (c_5 + c_6x + c_7y + c_8xy) \cdot (d_{24} + 2d_{28}x + d_{2(10)}y) \end{cases} \quad (3.110)$$

Then, the stress vector for each element can be obtained according to the following constitutive equations:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{pmatrix} = \frac{E}{1-\nu^2} \begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{pmatrix} \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{pmatrix} \quad (3.111)$$

With the displacement, strain and stress functions, the sub-matrices can be derived according to the formula in Section 3.3.3 for each element, which are listed in Table 3.9. Thus, the global equilibrium equations are assembled as:

$$\begin{pmatrix} [K_{11}] & [K_{12}] \\ [K_{21}] & [K_{22}] \end{pmatrix} \begin{pmatrix} [D_1] \\ [D_2] \end{pmatrix} = \begin{pmatrix} [F_1] \\ [F_2] \end{pmatrix} \quad (3.112)$$

where $[K_{ij}]$ is the 12×12 sub-matrix generated by considering the influence of flat-top PU unit u_{fpi} on unit u_{fpi} ; $[D_i]$ is the 12×1 unknowns for flat-top PU unit u_{fpi} ; $[F_i]$ is the 12×1 force matrix for unit u_{fpi} . After solving Eq. (3.112), the displacement, strain and stress can be obtained by substituting the values of unknowns into Eqs. (3.105) and (3.106), Eqs. (3.108), (3.109) and (3.110) and Eq. (3.111), respectively.

Table 3.9 Sub-matrices generated by each element in Model 2

Element	e_{fi1}	e_{pu1}	e_{fi2}
Sub-matrices	$[K_{11}]$	$[K_{11}], [K_{12}], [K_{21}], [K_{22}]$	$[K_{22}]$
generated	$[F_1]$	$[F_1], [F_2]$	$[F_2]$

In this example, in order to study the influence of relative length between flat-top elements and PU elements, the length for PU element l_{pu} is designed as $24mm$, $9.6mm$, $4.8mm$, $0.96mm$ and $0.48mm$, respectively; while the lengths for two flat-top elements are equal. The normalized numerical results for five series of three-element model are listed in Table 3.10. It is observed that the accuracy deteriorates with the increase of element size ratio (maximum element/minimum element), but the difference becomes ignorable when the degree p takes 3 and above. Therefore, it is concluded that the flat-top PU based high-order approximation is not sensitive to the relative length of flat-top elements and PU elements, and the construction of flat-top PU mesh is quite flexible. Further comparison between the numerical results from the one- and three-element models shows that increasing the number of flat-top PU units can improve the approximation accuracy.

Then, the convergence of stress is studied with Models 1 and 2, and the length of PU element in Model 2 is equal to $24mm$. The minimum principle stress $\sigma_{\min}$ at

Point B are compared with the reference solution $-0.2010mm$ (Tian and Yagawa, 2005). The numerical error is listed in Table 3.11, where stress convergence is observed when increasing the degree p and number of flat-top PU units.

Table 3.10 Normalized numerical results for displacement at point A from Model 2

$l_{pu} \backslash p$	1	2	3	4	5	6
24mm	0.342	0.788	0.964	0.987	0.996	1.000
9.6mm	0.272	0.726	0.962	0.983	0.995	0.999
4.8mm	0.243	0.698	0.962	0.982	0.994	0.999
0.96mm	0.211	0.654	0.962	0.981	0.994	0.998
0.48mm	0.206	0.644	0.962	0.981	0.994	0.998

Table 3.11 Error of minimum principle stress $\sigma_{\min}$ at point B (unit: %)

p	1	2	3	4	5	6
Model 1	84.6	44.6	25.4	29.2	23.7	25.4
Model 2 ($l_{pu} = 24mm$)	30.0	20.0	6.0	3.8	3.9	0.8

3.5.7 Thin plate

A thin plate clamped at its four edges and subjected to a point loading at its center is simulated. As shown in Figure 3.24, the dimensions for this plate are $1m \times 1m \times 0.02m$, and the point loading $P = 1kN$ is pointing downward. The Young's modulus E and Poisson's ratio ν are $10GPa$ and 0.3 , respectively. The coordinate system is defined as follows: the origin is located at the center of the plate, the y -axis is parallel to one edge of the plate, and the z -axis is pointing downward. According to the mechanics of elasticity, the maximum displacement occurs at the center of the plate with the magnitude as (Liu, 2013):

$$U_{z,\max} = \alpha \frac{Pa^2}{D} \quad (3.113)$$

where $a = 1m$ is the side length of the plate, $\alpha = 0.00560$ for a square, D is the bending stiffness determined as:

$$D = \frac{Eh^3}{12(1-\nu^2)} \quad (3.114)$$

where $h = 0.02m$ is the height of the plate.

Figure 3.24 A thin plate clamped at its four edges and subjected to a point loading

The plate is represented by one flat-top PU unit with one flat-top element. The degree p for local approximation polynomial ranges from 1 to 6. The vertical displacement U_z of the plate along a central line ($-0.5 \leq x \leq 0.5$, $y = 0$, $z = 0$) is numerically obtained and plotted in Figure 3.25 (a), which is obviously convergent with the increase of degree p . The vertical displacement contour for $p = 6$ is presented in Figure 3.25 (b), which shows a reasonable accuracy. In Table 3.12, The maximum displacement $U_{z\max}$ at the center of the plate is numerically obtained and compared with the analytical solution provided by Eq. (3.113). The numerical results are satisfactory with the errors for the last two cases ($p = 5$ and 6) as 2.6% and 0.9%, respectively. It is worth to note that the deformation of thin plate is non-polynomial, but a series of trigonometric and hyperbolic functions. The promising numerical results imply that the flat-top PU based high-order approximation can also approximate non-polynomial target functions well.

(a) Vertical displacement U_z along a central line of the plate (b) Vertical displacement contour for $p = 6$ (unit: mm)

Figure 3.25 Vertical displacement along a central line and contour for whole plate

Table 3.12 Maximum displacement ($U_{z\max}$) at the center of the plate

p	Numerical results						Analytical solution (mm)
	1	2	3	4	5	6	
$U_{z\max}$ (mm)	0.000	0.020	0.020	0.716	0.744	0.757	0.764
Error (%)	100	97.4	97.4	6.3	2.6	0.9	

3.5.8 Table-like structure

A table-like structure comprised of one thin plate and four columns is analyzed. Its geometry shape is depicted in Figure 3.26 (a). The uniformly distributed loading $q = 250kPa$ is applied at the center of the plate on an area of $0.2m \times 0.2m$, and the loading is pointing downward. The four columns are fixed at the bottom. The Young's modulus E and Poisson's ratio ν are $1GPa$ and 0.25 , respectively. The coordinate system is defined as follows: the x and y axes lie in the surface where the bottom of the four columns is located, the z -axis is pointing downward and the origin is the projection point of the plate center on the bottom surface.

Since analytical solution is not available for this problem, convergence test using the FEM is conducted. The convergent vertical displacement U_z at the center of thin plate is used as reference solution to calibrate the results from flat-top PU based high-order approximation. The reference FEM model employs 627 10-node (second order) tetrahedral elements, 1554 nodes and correspondingly $1554 \times 3 = 4662$ DOFs.

In flat-top PU based high-order approximation, the side-view discretization of table-like structure is presented in Figure 3.26 (b), where different lengths for flat-top elements and PU elements are used. The corresponding numerical model is illustrated in Figure 3.26 (c), where five flat-top PU units are generated, resulting in five flat-top elements and four PU elements. The degree p for local approximation

(a) Geometrical and loading information for the table-like structure

(b) Discretization of table-like structure with flat-top PU mesh (side view)

(c) Numerical model for flat-top PU based high-order approximation

Figure 3.26 A table-like structure and its numerical models

Table 3.13 Displacement U_z at the center of the thin plate

Flat-top PU based high-order approximation							FEM (mm)
p	1	2	3	4	5	6	
U_z (mm)	0.02	0.32	0.72	2.18	2.25	2.65	2.65
Error (%)	99.2	87.9	72.8	17.7	15.1	0	

polynomial ranges from 1 to 6. The displacement U_z of the plate at its center point is obtained from flat-top PU based high-order approximation and compared with the

reference solution in Table 3.13. The numerical results become identical to the reference solution when the degree p is increased to 6. Since five flat-top PU units are used in flat-top PU based high-order approximation, the total DOFs for $p = 6$ is $84 \times 3 \times 5 = 1260$, which is only 27% of the FEM model to achieve the same approximation accuracy.

3.6 Conclusions

This chapter mainly introduces the construction of flat-top PU mesh based on the traditional PU mesh, the high-order approximation and establishment of global equilibrium equations. Several numerical examples are presented and analyzed to verify the accuracy and efficiency of flat-top PU based high-order approximation. The following conclusions can be obtained:

1. The construction of flat-top PU mesh based on traditional PU mesh is easy and straightforward. Since it is not necessary for the flat-top PU mesh to comply with the geometry shape of boundary, it is quite flexible to construct the mesh, which is large enough to cover the whole problem domain.
2. Convergence with the increase of degree p for local approximation polynomial is evidently observed in all the numerical examples.
3. The accuracy of flat-top PU based high-order approximation can be improved with the increase of degree p and number of flat-top PU units.
4. The flat-top PU based high-order approximation is capable of simulating various structures, such as problem with large aspect ratio, curved and concave structure and problem with non-polynomial solutions.
5. The influence of relative length between flat-top elements and PU elements is ignorable, which ensures the numerical stability of flat-top PU based high-order approximation.
6. Compared with FEM, flat-top PU based high-order approximation is able to obtain comparable accuracy with much less DOFs, which can increase the efficiency of simulation considerably.

CHAPTER 4 LOCAL REFINEMENT WITH FLAT-TOP PU MESH

4.1 Introduction

In traditional PU mesh, the linear dependence problem has been widely recognized for high-order approximation. The global equilibrium equations in PU based high-order approximation is underdetermined and special equation solver is required, which is not as efficient as the common solvers (Riker and Holzer, 2009). The flat-top PU mesh has been proven effective in reducing the matrix condition number with high-order polynomial enrichment (Hong and Lee, 2013b) and it is automatically L^2 -stable (Schweitzer, 2012). The property of linear independence for regularly patterned flat-top PU mesh has been theoretically proven by An et al. (2011, 2012).

However, for stress concentration and stress singularity problems, such as the infinite plate with a hole and the L-shaped structure, local refinement for flat-top PU mesh is necessary. Therefore, in this chapter, the locally refined flat-top PU mesh in one- and two-dimensional cases is constructed and analyzed, respectively. In Section 4.2, a brief review of the equivalent p -version element and rank deficiency counting approach is presented. Then, the construction of locally refined flat-top PU mesh and the proof of its linear independence with high-order approximation are introduced in Sections 4.3 and 4.4 for one- and two-dimensional cases, respectively, where the locally refined flat-top PU mesh is constructed based on the regularly patterned mesh and the property of linear independence is proven from element level based on the basic theory in Section 4.2. In Section 4.5, two numerical examples modelling stress concentration and stress singularity are analyzed to verify the accuracy and efficiency of locally refined flat-top PU mesh. Finally, conclusions for this chapter are addressed in Section 4.6.

4.2 Basic Theory for proof of linear independence

Under the condition of high-order approximation, the properties of linear dependence for PU mesh and linear independence for regularly patterned flat-top PU mesh have been theoretically proven by An et al. (2011, 2012, 2014) and He et al. (2015) with the concepts of equivalent p -version element and rank deficiency counting approach. In this chapter, the proof of linear independence for locally

refined flat-top PU mesh is also addressed based on the equivalent p -version element and rank deficiency counting approach. Therefore, it is necessary to review these basic theories before carrying out the independence analysis.

4.2.1 Equivalent p -version element

Equivalent p -version element refers to the traditional p -version finite element, which can generate exactly the same approximation as the element in flat-top PU mesh. In the equivalent p -version element, besides the end nodes at vertices, additional internal nodes are distributed along the sides and over the surface. The total number of nodes for an equivalent p -version element is determined by the approximation space generated by the shape function and local approximation polynomial of the corresponding element in flat-top PU mesh.

(a) White-colored PU element e_i in a regularly patterned flat-top PU mesh

(b) Distribution of nodes in equivalent p -version element corresponding to a white-colored PU element

Figure 4.1 White-colored PU element and its equivalent p -version element

Take the white-colored PU element e_i in Figure 4.1 (a) as an example. In this regularly patterned flat-top PU mesh, the sizes for PU elements are smaller than the flat-top elements, which does not influence the accuracy of numerical results as studied in Section 3.5. For element e_i , the local approximation polynomial is of degree p and, according to Eq. (3.6), the space for its shape function is $\{1, x, y, xy\}$. Therefore, after multiplying the local approximation polynomial with the shape function, the approximation space for element e_i is the space for the complete polynomial of degree $(p + 2)$ minus the terms x^{p+2} and y^{p+2} . Correspondingly, the number of nodes for the equivalent p -version element is calculated as:

$$\text{No. of nodes} = \frac{(p+3)(p+4)}{2} - 2 \quad (4.1)$$

As a result, for the equivalent p -version element corresponding to element e_i , as illustrated in Figure 4.1 (b), besides the four nodes at vertices, there are p internal nodes within each side; and another $p(p - 1) / 2$ internal nodes are distributed within the element. The number of nodes for the equivalent p -version element corresponding to every other element within the flat-top PU mesh can be calculated in the same way.

In flat-top PU mesh, the number of independent DOFs introduced by each element can be represented as the number of nodes in the corresponding equivalent p -version element. Actually, for two-dimensional case, the number of independent DOFs for each element is equal to the number of nodes in corresponding equivalent p -version element multiplied by 2 (for x and y directions, respectively). However, when calculating the rank deficiency for every element, which will be introduced later in this section, the number of DOFs introduced by each unit should be also multiplied by 2 (for x and y directions, respectively). Therefore, for easy understanding, either the independent DOFs generated by the nodes in equivalent p -version element or the DOFs introduced by the local approximation polynomial for each unit will not be multiplied by 2, simultaneously.

4.2.2 Rank deficiency counting approach

In flat-top PU based high-order approximation, complete high-order polynomial with degree p is employed as local approximation for every flat-top PU unit, and the

coefficients of the monomials are recognized as unknowns. Therefore, in one- and two-dimensional cases, respectively, the number of DOFs introduced into the system by each unit can be calculated as:

$$\text{No. of DOFs} = \begin{cases} p+1 & (1\text{D}) \\ \frac{(p+1)(p+2)}{2} & (2\text{D}) \end{cases} \quad (4.2)$$

For each element, the rank deficiency equals to the number of newly introduced DOFs subtracting the number of newly added nodes of the equivalent p -version element, i.e. the number of independent DOFs for corresponding element in flat-top PU mesh. The calculation procedure for each element will be introduced in detail in Sections 4.3 and 4.4. Moreover, in order to ensure that the rank deficiency is a non-negative integer, one more principle should be explained here, i.e. the principle of increase of added rank deficiency (An et al., 2011, 2012). This principle states that the rank deficiency RD_{ei} for every element e_i , whether it is flat-top element or PU element, should be non-negative. According to Eqs. (4.1) and (4.2), the principle can be expressed as:

$$RD_{e_i} = \begin{cases} 0 & ((\text{No. of DOFs} - \text{No. of nodes}) \leq 0) \\ \text{No. of DOFs} - \text{No. of nodes} & ((\text{No. of DOFs} - \text{No. of nodes}) > 0) \end{cases} \quad (4.3)$$

Then, the rank deficiency for the whole flat-top PU mesh, i.e. RD , is calculated as the sum of the rank deficiency for every element, that is:

$$RD = \sum_i RD_{e_i} \quad (4.4)$$

As shown by Eq. (4.3), for linearly independent case in two-dimensional space, no matter whether the number of nodes in Eq. (4.1) and the number of DOFs in Eq. (4.2) are simultaneously multiplied by 2 or not, the rank deficiency is equal to zero. As for the linearly dependent case in two-dimensional space, the rank deficiency can be obtained by multiplying the results of Eqs. (4.3) and (4.4) by 2, respectively.

4.3 Local refinement and proof of linear independence for 1D case

Practically, the domain, which is required to be locally refined, can be located anywhere within the flat-top PU mesh. Therefore, for one-dimensional case, based

on the introduction of different types of elements in a regularly patterned mesh (An et al., 2014), three basic cases of local refinement are discussed, and all the other local refining situations can be treated as a simple combination of these three basic cases. According to Eq. (4.4), as long as the rank deficiencies for all these three basic cases are equal to zero, the rank deficiency for any combination of the three basic cases is equal to zero. Moreover, for the case that further local refinement within the refined elements is necessary, the way to count rank deficiencies for the further refined elements is exactly the same, namely the global stiffness matrix for further local refinement within the refined elements is also linearly independent. The conclusions about combination and further local refinement are also applicable for the situation in two-dimensional case in Section 4.4.

4.3.1 Element types in regularly patterned flat-top PU mesh

In a regularly patterned flat-top PU mesh, all the elements are numbered as e_i ($i = 1, 2, \dots, n$) in sequence, as illustrated in Figure 4.2. The subscript number indicates the sequence the elements assembled into the system. Correspondingly, the elements can be categorized into three different types, i.e. Types I, II and III, according to the number of flat-top PU units and the number of nodes newly introduced by assembling the elements (An et al., 2014). Specifically, the first element e_1 is denoted as Type I element, the even-numbered elements e_i ($i = 2, 4, 6, \dots$) are recognized as Type II element, and the odd-numbered elements but the first one, i.e. e_i ($i = 3, 5, 7, \dots$), are defined as Type III element.

Figure 4.2 Different element types in a regularly patterned flat-top PU mesh

The properties for three types of elements are listed in Table 4.1. Take the Type II element e_2 as an example. When it is assembled into the mesh, one new flat-top PU unit with element e_3 as elemental support is introduced into the system. According to Eq. (4.2), the number of DOFs newly added in global equilibrium equations is $(p + 1)$. Moreover, for element e_2 , the local approximation polynomial is of degree p ,

and the space for its shape function is $\{1, x\}$. Thus, after multiplying the local approximation polynomial with the shape function, the approximation space is the space for the complete polynomial of degree $(p + 1)$. So there are $(p + 2)$ nodes in the equivalent p -version element corresponding to element e_2 . As shown in Figure 4.3, besides two end nodes as vertices, there are p internal nodes distributed within the element. Since the red-colored end node is shared with elements e_1 , it is introduced into the system when element e_1 is assembled. Therefore, the nodes introduced by element e_2 are the green-colored ones, which include one end node and p internal nodes. According to Eq. (4.3), the rank deficiency for element e_2 , i.e. RD_{e_2} , can be calculated as:

$$RD_{e_2} = (p + 1) - (1 + p) = 0 \quad (4.5)$$

which is the same as listed in the row for Type II element in Table 4.1. The rank deficiencies for Types I and III are also calculated according to the data presented in Table 4.1, which are equal to zero as list in the last column. According to Eq. (4.4), the rank deficiency for the whole mesh is equal to zero.

Table 4.1 Properties for three different types of elements in one-dimensional case

Element type	No. of unit introduced	DOFs for each unit	No. of nodes introduced			Rank deficiency
			End	Internal	Total	
I	1		2	$p - 1$	$p + 1$	0
II	1	$p + 1$	1	p	$p + 1$	0
III	0		1	$p - 1$	p	0 ($-p < 0$)
Rank deficiency for the whole mesh						0

Figure 4.3 Nodes distribution in equivalent p -version element for element e_2

4.3.2 First basic local refinement case

The first basic case refers to the local refinement within Type I element, as illustrated in Figure 4.4. Actually, it is equal to replacing the original element e_1 with a new flat-top PU mesh with smaller-sized elements. The new elements are numbered as $e_{1,j}$ ($j = 1, 2, 3, \dots, n$) in sequence. Accordingly, the shape function for element e_1 is also replaced. As shown in Figure 4.4, the shape functions for new elements have the same properties as for the original elements in Figure 4.2, which are also derived according to Eqs. (3.2) and (3.3).

When implementing local refinement, the only requirement is the connection between original and new elements. In order to ensure the continuity, the shape functions for the first and last new elements, i.e. $e_{1,1}$ and $e_{1,n}$, should have the same properties as for the original element e_1 . Take the locally refined mesh in Figure 4.4 as an example. Since element e_1 is a flat-top element, the first and last new elements, i.e. $e_{1,1}$ and $e_{1,5}$, are also flat-top elements. For the other two cases, the requirement for implementing local refinement is the same as this first case.

In locally refined flat-top PU mesh, the elements are assembled according to the following principle: 1) they are constructed in the sequence as the original elements numbered; 2) for the locally refined elements, they are constructed in the sequence as the newly generated elements numbered. It is the same assemblage principle for the two-dimensional locally refined flat-top PU mesh in Section 4.4. Take the mesh in Figure 4.4 as an example. In the original mesh, the element e_1 should be firstly constructed. Therefore, the elements $e_{1,j}$ ($j = 1 \sim 5$) are firstly assembled in the sequence as indicated by the subscript j . Then, the elements e_i ($i = 2 \sim 5$) are assembled in the sequence indicated by the subscript i .

Figure 4.4 Local refinement within Type I element

In Figure 4.4, it is observed that no new element type is generated in the first basic local refinement case. Specifically, the first element $e_{1,1}$ is Type I element; the

elements $e_{1,j}$, where j is an even number, are Type II elements; and the elements $e_{1,j}$, where j is an odd number but one, belongs to Type III. Therefore, according to Table 4.1, the rank deficiency for first basic local refinement case is equal to zero, which means that the corresponding locally refined flat-top PU mesh is linearly independent.

4.3.3 Second basic local refinement case

The second basic case is the local refinement within Type II element, as illustrated in Figure 4.5, where the element e_4 is locally refined as an example. The new elements are numbered as $e_{i,j}$ ($j = 1, 2, \dots, n$) in sequence, where i is the number for original element and $i = 4$ in Figure 4.5. The shape functions for newly generated elements are also derived according to Eqs. (3.2) and (3.3). According to the requirement for implementing local refinement in one-dimensional case, the first and last new elements, $e_{4,1}$ and $e_{4,5}$, are PU elements, which are the same as the original element e_4 . According to the assemblage principle, the elements e_i ($i = 1 \sim 3$) are firstly constructed in the sequence as indicated by subscript i ; then, the elements $e_{4,j}$ ($j = 1 \sim 5$) are assembled in the sequence as indicated by subscript j ; at last, the element e_5 is added into the mesh.

In this second basic local refinement case, it can also be observed that no new element type is generated. The elements $e_{i,j}$, where j is an odd number, are Type II elements; and the elements $e_{i,j}$, where j is an even number, are Type III elements. Therefore, according to Table 4.1, the rank deficiency for second basic local refinement case is equal to zero, which means that the corresponding locally refined flat-top PU mesh is linearly independent.

Figure 4.5 Local refinement within Type II element

4.3.4 Third basic local refinement case

The last basic case refers to the local refinement within Type III element, as illustrated in Figure 4.6, where the element e_3 is locally refined as an example. Same as the other two cases, the new elements are numbered as e_{ij} ($j = 1, 2, \dots, n$) in sequence and $i = 3$ in Figure 4.6; The shape functions for newly generated elements are derived according to Eqs. (3.2) and (3.3). According to the requirement for implementing local refinement in one-dimensional case, the first and last new elements, $e_{3,1}$ and $e_{3,5}$, are flat-top elements, which are the same as the original element e_3 . According to the assemblage principle, the elements e_1 and e_2 are constructed in the sequence as indicated by subscript; then, the elements $e_{3,j}$ ($j = 1 \sim 5$) are assembled in the sequence as indicated by subscript j ; at last, the elements e_4 and e_5 are added into the mesh one by one.

In this case, it can still be observed that no new element type is generated. The elements e_{ij} , where j is an odd number, are Type III elements; and the elements e_{ij} , where j is an even number, are Type II elements. Therefore, according to Table 4.1, the rank deficiency for last basic case is equal to zero, which means that the corresponding locally refined flat-top PU mesh is linearly independent.

Figure 4.6 Local refinement within Type III element

As stated in the beginning of Section 4.3, since all the three basic cases have been proven linearly independent, the combination of them and the further local refinement within refined elements are also linearly independent.

4.4 Local refinement and proof of linear independence for 2D case

For two-dimensional case, based on the introduction of different types of elements in a regularly patterned mesh (An et al., 2014), three basic cases of local refinement within flat-top elements are discussed. The property of linear independence for the newly generated element types is explicitly proven from element level based on the

theory in Section 4.2. For the local refinement of PU elements, it can be realized through the combination of the three basic cases. Similar with the one-dimensional case, as long as the flat-top PU meshes in three basic cases are proven linearly independent, all the other local refining situations and the further local refinement within refined elements are also linearly independent.

4.4.1 Element types in regularly patterned flat-top PU mesh

In regularly patterned flat-top PU mesh, all the elements are numbered as e_i ($i = 1, 2, \dots, n$) in sequence, and the subscript i for each element is shown in Figure 4.7 (a), which indicates the sequence the elements assembled into the system. According to the number of flat-top PU units and the number of nodes newly introduced by assembling the elements, all the elements can be categorized into four different types, i.e. Types I to IV (An et al., 2014), as shown in Figure 4.7 (b). The properties for each element type are listed in Table 4.2, which presents the basic information for calculating rank deficiency for every element type. It is worth to note that the number of nodes newly introduced by assembling Type III element is not listed in Table 4.2. It is because that the Type III elements refer to all the elements that do not introduce any new flat-top PU unit into the mesh. Therefore, according to Eq. (4.3), no matter how many new nodes introduced by Type III elements, the rank deficiency is equal to zero. According to the results in Table 4.2, it can be concluded that the regularly patterned flat-top PU mesh is linearly independent (An et al., 2014).

1	2	3	4	5
6	7	8	9	10
11	12	13	14	15
16	17	18	19	20
21	22	23	24	25

(a) Assemblage sequence of elements

(b) Four different element types

Figure 4.7 Assemblage sequence and element types in a regularly patterned flat-top PU mesh

4.4.2 First basic local refinement case

The first basic case refers to the local refinement for Type I element, i.e. flat-top element e_1 , as shown in Figure 4.8 (a). It is observed that, not only the element e_1 , but also its adjacent blue-colored PU elements, e_2 and e_6 , are divided into smaller-sized elements. When implementing local refinement in two-dimensional flat-top PU mesh, in order to ensure the global continuity, it is required that: 1) the numbers of rows and columns for the new elements replacing the original flat-top element should be odd, with the side rows and columns distributed with new flat-top elements and blue-colored PU elements; 2) all the blue-colored PU elements adjacent to locally refined flat-top element are discretized into new PU elements.

The new elements in Figure 4.8 (a) are numbered as $e_{i,j}$ ($j = 1, 2, \dots, n$) in sequence, and the subscript indicates the j th new element generated by discretizing the i th original element. It is observed that a new type of PU elements, i.e. the orange-colored ones, is generated by dividing the blue-colored PU elements. The shape functions for these orange-colored elements are also derived according to Eq. (3.6). Since every orange-colored element connects three elemental supports, its four shape functions pertain to three different flat-top PU units, which are assigned in the same way as for the blue- and white-colored PU elements in Section 3.2.2. Take the

Table 4.2 Properties for different types of elements in two-dimensional case

Element type	No. of unit introduced	No. of nodes introduced in equivalent p -version element									Rank deficiency
		Vertex	Type 1 side	Type 2 side	Type 3 side	Type 1 element	Type 2 element	Type 3 element	Type 4 element	Type 5 element	
I	1	4	4	0	0	1	0	0	0	0	0
II	1	2	1	2	0	0	1	0	0	0	0
III	0	-	-	-	-	-	-	-	-	-	< 0
IV	1	1	0	2	0	0	0	1	0	0	0
V	1	2	0	2	1	0	0	0	1	0	0
VI	1	2	1	2	0	0	0	0	1	0	< 0
VII	1	1	0	2	0	0	0	0	0	1	< 0
Rank deficiency for the whole mesh											0

Note: Type 1 side: each has $(p - 1)$ internal nodes

Type 2 side: each has p internal nodes

Type 3 side: each has $[(p - 1) - (n - 1)] / n$ internal nodes

Type 1 element: each has $(p + 1)(p + 2) / 2 - 4 - 4(p - 1)$ internal nodes

Type 2 element: each has $[(p + 2)(p + 3) / 2 - 1] - 4 - 2(p - 1) - 2p$ internal nodes

Type 3 element: each has $[(p + 3)(p + 4) / 2 - 2] - 4 - 4p$ internal nodes

Type 4 element: each has $[(p + 2)(p + 3) / 2 - 1] - 4 - (p - 1) - 2p - [(p - 1) - (n - 1)] / n$ internal nodes

Type 5 element: each has $[(p + 3)(p + 4) / 2 - 3] - 4 - 3p - [(p - 1) - (n - 1)] / n$ internal nodes

element $e_{2,2}$ in Figure 4.8 (a) as an example. Assuming its four vertices are numbered from one to four in counter-clockwise direction as in Figure 3.3 (a), the shape functions $w_{(2,2)1}$ and $w_{(2,2)4}$ belong to the flat-top PU units with elements $e_{1,9}$ and $e_{1,3}$ as elemental supports, respectively; while $w_{(2,2)2}$ and $w_{(2,2)3}$ pertain to the unit with element e_3 as elemental support.

The elements in two-dimensional locally refined flat-top PU mesh are also assembled according to the principle stated in Section 4.3.2. Specifically, for this first basic local refinement case, since element e_1 should be firstly constructed in the original mesh, the elements $e_{1,j}$ are firstly assembled in the sequence as indicated by subscript j . Then, the elements e_i ($i \geq 2$) are added into the mesh in the sequence indicated by subscript i . Based on the number of flat-top PU units and the number of nodes newly introduced by assembling the elements, all the elements in this first basic case can be categorized into five types, as shown in Figure 4.8 (b). Compared with the element types in regularly patterned flat-top PU mesh, a new type, Type V, is generated in this case.

Before the rank deficiency calculation of Type V elements, the nodes distribution in equivalent p -version elements corresponding to new elements generated by discretizing original blue-colored PU elements should be analyzed. Here the elements $e_{2,j}$ ($j = 1, 2, \dots, n$) in Figure 4.8 (a) are taken as an example. For the original PU element e_2 , according to Figure 4.7 and Table 4.2, there are totally $(p - 1)$ internal nodes within the right side, which is shared with the flat-top element e_3 . It is assumed that the original PU element e_2 is discretized into n smaller-sized PU elements. According to the requirement of implementing local refinement in two-dimensional flat-top PU mesh, n is an odd number greater than one, namely $n \geq 3$. Therefore, theoretically, for each new element $e_{2,j}$, after subtracting the end nodes as vertices, there are $[(p - 1) - (n - 1)] / n$ internal nodes within the right side which is shared with flat-top element e_3 . This new kind of sides are defined as Type 3 side as described in the note below Table 4.2. Therefore, the nodes distribution of equivalent p -version element for Type V element $e_{2,1}$ can be described as in Figure 4.8 (c). Besides four vertices, the equivalent p -version element has one side with $(p - 1)$ internal nodes, two sides each with p internal nodes and another side with $[(p -$

1) - (n - 1)] / n internal nodes. Since the local approximation polynomial for element $e_{2,1}$ is of degree p and the space for its shape function is $\{1, x\}$, the approximation space for Type V element $e_{2,1}$ is equal to the space for complete polynomial with degree $(p + 1)$ minus the term y^{p+1} . Therefore, theoretically, the number of internal nodes within a Type V element is calculated as:

$$\left[\frac{(p+2)(p+3)}{2} - 1 \right] - 4 - (p-1) - 2p - \frac{(p-1) - (n-1)}{n} = \frac{p(p-1)}{2} - \frac{p}{n} \quad (4.6)$$

and this element, in which the vertices and sides are not included, is denoted as Type 4 element as described in the note below Table 4.2.

As demonstrated in Figure 4.8 (c), the red-colored parts, which include two end nodes and one side with $p - 1$ internal nodes, have been introduced by element $e_{1,3}$ already, before element $e_{2,1}$ is constructed. While the green-colored parts, i.e. two end nodes, three sides and one element, are introduced by assembling element $e_{2,1}$. Hence, the number of nodes newly introduced by a Type V element is equal to:

$$2 + 2p + \frac{(p-1) - (n-1)}{n} + \left[\frac{p(p-1)}{2} - \frac{p}{n} \right] = \left[\frac{(p+2)(p+3)}{2} - 1 \right] - 2 - (p-1) \quad (4.7)$$

1,1	1,2	1,3	2,1			
1,4	1,5	1,6	2,2	3	4	5
1,7	1,8	1,9	2,3			
6,1	6,2	6,3	7	8	9	10
11		12		13	14	15
16		17		18	19	20
21		22		23	24	25

(a) Assemblage sequence of elements in first basic local refinement case

(b) Five different element types

(c) Distribution of nodes in equivalent p -version element for element $e_{2,1}$

Figure 4.8 First basic local refinement case

Obviously, each Type V element introduces one new flat-top PU unit into the mesh. Therefore, according to Eq. (4.2), the rank deficiency for a Type V element, RD_V , can be calculated as:

$$RD_V = \frac{(p+1)(p+2)}{2} - \left\{ \left\lceil \frac{(p+2)(p+3)}{2} \right\rceil - 1 - 2 - (p-1) \right\} = 0 \quad (4.8)$$

The properties for Type V elements are also listed in Table 4.2. For the rest elements $e_{2,j}$ ($j \geq 2$) generated by discretizing the original blue-colored PU element

e_2 , all of them are Type III elements. Moreover, it is the same situation with elements $e_{6,j}$ ($j = 1, 2, \dots, n$), which are generated by dividing the original PU element e_6 . From Eq. (4.8), it is observed that, for Type V elements, the parameter n has no influence on the rank deficiency.

Since the rank deficiency for every element type in this first basic local refinement case is equal to zero, according to Eq. (4.4), the rank deficiency for the whole mesh is also equal to zero. Therefore, it is linearly independent for this first basic case.

4.4.3 Second basic local refinement case

The second basic case is the local refinement for a flat-top element, which is located in the first row (upper row) or first column (left column) of the regularly patterned flat-top PU mesh, except the element e_1 . Here the element e_3 in Figure 4.7 (a) is locally refined as an example. As shown in Figure 4.9 (a), the local refinement is implemented according to the requirement stated in Section 4.4.2, and the new elements are numbered and assembled in the same way as for the first basic case. Correspondingly, all the elements in this second basic case can be classified into seven types in total, as presented in Figure 4.9 (b). It is observed that only two new element types, i.e. Type VI and Type VII, are generated, of which the rank deficiency will be calculated in detail.

1	2,1	3,1	3,2	3,3	4,1	5
	2,2	3,4	3,5	3,6	4,2	
	2,3	3,7	3,8	3,9	4,3	
6	7	8,1	8,2	8,3	9	10
11	12	13			14	15
16	17	18			19	20
21	22	23			24	25

(a) Assemblage sequence of elements in second basic local refinement case

(b) Seven different element types

Figure 4.9 Flat-top PU mesh for second basic local refinement case

For Type VI element, the element $e_{2,1}$ in Figure 4.9 (a) is taken as an example. It has the exactly same approximation space as Type V element $e_{2,1}$ in Figure 4.8 (a). So there are totally $(p + 2)(p + 3) / 2 - 1$ nodes in the corresponding equivalent p -version element. Moreover, this equivalent p -version element has the exactly same vertices, sides and element with Type V elements. As illustrated in Figure 4.10 (a), there are four vertices, one side with $(p - 1)$ internal nodes, two sides each with p internal nodes, one another side with $[(p - 1) - (n - 1)] / n$ internal nodes and one element with the same number of internal nodes as expressed in Eq. (4.6). However, in this case, the red-colored parts, which are introduced before element $e_{2,1}$ is assembled, are different from Type V element. As shown in Figure 4.10 (a), the red-colored parts include two vertices and one side with $[(p - 1) - (n - 1)] / n$ internal nodes. Therefore, the number of new nodes introduced by a Type VI element is:

$$\left[\frac{(p+2)(p+3)}{2} - 1 \right] - 2 - \frac{(p-1)-(n-1)}{n} = 2p + (p-1) + \left[\frac{p(p-1)}{2} - \frac{p}{n} \right] \quad (4.9)$$

which are marked as the green-colored parts in Figure 4.10 (a). Since each Type VI element introduces a new flat-top PU unit into the mesh, its rank deficiency RD_{VI} is calculated as:

$$RD_{VI} = \frac{(p+1)(p+2)}{2} - \left\{ 2p + (p-1) + \left[\frac{p(p-1)}{2} - \frac{p}{n} \right] \right\} = p \left(\frac{1}{n} - 1 \right) < 0 \quad (4.10)$$

According to the principle of increase of added rank deficiency in Section 4.2.2:

$$RD_{VI} = 0 \quad (4.11)$$

For Type VII element, the element $e_{2,2}$ in Figure 4.9 (a) is taken as an example, which is an orange-colored PU element. Since its four shape functions pertain to three flat-top PU units, as stated in Section 4.4.2, the space for the shape functions is $\{1, x, xy\}$. Therefore, after multiplying each shape function with corresponding local approximation polynomial of degree p , the approximation space for Type VII element $e_{2,2}$ is equal to the space of a complete polynomial with degree $p + 2$ minus the three terms: y^{p+1} , x^{p+2} and y^{p+2} . Hence, the number of nodes in the corresponding equivalent p -version element is equal to $(p + 3)(p + 4) / 2 - 3$. Since this equivalent p -version element has four vertices, three sides each with p internal nodes and one another side with $[(p - 1) - (n - 1)] / n$ internal nodes, as shown in Figure 4.10 (b), the number of nodes distributed within this element is equal to:

$$\left[\frac{(p+3)(p+4)}{2} - 3 \right] - 4 - 3p - \frac{(p-1)-(n-1)}{n} = \frac{p(p+1)}{2} - \frac{p}{n} \quad (4.12)$$

and this element, in which the vertices and sides are not included, is defined as Type 5 element as described in the note below Table 4.2.

As demonstrated in Figure 4.10 (b), the red-colored parts, which include three end nodes, one side with p internal nodes and another side with $[(p - 1) - (n - 1)] / n$ internal nodes, have been introduced by elements e_1 and $e_{2,1}$ already, before element $e_{2,2}$ is constructed. While the green-colored parts, i.e. one end node, two sides and one element, are introduced by assembling element $e_{2,2}$. Hence, the number of nodes newly introduced by a Type VII element is equal to:

$$\left[\frac{(p+3)(p+4)}{2} - 3 \right] - 3 - p - \frac{(p-1)-(n-1)}{n} = 1 + 2p + \left[\frac{p(p+1)}{2} - \frac{p}{n} \right] \quad (4.13)$$

Obviously, each Type VII element introduces one new flat-top PU unit into the mesh. Therefore, the rank deficiency for a Type VII element, RD_{VII} , can be as:

$$RD_{VII} = \frac{(p+1)(p+2)}{2} - \left\{ 1 + 2p + \left[\frac{p(p+1)}{2} - \frac{p}{n} \right] \right\} = p \left(\frac{1}{n} - 1 \right) < 0 \quad (4.14)$$

According to the principle of increase of added rank deficiency in Section 4.2.2:

$$RD_{VII} = 0 \quad (4.15)$$

(c) Distribution of nodes in equivalent p -version element for element $e_{2,1}$

(d) Distribution of nodes in equivalent p -version element for element $e_{2,2}$

Figure 4.10 Nodes distribution for two different Types of elements

The properties for Types VI and VII elements are listed in Table 4.2. Moreover, from Eqs. (4.10) and (4.14), it is observed that, although the formula for rank deficiencies of Types VI and VII elements contains the parameter n , since n is an odd integer and $n \geq 3$, it has no influence on the rank deficiencies.

According to the analysis in this section and Eq. (4.4), it can be concluded that the rank deficiency for the locally refined flat-top PU mesh in this second basic case is equal to zero. Therefore, the global equilibrium equations are linearly independent in this case.

4.4.4 Third basic local refinement case

The last basic case refers to the local refinement for a flat-top element, which is not located in the first row (upper row) or first column (left column) of the regularly patterned flat-top PU mesh. Here the element e_{13} in Figure 4.7 (a) is locally refined as an example. As shown in Figure 4.11 (a), the local refinement is implemented according to the requirement stated in Section 4.4.2, and the new elements are numbered and assembled in the same way as for the first basic case. Correspondingly, all the elements in this last basic case can be classified into five types in total, as presented in Figure 4.11 (b). It is observed that no more new element type is generated in this case. Therefore, it is easy to make the conclusion that the rank deficiency for the locally refined flat-top PU mesh in last basic case is equal to zero and the corresponding global equilibrium equations are linearly independent.

1	2	3			4	5
6	7	8,1	8,2	8,3	9	10
11	12,1	13,1	13,2	13,3	14,1	15
	12,2	13,4	13,5	13,6	14,2	
	12,3	13,7	13,8	13,9	14,3	
16	17	18,1	18,2	18,3	19	20
21	22	23			24	25

(a) Assemblage sequence of elements in second basic local refinement case

(b) Five different element types

Figure 4.11 Flat-top PU mesh for third basic local refinement case

Same with the one-dimensional case, since all the three basic cases have been proven linearly independent, the combination of them and the further local refinement within refined elements are also linearly independent.

4.4.5 Local refinement for PU elements

From Section 4.3.3, it can be observed that it is easy and straightforward to implement local refinement within the PU elements in one-dimensional case. However, for two-dimensional case, it is not cost-efficient to implement local refinement within the PU elements. That is, in order to ensure the compatibility between original and new elements, once a PU element is discretized into a flat-top PU mesh, which includes both flat-top elements and PU elements, all the other PU elements in the original mesh should be discretized. Therefore, it is desirable to implement local refinement within the flat-top elements, which are adjacent to the PU element concerned. Take the blue-colored PU element e_{12} and white-colored PU element e_9 in Figure 4.7 (a) as examples. The local refinement for element e_{12} is realized through refining its adjacent flat-top elements e_{11} and e_{13} , as shown in Figure 4.12 (a); and for element e_9 , it is realized by refining its adjacent flat-top elements e_3 , e_5 , e_{13} and e_{15} , as illustrated in Figure 4.12 (b).

It is obvious that the local refinement for PU elements can be treated as a simple combination of the three basic cases. According to the rank deficiency counting approach, it is linearly independent for refining PU elements in this cost-efficient way.

(a) Local refinement for blue-colored PU element e_{12}

(b) Local refinement for white-colored PU element e_9

Figure 4.12 Local refinement for PU elements

4.5 numerical examples

Two numerical examples, i.e. an infinite plate with a circular hole and an L-shaped structure, are simulated and analyzed in this section to verify the necessity and efficiency of local refinement in flat-top PU mesh.

4.5.1 Infinite plate with a circular hole

For the problem of infinite plate with a circular hole, due to its property of symmetry, only a quarter of the domain will be simulated. As illustrated in Figure 4.13, a $10m \times 10m$ plate with a quarter circular hole located at its lower left corner is modelled, and the radius of the circle is $R = 1m$. The left boundary is fixed in x direction and the lower boundary is fixed in y direction. The right boundary is subjected to uniform tension with $q = 1MPa$. The Young's modulus E and Poisson's ratio ν for this problem are $10GPa$ and 0.3 , respectively.

Figure 4.13 Simplified model for an infinite plate with a circular hole subjected to uniformly distributed tension force

The analytical solution for the stress in x direction σ_x in this problem is given by Tian and Yagawa (2005) as:

$$\sigma_x = q \left[1 - \frac{R^2}{r^2} \left(\frac{3}{2} \cos 2\theta + \cos 4\theta \right) + \frac{3R^4}{2r^4} \cos 4\theta \right] \quad (4.16)$$

where (r, θ) are the polar coordinates, and the relationship between polar coordinates (r, θ) and Cartesian coordinates (x, y) can be expressed as:

$$\begin{cases} x = r \cos \theta \\ y = r \sin \theta \end{cases} \quad (4.17)$$

Three numerical models, i.e. Models 1, 2 and 3 in Figure 4.14, are constructed to analyze this problem. For Model 1 in Figure 4.14 (a), the problem domain is coarsely discretized by a regularly patterned flat-top PU mesh with 4 flat-top PU units, which includes 4 flat-top elements and 5 PU elements. While for Model 2 in Figure 4.14 (b), in order to obtain more accurate numerical results, the plate is discretized into 49 flat-top PU units with 49 flat-top elements and 120 PU elements, which is still regularly patterned flat-top PU mesh. However, for this kind of stress concentration problem, the stress concentration is around the hole area only, while the stress distribution further away from the hole is very monotonous. Therefore, compared with Model 2, Model 3 in Figure 4.14 (c) is more cost-efficient. This model utilizes the local refinement theory introduced in this chapter, and the local refinement is implemented based on the regularly patterned flat-top PU mesh in Model 1. That is the flat-top element containing the quarter circular hole in Model 1 is locally-refined in Model 3. Correspondingly, 28 flat-top PU units are generated in this locally-refined model, which include 27 flat-top elements and 77 PU elements.

(a) Model 1

(b) Model 2

(c) Model 3

Figure 4.14 Three different numerical models

In order to compare the accuracy of numerical results for different models, the corner point A in Figure 4.13 is chosen as an observation point. The errors of σ_x at point A in three different models with different local approximation degree p are listed in Table 4.3. For the purpose of obtaining an insight analysis of the cost-efficiency for these three models, the numbers of DOFs for each flat-top PU mesh with different local approximation degree p are also listed in Table 4.3.

From Table 4.3, it can be observed that, with the same local approximation degree p , the number of DOFs for Model 1 is smallest and, accordingly, its error is largest compared with other two models. Moreover, compared with Model 2, the number of DOFs in Model 3 is less, while the numerical results in Model 3 are generally more accurate. To be specific, the error for $p = 4$ in Model 3 is only 7.91%, which is less than that (8.54%) for $p = 5$ in Model 2. Meanwhile, the number of DOFs in Model 3 is only 40.8% of that in Model 2. It is also observed that the error for $p = 6$ in Model 3 is only 0.43%. While with almost the same number of DOFs, the error is 9.29% in Model 2 with $p = 4$.

Furthermore, the left boundary AB in Figure 4.13 is taken as an observation line in Models 2 and 3. The numerical results for σ_x along line AB with $p = 1, 3$ and 6 and the corresponding analytical solution are plotted in Figure 4.15, where Figure 4.15 (a) is for Model 2, while Figure 4.15 (b) for Model 3. From these two figures, it can be concluded that: 1) the improvement of numerical accuracy with the increase of

degree p is observed for both models; 2) the numerical results near Point A ($1 \leq y \leq 3$) are more accurate in Model 3, especially when the degree p is small.

Table 4.3 Error of σ_x at point A and number of DOFs for three different models with different local approximation degree p

	Model	Degree p					
		1	2	3	4	5	6
Error (%)	1	66.09	64.81	61.64	55.96	48.74	41.55
	2	53.19	25.10	13.15	9.29	8.54	3.72
	3	46.51	25.93	13.24	7.91	5.63	0.43
No. of DOFs	1	12	24	40	60	84	112
	2	147	294	490	735	1029	1372
	3	84	168	280	420	588	784

(a) σ_x along line AB for Model 2 with different local approximation degree p

(b) σ_x along line AB for Model 3 with different local approximation degree p

Figure 4.15 σ_x along line AB for Models 2 and 3 with different degree p

(a) Contour for σ_x of Model 3 with degree $p = 6$ (unit: MPa)

(b) Contour for σ_x of analytical solution (unit: MPa)

Figure 4.16 Contours for σ_x of Model 3 ($p = 6$) and analytical solution

The contours for σ_x of Model 3 with $p = 6$ and analytical solution are presented in Figure 4.16 (a) and Figure 4.16 (b), respectively. From these two contours, it can be observed that the numerical results of Model 3 for the whole problem domain are quite accurate compared with the analytical solution. The highest tensile stress appears at the upper corner of the quarter circular hole, which is around 3 MPa . It is also found that the distribution of stress σ_x in the area a little far away from the hole is quite monotonous, which also strongly supports the necessity of implementing local refinement. So that, the numerical analysis can be more cost-efficient.

4.5.2 L-shaped structure

An L-shaped structure with a sharp re-entrant corner subjected to uniformly distributed shear force is analyzed. The dimensions, boundary conditions and material properties are illustrated in Figure 4.17. Four numerical models are utilized for simulation as shown in Figure 4.18. Model 1 is generated by cutting a regularly patterned flat-top PU mesh with the physical boundary. While Models 2, 3 and 4 are constructed by implementing local refinement within the L-shaped flat-top element in Models 1, 2 and 3, respectively. The last step of local refinement in Model 4 is enlarged as illustrated in Figure 4.18 (d). In the four models, the sizes for flat-top PU unit at the sharp re-entrant corner are $0.4m$, $0.14m$, $0.04m$ and $0.014m$, respectively.

Figure 4.17 Basic information for an L-shaped domain

Figure 4.18 Four different numerical models

The total displacement U and von Mises stress σ_v at Point A in Figure 4.17 will be observed in these models, which are calculated as:

$$\begin{aligned}
 U &= \sqrt{U_x^2 + U_y^2} \\
 \sigma_v &= \sqrt{\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\sigma_{xy}^2}
 \end{aligned} \tag{4.18}$$

Theoretically, along with the further local refinement in Figure 4.18 and the increase of local approximation degree p , the displacement U at Point A should converge to a finite value; while the von Mises stress σ_v at Point A should be infinite because of stress singularity (Kurowski, 2016). The numerical results

obtained from four flat-top PU models with different local approximation degree p are listed in Table 4.4. The results of ABAQUS are presented in the last column as reference, where 7500 8-node quadrilateral elements with the size of $0.01m$ is utilized.

Table 4.4 Displacement U and von Mises stress σ_v at Point A for different numerical models

	Model	Degree p						Reference
		1	2	3	4	5	6	
U (mm)	1	0.21	0.65	0.69	0.69	0.69	0.70	0.65
	2	0.23	0.68	0.69	0.69	0.69	0.69	
	3	0.22	0.68	0.68	0.68	0.68	0.68	
	4	0.22	0.67	0.68	0.68	0.68	0.68	
σ_v (MPa)	1	1.40	4.06	5.89	6.36	7.77	8.03	44.62
	2	4.49	9.58	12.26	12.60	12.80	15.09	
	3	7.30	16.50	16.94	18.20	21.77	22.02	
	4	12.48	26.46	28.57	29.70	29.98	30.40	

From Table 4.4, it can be observed that: 1) the displacement U at Point A converges to $0.68mm$ in flat-top PU models, which is close to the reference value of $0.65mm$, 2) the von Mises stress σ_v at Point A keeps increasing along with the further local refinement of flat-top PU mesh and the increase of local approximation degree p , and the reference obtained from a smaller-sized FEM mesh is even larger than the results of flat-top PU models.

For the cases that the local approximation degree p is equal to 1, 3 and 6, respectively, the relationship between von Mises stress σ_v and unit size at Point A in flat-top PU models is depicted in Figure 4.19. It can be observed that, for each case,

the von Mises stress σ_v increases rapidly and tends to be infinite when the unit size decreases from $0.05m$. However, for the same flat-top PU mesh with fixed unit size, the von Mises stress σ_v tends to converge when increasing the local approximation degree p , which can also be observed in Table 4.4.

Figure 4.19 Relationship between von Mises stress σ_v and unit size at Point A

From the analysis of these two numerical examples, conclusions can be made that: 1) for stress concentration problems, compared with the regularly patterned flat-top PU mesh, the local refinement theory introduced in this chapter can efficiently reduce the number of DOFs and obtain a more accurate numerical result; 2) for stress singularity problems, even though the stress increases along with the local refinement within flat-top PU mesh as theoretically predicted, a stable result for displacement can still be achieved. Therefore, it is quite necessary and efficient for implementing local refinement when solving the stress concentration and stress singularity problems.

4.6 Conclusions

In this chapter, with the brief review of equivalent p -version element and rank deficiency counting approach, the construction of locally refined flat-top PU mesh and proof of its linear independence are presented from element level for one- and two-dimensional space, respectively. Based on this solid theoretical foundation, two numerical examples concerning about stress concentration and stress singularity are simulated and analyzed in detail to verify the necessity and efficiency of

implementing local refinement. Several conclusions can be addressed for this chapter as follows:

1. The local refinement within regularly patterned flat-top PU mesh in one- and two-dimensional spaces is easy to implement according to the requirement for ensuring continuity between original and new elements.
2. Based on the equivalent p -version element and rank deficiency counting approach, the property of linear independence for locally refined flat-top PU mesh can be proven from element level for both one- and two-dimensional cases.
3. As for the three-dimensional locally refined flat-top PU mesh with hexahedral element, its construction and proof of linear independence can be potentially implemented in the same way as in one- and two-dimensional cases.
4. For stress concentration problems, compared with the regularly patterned flat-top PU mesh, the numerical results in locally refined mesh can be obtained with higher accuracy and less DOFs.
5. For the stress singularity problem, as theoretically predicted, the stress at singular point tends to be infinite along with the further local refinement of flat-top PU mesh; meanwhile, convergence of displacement results can still be achieved.
6. In summary, it is necessary and efficient to implement local refinement within flat-top PU mesh considering stress concentration and stress singularity problems.

CHAPTER 5 FLAT-TOP PU BASED HIGH-ORDER DDA METHOD

5.1 Introduction

In Chapters 3 and 4, the flat-top PU based high-order approximation is used to simulate continuous problem domain. However, the existence of discontinuity significantly influences the mechanical behavior of materials (Bieniawski et al., 1969; Taylor et al., 1986; Gentier et al., 2000). Therefore, DDA method was developed to simulate large displacement and large deformation of discontinuous system (Shi, 1988, 2001; Tsesarsky et al., 2005; Tsesarsky and Hatzor, 2006). Compared with other numerical methods in simulating discontinuous system, the DDA method has the following three advantages: 1) it is fundamentally discontinuous between blocks; 2) it is an implicit displacement method, which is fast, accurate and has correct energy consumption; 3) it complies with strict postulate of equilibrium and complete kinematic theory.

Therefore, in this chapter, flat-top PU based high-order approximation is extended to simulating discontinuous problem domain by coupling with DDA method. To be specific, in flat-top PU based high-order DDA method, the flat-top PU mesh is used to discretize the continuous parts of the problem domain, and the interaction between discontinuous blocks is simulated with the improved contact algorithm derived according to original DDA method. So that, only main discontinuities are considered in flat-top PU based high-order DDA method, instead of dividing the whole problem domain into many more small blocks as in original DDA method. As a result, the time for contact detection and iteration can be reduced.

In this chapter, the construction of flat-top PU mesh for discontinuous problem domain is introduced in Section 5.2 for both one- and two-dimensional cases, where optimization for flat-top PU mesh is implemented according to the calculation of percentage of effective area for each flat-top PU unit. Then, based on the brief review of contact algorithm in original DDA method, the improved contact algorithm for one- and two-dimensional flat-top PU mesh is presented in Section 5.3. After that, several numerical examples are simulated and analyzed to verify the flat-top PU based high-order DDA from different perspectives in Section 5.4. Finally, conclusions for this chapter are drawn in Section 5.5.

5.2 Construction of flat-top PU mesh with discontinuity

In this section, the construction of regularly patterned flat-top PU mesh for discontinuous problem domain is presented for both one- and two-dimensional cases. According to the formula derivation in Section 3.3, it is observed that all the elements in the global stiffness matrix are non-negative. If a flat-top PU unit is too small after intersecting the mesh with the physical boundary and discontinuity, the non-zero elements in the rows and columns corresponding to this small unit will be much smaller than the other non-zero elements, and the difference can be orders of magnitude. Thus, the condition number of the global stiffness matrix is high, and the global stiffness matrix is ill-conditioned (Li et al., 2001). In order to avoid the generation of ill-conditioned stiffness matrix, optimization for flat-top PU mesh is implemented based on the calculation of percentage of effective area for each flat-top PU unit. Actually, the construction of locally refined flat-top PU mesh for discontinuous problem domain is in the exactly same way as for the regularly patterned mesh, which will not be introduced in this chapter.

5.2.1 One-dimensional case

The construction of flat-top PU mesh for discontinuous problem domain in one-dimensional space is easy and straightforward. Based on the different element types, i.e. flat-top elements and PU elements, two cases are introduced in this section.

For the first case, the discontinuity is located within a flat-top element. As illustrated in Figure 5.1 (a), the flat-top PU mesh is designed large enough to cover the whole discontinuous problem domain, and the discontinuity is located within flat-top element e_3 . After intersecting the flat-top PU mesh with physical boundary and discontinuity, the numerical model is generated as shown in Figure 5.1 (b), where the element e_3 is divided into two flat-top elements, i.e. $e_{3,1}$ and $e_{3,2}$. Correspondingly, the original flat-top PU unit u_3 is discretized into two independent units, namely $u_{3,1}$ and $u_{3,2}$. There is no PU element shared by units $u_{3,1}$ and $u_{3,2}$, which ensures the discontinuity between them. Thus, due to the appearance of discontinuity within the flat-top element, two new flat-top PU units are generated to replace the original one in the numerical model.

(a) Flat-top PU mesh over a discontinuous problem domain

(b) Numerical model with discontinuity located in flat-top element

Figure 5.1 Generation of numerical model through intersecting flat-top element with discontinuity

In the second case, the discontinuity is located within a PU element. As shown in Figure 5.2 (a), the discontinuity is designed within PU element e_4 , which is divided into two PU elements $e_{4,1}$ and $e_{4,2}$, as illustrated in Figure 5.2 (b). Since two flat-top PU units, u_3 and u_5 , overlap on element e_4 , each of them is discretized into two independent new units, which are $u_{3,1}$, $u_{3,2}$, $u_{5,1}$ and $u_{5,2}$, respectively. In this case, it is observed that the area for unit $u_{3,2}$ is quite small compared with other units. Therefore, in order to avoid ill-conditioned stiffness matrix in global equilibrium equations, the concept of percentage of effective area for a flat-top PU unit u_i , i.e. A_{pei} , is introduced here, which can be calculated as:

$$A_{pei} = \frac{A_{ei}}{A_e} \times 100\% \quad (5.1)$$

where A_{ei} is the effective area for unit u_i , which is expressed as:

$$A_{ei} = \int_{u_i} w_{fpi} du_i \quad (5.2)$$

where w_{fpi} is the shape function for unit u_i ; and A_e is the effective area for a complete flat-top PU unit, which can be calculated by Eq. (5.2) with flat-top PU unit u_i as a complete unit. According to Eq. (5.1), it can be calculated that the

percentage of effective area for unit $u_{3,2}$ in Figure 5.2 (b) is only 2.2%. Therefore, in order to avoid ill-conditioned stiffness matrix, the flat-top PU mesh in Figure 5.2 (b) should be optimized. Generally, with the calculation of percentage of effective area, the flat-top PU mesh is optimized according to the following requirements: 1) the flat-top PU unit u_i , whose percentage of effective area is less than 3%, will be taken over by its adjacent unit u_j , whose percentage of effective area is equal to or greater than 3%; 2) the units u_i and u_j should be located within the same continuous block; 3) for the case when unit u_i has more than one adjacent units with percentage of effective areas not less than 3%, unit u_j should be the one which shares the largest common area with unit u_i . It is worth to note that the parameter 3% is determined by trial-and-error in the numerical examples in Section 5.4. If the minimum percentage of effective area in numerical model is less than 3%, the numerical results will be unreasonable due to ill-conditioned stiffness matrix. Therefore, in Figure 5.2 (b), the flat-top PU unit $u_{3,2}$ is taken over by its adjacent unit $u_{5,2}$, and the final numerical model is generated as illustrated in Figure 5.2 (c).

(a) Flat-top PU mesh over a discontinuous problem domain

(b) Intersection of PU element and discontinuity

(c) Numerical model after optimizing the flat-top PU mesh

Figure 5.2 Generation of numerical model through intersecting PU element with discontinuity

5.2.2 Two-dimensional case

The construction of flat-top PU mesh for discontinuous problem domain in two-dimensional space is also easy and convenient, where the procedure is similar with the one-dimensional case. Firstly, the problem domain shown in Figure 5.3 (a) is divided into two blocks by the discontinuity, i.e. Block 1 and Block 2. Then, a flat-top PU mesh, which is designed large enough to cover the whole problem domain, is placed over the problem domain, as illustrated in Figure 5.3 (b). After intersected by the physical boundary and discontinuity, the flat-top PU mesh is generated as shown in Figure 5.3 (c). It is observed that the curved line of physical boundary is approximated by straight segments for the convenience of numerical analysis. Moreover, if necessary, more vertices can be added in the straight boundary segments to improve the numerical accuracy in approximating the curved boundary line. Similar with the one-dimensional case, each flat-top PU unit cut through by the discontinuity is discretized into two independent units, which are located within Block 1 and Block 2, respectively.

(a) Problem domain with discontinuity

(b) Problem domain covered by flat-top PU mesh

(c) Flat-top PU mesh after intersected with boundary and discontinuity

Figure 5.3 Intersecting flat-top PU mesh with boundary and discontinuity

According to Eq. (5.1), the percentage of effective area for each flat-top PU unit in Figure 5.3 (c) is calculated. In order to avoid ill-conditioned stiffness matrix in global equilibrium equations, three flat-top PU units need to be optimized according to the requirement in Section 5.2.1. The three units are located within the areas surrounded by the red line in Figure 5.3 (c), which are enlarged as illustrated in Figures 5.4 (a) and 5.4 (c), respectively. Firstly, in Figure 5.4 (a), the PU element e_2 is shared by two flat-top PU units. One of the units takes flat-top element e_1 as elemental support, which is denoted as unit u_1 ; while another unit contains only one element e_2 after cutting by the physical boundary and discontinuity, and this latter unit is recognized as u_2 . After calculation, the percentage of effective area for the unit u_2 is only 0.7%. Therefore, according to the requirement of optimization for flat-top PU mesh in Section 5.2.1, the unit u_2 is taken over by its adjacent unit u_1 , as illustrated in Figure 5.4 (b). Secondly, for the elements in Figure 5.4 (c), the percentage of effective area for the flat-top PU unit, which contains elements e_3 , e_5 and e_6 , is only 1.4%, and, correspondingly, this unit is taken over by its adjacent unit with flat-top element e_4 as elemental support; meanwhile, the percentage of effective area for the flat-top PU unit, which is composed of elements e_5 , e_6 , e_8 and e_9 , is only 2.7%, and this unit is taken over by its adjacent unit with flat-top element e_{10} as elemental support. Therefore, after optimization, the elements in Figure 5.4 (d) are generated by combining the adjacent flat-top PU units.

Finally, the numerical model is generated as demonstrated in Figure 5.5, where the percentage of effective area for every flat-top PU unit is not less than 3%.

Figure 5.4 Optimization of elements in Block 1 and Block 2

Figure 5.5 Numerical model for discontinuous problem domain

5.3 Contact algorithm within the framework of flat-top PU method

For the simulation of discontinuous problem domain, contact between blocks is significantly important. Therefore, the contact algorithm in original DDA method is briefly reviewed in Section 5.3.1, and the improved contact algorithm for flat-top PU mesh in one- and two-dimensional cases are presented in Sections 5.3.2 and 5.3.3, respectively.

5.3.1 Contact algorithm in original DDA method

In original DDA method, the contact between blocks can be categorized into three different types: angle to angle, angle to edge and edge to edge, as illustrated in Figure 5.6 (Shi, 1988).

Figure 5.6 Three different contact types in original DDA method (Shi, 1988)

All the three contact types in Figure 5.6 can be transferred into point-line crossing inequalities, and the point-line crossing inequalities will be introduced in detail in Sections 5.3.2 and 5.3.3. In original DDA method, the point-line crossing inequalities can be classified into three types, i.e. convex angle vs. concave angle, convex angle vs. edge and convex angle vs. convex angle. Here, the contact between convex angle and concave angle is taken as an example. As shown in Figure 5.7, the bold lines P_2P_3 and P_2P_4 are recognized as reference lines. The relative position between convex angle vertex P_1 and two reference lines can be determined by inequalities, which will be introduced later in this section. As long as Point P_1 is determined penetrating into one of the reference lines, stiff spring will be added between them. For other two cases, the contact algorithm is exactly same with this one. Therefore, the numerical procedure for contact algorithm can be concluded as: 1) the reference lines are firstly selected; 2) the relative position between vertex and reference lines is determined by inequalities; 3) as long as penetration happens, stiff spring will be added correspondingly.

Figure 5.7 Contact between convex angle and concave angle (Shi, 1988)

In original DDA method, the problem domain is discretized into numbers of independent blocks to improve the numerical accuracy. Therefore, the detection and determination of contact between blocks and the adding and removing of stiff springs to achieve equilibrium are time-consuming. While in flat-top PU based high-order DDA method, since the displacement and stress fields can be approximated by flat-top PU mesh and high-order polynomial, only main discontinuities are considered. Therefore, it will be more efficient regarding to contact algorithm.

5.3.2 Contact algorithm for one-dimensional flat-top PU mesh

In one-dimensional space, the contact algorithm is introduced by taking the numerical model in Figure 5.2 (c) as an example. As shown in Figure 5.8 (a), all the vertices distributed on the boundary are recognized as potential penetration nodes, which are marked as red nodes. Correspondingly, all the lines connecting two adjacent potential penetration nodes are denoted as potential penetration lines, which are marked as green lines. In every time step, after solving the global equilibrium equations, all the potential penetration nodes are checked against each potential penetration line. Take the potential penetration node P_1 and potential penetration line P_2P_3 in Figure 5.8 (b) as an example. The red dot and green solid line represent the positions for potential penetration node and potential penetration line, respectively, at the beginning of current time step, which satisfies the following inequality:

$$(x_1 - x_2)(x_1 - x_3) \geq 0 \quad (5.3)$$

where x_i ($i = 1, 2, 3$) is the coordinate for vertex P_i at the beginning of current time step. After solving the global equilibrium equations, the new coordinate x'_i for vertex P_i can be obtained as:

$$x'_i = x_i + U_{P_i} \quad (5.4)$$

where U_{P_i} is the displacement for vertex P_i obtained by solving the global equilibrium equations. Substituting the coordinate x_i in Eq. (5.3) with new coordinate x'_i , if the inequality is still true, it is determined that no penetration happens between the node P_1 and line P_2P_3 . If not, the node P_1 and line P_2P_3 will be determined as penetration node and penetration line, respectively, which are represented by the red circle and green dashed line, respectively, in Figure 5.8 (b). At this time, a stiff spring is added between the node P_1 and line P_2P_3 .

(a) Potential penetration nodes in numerical model

(b) Determination of contact position

Figure 5.8 Contact algorithm in one-dimensional flat-top PU mesh

In original DDA method, the stiff spring is added at the beginning of current time step, i.e. from the moment when the coordinate for vertex P_i is x_i . Under this situation, since the stiffness for contact spring is quite large, energy loss happens in order to overcome excessive constraint, which will deteriorate the numerical accuracy, especially for the simulation of collision. Therefore, in flat-top PU based high-order DDA method, in order to improve the numerical accuracy in energy consumption, the exact contact position for node P_1 and line P_2P_3 is calculated, and the stiff spring is added from this exact contact position.

Firstly, the distance d_0 between node P_1 and line P_2P_3 at the beginning of current time step can be calculated as:

$$d_0 = \min \{|x_1 - x_2|, |x_1 - x_3|\} \quad (5.5)$$

Meanwhile, the vertex, which is closer to node P_1 , can be determined, regarding to two vertices P_2 and P_3 . For the example in Figure 5.8 (b), the vertex P_3 is closer to node P_1 at the beginning of current time step. Therefore, the penetration distance d_1 after solving the global equilibrium equations can be calculated as:

$$d_1 = |x'_1 - x'_3| \quad (5.6)$$

Supposing the displacement for every point is linear with time t within each time step Δt , the time t_0 spent on moving to the exact contact position can be calculated as:

$$t_0 = \frac{d_0}{d_0 + d_1} \Delta t \quad (5.7)$$

Hence, at the exact contact position, the coordinate x''_i for vertex P_i is calculated as:

$$x''_i = x_i + \frac{t_0}{\Delta t} U_{P_i} = x_i + \lambda U_{P_i} \quad (5.8)$$

where $\lambda = t_0/\Delta t$, which can be calculated by Eq. (5.7). At this time, the node P_1 and line P_2P_3 are represented by the purple dot and blue solid line, respectively, in Figure 5.8 (b), where the vertices P_1 and P_3 overlap at the purple dot. In flat-top PU based high-order DDA method, a spring with stiffness k_n is added from the moment when the vertex P_i moves to the contact position x''_i . Therefore, the potential energy Π_{kn} contributed by this stiff spring can be derived as:

$$\Pi_{k_n} = \frac{k_n}{2} \left\{ (x''_1 + \lambda' U(x_1)) - (x''_3 + \lambda' U(x_3)) \right\}^2 \quad (5.9)$$

where $\lambda' = (1 - \lambda)$; and $\lambda' U(x_i)$ is the displacement for vertex P_i , after the node P_1 and line P_2P_3 contact each other. According to Eq. (3.24), $U(x_i)$ can be expressed as:

$$U(x_i) = \sum_j [T'_{ij}(x_i)] [D_{ij}] \quad (5.10)$$

where the subscript ij refers to the j th flat-top PU unit u_{ij} that contains the element with vertex P_i as boundary. In one-dimensional space, $j = 1$ for a flat-top element, and $j = 1, 2$ for a PU element. Since $x''_1 = x''_3$ as illustrated by the purple dot in Figure 5.8 (b), substituting Eq. (5.10) into Eq. (5.9), it can be obtained that:

$$\begin{aligned} \Pi_{k_n} &= \frac{k_n (\lambda')^2}{2} \left(\sum_{1j} [T'_{1j}(x_1)] [D_{1j}] - \sum_{3j} [T'_{3j}(x_3)] [D_{3j}] \right)^2 \\ &= \frac{k_n (\lambda')^2}{2} \left(\sum_{1j} [T'_{1j}(x_1)] [D_{1j}] \right)^2 + \frac{k_n (\lambda')^2}{2} \left(\sum_{3j} [T'_{3j}(x_3)] [D_{3j}] \right)^2 - \\ &\quad k_n (\lambda')^2 \left(\sum_{1j} [T'_{1j}(x_1)] [D_{1j}] \right) \left(\sum_{3j} [T'_{3j}(x_3)] [D_{3j}] \right) \end{aligned} \quad (5.11)$$

According to the principle of virtual work, the derivative of Π_{kn} with respect to the unknowns $[D_{1j}]$ and $[D_{3j}]$ can be obtained as follows:

$$\begin{aligned} k_n (\lambda')^2 [T'_{1j}(x_1)]^T [T'_{1j'}(x_1)] &\rightarrow [K_{(1j)(1j')}] \\ k_n (\lambda')^2 [T'_{1j}(x_1)]^T [T'_{3j'}(x_3)] &\rightarrow [K_{(1j)(3j')}] \\ k_n (\lambda')^2 [T'_{3j}(x_3)]^T [T'_{1j'}(x_1)] &\rightarrow [K_{(3j)(1j')}] \\ k_n (\lambda')^2 [T'_{3j}(x_3)]^T [T'_{3j'}(x_3)] &\rightarrow [K_{(3j)(3j')}] \end{aligned} \quad (5.12)$$

where $j' = 1$ for a flat-top element, and $j' = 1, 2$ for a PU element; $[K_{(ij)(i'j)}]$ ($i, i' = 1, 3$) represents the sub-matrix positioned at the intersection of the row for flat-top PU unit u_{ij} and the column for unit $u_{i'j'}$ in global stiffness matrix.

It can be observed that, based on the different element types in flat-top PU mesh, the penetration can be classified into four groups, i.e. Types I ~ IV, as listed in Table 5.1. For different penetration types, the values for j and j' and the number of sub-matrices obtained from Eq. (5.12) are different, which are also presented in Table 5.1.

Table 5.1 Four types of penetration in one-dimensional case

Penetration type	I	II	III	IV
Element type for node P_1	Flat-top	Flat-top	PU	PU
$j =$	1	1	1, 2	1, 2
Element type for line P_2P_3	Flat-top	PU	Flat-top	PU
$j' =$	1	1, 2	1	1, 2
No. of sub-matrices	4	9	9	16

For every position where stiff spring has been added before establishing the global equilibrium equations, the contact force induced by the stiff spring is calculated after obtaining the new coordinates for penetration node and penetration line by solving the equations. If the contact force is determined as tension, the stiff spring will be removed to establish new global equilibrium equations.

With the newly added and removed stiff springs at contact positions, new global equilibrium equations can be established. Then, the determination of adding and removing stiff springs will be repeated after solving the new equations. This iteration continues within current time step until the requirement of no penetration and no tension is reached.

5.3.3 Contact algorithm for two-dimensional flat-top PU mesh

The basic procedure in implementing contact algorithm for two-dimensional flat-top PU mesh is similar with one-dimensional case. However, the process of deriving sub-matrices induced by adding stiff spring is different. Here, the numerical model in Figure 5.5 is taken as an example. As shown in Figure 5.9 (a), similar with one-dimensional case, all the red-colored vertices distributed along the boundary are potential penetration nodes, and all the green-colored sides connecting two adjacent potential penetration nodes are potential penetration lines. In every time step, after solving the global equilibrium equations, all the potential penetration nodes are checked against each potential penetration line. As illustrated in Figure 5.9 (b), the red dot P_1 and green solid line P_2P_3 represent the positions for potential penetration node and potential penetration line, respectively, before contact happens. The distance d_0 between node P_1 and line P_2P_3 is calculated as:

$$d_0 = \frac{1}{\sqrt{(x_3 - x_2)^2 + (y_3 - y_2)^2}} \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix} = \frac{S}{l} \leq 0 \quad (5.13)$$

where (x_i, y_i) are the coordinates for vertex P_i ; S represents the 3×3 determinant; and l is the length for the potential penetration line P_2P_3 . After solving the global equilibrium equations, the new coordinates (x'_i, y'_i) for vertex P_i can be obtained as:

$$\begin{cases} x'_i = x_i + U_{xP_i} \\ y'_i = y_i + U_{yP_i} \end{cases} \quad (5.14)$$

where (U_{xP_i}, U_{yP_i}) represent the displacement for vertex P_i obtained by solving the global equilibrium equations. Substituting the coordinates (x_i, y_i) in Eq. (5.13) with new coordinates (x'_i, y'_i) in Eq. (5.14), if the inequality is still true, it is determined that no penetration happens between the node P_1 and line P_2P_3 . If not, the node P_1 and line P_2P_3 will be determined as penetration node and penetration line, respectively, which are represented by the red circle and green dashed line, respectively, in Figure 5.9 (b). At this time, a stiff spring is added between the node P_1 and line P_2P_3 in the direction normal to line P_2P_3 . Same with one-dimensional case, the stiff spring is added from the position where the node P_1 and line P_2P_3 start to contact each other.

After solving the global equilibrium equations, the penetration distance d_1 , i.e. the distance between the red circle and green dashed line in Figure 5.9 (b), can be obtained by substituting coordinates (x_i, y_i) in Eq. (5.13) with new coordinates (x'_i, y'_i) in Eq. (5.14), which can be expressed as:

$$d_1 = \frac{\begin{vmatrix} 1 & x_1 + U_{xP_1} & y_1 + U_{yP_1} \\ 1 & x_2 + U_{xP_2} & y_2 + U_{yP_2} \\ 1 & x_3 + U_{xP_3} & y_3 + U_{yP_3} \end{vmatrix}}{\sqrt{(x_3 + U_{xP_3} - x_2 - U_{xP_2})^2 + (y_3 + U_{yP_3} - y_2 - U_{yP_2})^2}} > 0 \quad (5.15)$$

(a) Potential penetration nodes and lines in numerical model

(b) Determination of contact position

Figure 5.9 Contact algorithm in two-dimensional flat-top PU mesh

Since the time step Δt is short, the displacement (U_{xP_i}, U_{yP_i}) is so small that its second-order infinitesimal is negligible. Therefore, Eq. (5.15) can be simplified as:

$$d_1 = \frac{1}{l} \left(S + \begin{vmatrix} 1 & x_1 & U_{yP_1} \\ 1 & x_2 & U_{yP_2} \\ 1 & x_3 & U_{yP_3} \end{vmatrix} + \begin{vmatrix} 1 & U_{xP_1} & y_1 \\ 1 & U_{xP_2} & y_2 \\ 1 & U_{xP_3} & y_3 \end{vmatrix} \right) \quad (5.16)$$

According to Eq. (5.16), the contact between node P_1 and line P_2P_3 occurs when:

$$d_{t_0} = \frac{1}{l} \left(S + \begin{vmatrix} 1 & x_1 & \lambda U_{yP_1} \\ 1 & x_2 & \lambda U_{yP_2} \\ 1 & x_3 & \lambda U_{yP_3} \end{vmatrix} + \begin{vmatrix} 1 & \lambda U_{xP_1} & y_1 \\ 1 & \lambda U_{xP_2} & y_2 \\ 1 & \lambda U_{xP_3} & y_3 \end{vmatrix} \right) = 0 \quad (5.17)$$

where $\lambda = t_0/\Delta t$, and t_0 is the time needed for the node P_1 and line P_2P_3 moving from the red dot and green solid line to the purple dot and blue solid line, respectively, in Figure 5.9 (b). By solving Eq. (5.17), it can be obtained that:

$$\lambda = -S / \left(\begin{vmatrix} 1 & x_1 & U_{yP_1} \\ 1 & x_2 & U_{yP_2} \\ 1 & x_3 & U_{yP_3} \end{vmatrix} + \begin{vmatrix} 1 & U_{xP_1} & y_1 \\ 1 & U_{xP_2} & y_2 \\ 1 & U_{xP_3} & y_3 \end{vmatrix} \right) \quad (5.18)$$

Therefore, at the exact contact position, the coordinates (x''_i, y''_i) for vertex P_i can be calculated as:

$$\begin{cases} x''_i = x_i + \lambda U_{xP_i} \\ y''_i = y_i + \lambda U_{yP_i} \end{cases} \quad (5.19)$$

Then, a normal spring with stiffness k_n is added from the moment when the vertex P_i moves to the contact position (x''_i, y''_i) . Therefore, the potential energy Π_{kn} contributed by this stiff spring can be derived as:

$$\begin{aligned} \Pi_{k_n} &= \frac{k_n}{2} \frac{\begin{vmatrix} 1 & x''_1 + \lambda' U_x(x_1, y_1) & y''_1 + \lambda' U_y(x_1, y_1) \\ 1 & x''_2 + \lambda' U_x(x_2, y_2) & y''_2 + \lambda' U_y(x_2, y_2) \\ 1 & x''_3 + \lambda' U_x(x_3, y_3) & y''_3 + \lambda' U_y(x_3, y_3) \end{vmatrix}^2}{\left(x_3 + U_{xP_3} - x_2 - U_{xP_2} \right)^2 + \left(y_3 + U_{yP_3} - y_2 - U_{yP_2} \right)^2} \\ &\approx \frac{k_n}{2l^2} \left(\begin{vmatrix} 1 & x''_1 & \lambda' U_y(x_1, y_1) \\ 1 & x''_2 & \lambda' U_y(x_2, y_2) \\ 1 & x''_3 & \lambda' U_y(x_3, y_3) \end{vmatrix} + \begin{vmatrix} 1 & \lambda' U_x(x_1, y_1) & y''_1 \\ 1 & \lambda' U_x(x_2, y_2) & y''_2 \\ 1 & \lambda' U_x(x_3, y_3) & y''_3 \end{vmatrix} \right)^2 \\ &= \frac{k_n (\lambda')^2}{2l^2} \left\{ (a_1 \ a_2) \begin{pmatrix} U_x(x_1, y_1) \\ U_y(x_1, y_1) \end{pmatrix} + (a_3 \ a_4) \begin{pmatrix} U_x(x_2, y_2) \\ U_y(x_2, y_2) \end{pmatrix} + (a_5 \ a_6) \begin{pmatrix} U_x(x_3, y_3) \\ U_y(x_3, y_3) \end{pmatrix} \right\}^2 \end{aligned} \quad (5.20)$$

where $\lambda' = (1 - \lambda)$; $(\lambda' U_x(x_i, y_i), \lambda' U_y(x_i, y_i))$ represent the displacement for vertex P_i , after the node P_1 and line P_2P_3 contact each other; the second-order infinitesimal of displacement is omitted when expanding the determinant in the first row; since the node P_1 is located on line P_2P_3 when they contact each other, the following equality is adopted when expanding the determinant in the first row:

$$\begin{vmatrix} 1 & x''_1 & y''_1 \\ 1 & x''_2 & y''_2 \\ 1 & x''_3 & y''_3 \end{vmatrix} = 0 \quad (5.21)$$

and the coefficients a_i ($i = 1 \sim 6$) can be calculated by the coordinates for the purple dot and two vertices of the blue solid line in Figure 5.9 (b) as follows:

$$\begin{aligned} a_1 &= y_2'' - y_3'' & a_2 &= x_3'' - x_2'' & a_3 &= y_3'' - y_1'' \\ a_4 &= x_1'' - x_3'' & a_5 &= y_1'' - y_2'' & a_6 &= x_2'' - x_1'' \end{aligned} \quad (5.22)$$

According to Eq. (3.24), $U(x_i, y_i) = (U_x(x_i, y_i), U_y(x_i, y_i))^T$ can be expressed as:

$$U(x_i, y_i) = \sum_j [T'_{ij}(x_i, y_i)] [D_{ij}] \quad (5.23)$$

where the subscript ij refers to the j th flat-top PU unit u_{ij} that contains the element with point P_i as vertex. In two-dimensional case, $j = 1$ for a flat-top element, $j = 1, 2$ for a blue-colored PU element, and $j = 1 \sim 4$ for a white-colored PU element.

Substituting Eq. (5.23) into Eq. (5.20), it can be obtained that:

$$\Pi_{k_n} = \frac{k_n (\lambda')^2}{2l^2} \left\{ \begin{aligned} & (a_1 \ a_2) \sum_j [T'_{1j}(x_1, y_1)] [D_{1j}] + (a_3 \ a_4) \sum_j [T'_{2j}(x_2, y_2)] [D_{2j}] + \\ & (a_5 \ a_6) \sum_j [T'_{3j}(x_3, y_3)] [D_{3j}] \end{aligned} \right\}^2 \quad (5.24)$$

where $[T'_{2j}(x, y)] = [T'_{3j}(x, y)]$ and $[D_{2j}] = [D_{3j}]$, because the points P_2 and P_3 are the two vertices of line P_2P_3 , which are located within the same element. Therefore, Eq. (5.24) can be written as:

$$\Pi_{k_n} = \frac{k_n (\lambda')^2}{2l^2} \left\{ \begin{aligned} & (a_1 \ a_2) \sum_j [T'_{1j}(x_1, y_1)] [D_{1j}] + \\ & \sum_j \left\{ (a_3 \ a_4) [T'_{2j}(x_2, y_2)] + (a_5 \ a_6) [T'_{2j}(x_3, y_3)] \right\} [D_{2j}] \end{aligned} \right\}^2 \quad (5.25)$$

According to the principle of virtual work, the derivative of Π_{k_n} with respect to the unknowns $[D_{1j}]$ and $[D_{2j}]$ can be obtained as follows:

$$\begin{aligned} & \frac{k_n (\lambda')^2}{l^2} [T'_{1j}(x_1, y_1)]^T (a_1 \ a_2)^T (a_1 \ a_2) [T'_{1j'}(x_1, y_1)] \rightarrow [K_{(1j)(1j')}] \\ & \frac{k_n (\lambda')^2}{l^2} [T'_{1j}(x_1, y_1)]^T (a_1 \ a_2)^T \left\{ \begin{aligned} & (a_3 \ a_4) [T'_{2j'}(x_2, y_2)] + \\ & (a_5 \ a_6) [T'_{2j'}(x_3, y_3)] \end{aligned} \right\} \rightarrow [K_{(1j)(2j')}] \\ & \frac{k_n (\lambda')^2}{l^2} \left\{ \begin{aligned} & (a_3 \ a_4) [T'_{2j}(x_2, y_2)] + \\ & (a_5 \ a_6) [T'_{2j}(x_3, y_3)] \end{aligned} \right\}^T (a_1 \ a_2) [T'_{1j'}(x_1, y_1)] \rightarrow [K_{(2j)(1j')}] \\ & \frac{k_n (\lambda')^2}{l^2} \left\{ \begin{aligned} & (a_3 \ a_4) [T'_{2j}(x_2, y_2)] + \\ & (a_5 \ a_6) [T'_{2j}(x_3, y_3)] \end{aligned} \right\}^T \left\{ \begin{aligned} & (a_3 \ a_4) [T'_{2j'}(x_2, y_2)] + \\ & (a_5 \ a_6) [T'_{2j'}(x_3, y_3)] \end{aligned} \right\} \rightarrow [K_{(2j)(2j')}] \end{aligned} \quad (5.26)$$

where $j' = 1$ for a flat-top element, $j' = 1, 2$ for a blue-colored PU element; and $j' = 1 \sim 4$ for a white-colored PU element.

For two-dimensional case, based on the different element types, the penetration can be classified into nine groups, i.e. Types I ~ IX, as listed in Table 5.2.

Same with one-dimensional case, within each iteration, the contact force for each previously added stiff spring is calculated, and the spring will be removed if the contact force is determined as tension. The iteration of adding and removing stiff springs continues within current time step until the requirement of no penetration and no tension is reached.

It is worth to mention that the improved contact algorithm in Sections 5.3.2 and 5.3.3 is also applicable for the original DDA by setting the vertices of blocks as potential penetration nodes and the boundaries of blocks as potential penetration lines.

5.4 Numerical examples

In this section, numerical examples are presented and analyzed to test the improved contact algorithm for flat-top PU mesh. Since the test is implemented within the framework of dynamic analysis, two examples of free dropping and swing of pendulum are firstly analyzed to verify the efficiency and stability of flat-top PU mesh for dynamic analysis.

5.4.1 Free dropping

As show in Figure 5.10, a $1m \times 1m$ block is discretized into four flat-top PU units, which include four flat-top elements and five PU elements. The density ρ , Young's modulus E , Poisson's ratio ν and gravitational acceleration g for this problem domain are $2.65 \times 10^3 kg/m^2$, $66.25 GPa$, 0.3 and $9.8 m/s^2$, respectively. Therefore, in the situation of free dropping, the analytical solutions for vertical displacement U_y and vertical velocity V_y can be expressed as:

$$U_y = -\frac{g}{2}t^2 \quad V_y = -gt \quad (5.27)$$

where $t = \sum \Delta t$ is the total time consumed.

Table 5.2 Nine types of penetration in two-dimensional case

Penetration type	I	II	III	IV	V	VI	VII	VIII	IX
Element type for node P_1	Flat-top	Flat-top	Flat-top	Blue- colored PU	Blue- colored PU	Blue- colored PU	White- colored PU	White- colored PU	White- colored PU
$j =$	1	1	1	1, 2	1, 2	1, 2	1 ~ 4	1 ~ 4	1 ~ 4
Element type for line P_2P_3	Flat-top	Blue- colored PU	White- colored PU	Flat-top	Blue- colored PU	White- colored PU	Flat-top	Blue- colored PU	White- colored PU
$j' =$	1	1, 2	1 ~ 4	1	1, 2	1 ~ 4	1	1, 2	1 ~ 4
No. of sub-matrices	4	9	25	9	16	36	25	36	64

Figure 5.10 Numerical model for a free dropping block

In Figure 5.10, l_{ft} represents the length of sides for flat-top elements, while l_{pu} is the length of sides for PU elements that are not shared with flat-top elements. Therefore, the length ratio l_r is defined as:

$$l_r = l_{pu} / l_{ft} \quad (5.28)$$

In this example, six different values of l_r , i.e. 0.22, 0.50, 1.00, 1.33, 3.00 and 8.00, are selected to study its influence on the numerical results of dynamic analysis. For each case, the percentages of effective area for all four flat-top PU units are the same with each other. Furthermore, corresponding to six different values of l_r , the percentages of effective area are 82.6%, 69.4%, 56.3%, 51.0%, 39.1% and 30.9%, respectively. Considering each l_r , the time step is set as $\Delta t = 0.005s$ and $0.001s$, respectively. Hence, there are twelve cases in total. For each case, the displacement U_y and velocity V_y at the center of problem domain within first 3s are demonstrated and compared with analytical solutions, respectively, as illustrated in Figure 5.11. It is observed that the numerical results in every case are almost identical with the analytical solutions. The maximum error for all cases is less than 0.0075%. Therefore, it can be concluded that the influence of length ratio l_r is negligible and the flat-top PU mesh is efficient and stable in dynamic analysis.

(a) Displacement U_y for different l_r when $\Delta t = 0.005s$ (b) Velocity V_y for different l_r when $\Delta t = 0.005s$ (c) Displacement U_y for different l_r when $\Delta t = 0.001s$

(d) Velocity V_y for different l_r when $\Delta t = 0.001s$ Figure 5.11 Displacement U_y and velocity V_y for different length ratio l_r and time step Δt

5.4.2 Swing of pendulum

The dimensions for the pendulum are shown in Figure 5.12 (a), which is of large aspect ratio. Its amplitude θ is equal to 5° . In order to compare with the analytical solution, the unit weight of upper part with green-line boundary is set as zero, while the unit weight for the lower part with black-line boundary is $-71.932 \times 10^3 \text{ kg/m} \cdot \text{s}^2$. For both parts, the Young's modulus E and Poisson's ratio ν are 100 GPa and 0.25 , respectively. The upper part is hinged at the middle of its top side with the length of 0.006 m . With the assumption as follows: 1) the lower part circle is treated as a small particle; 2) $\sin\theta \approx \theta$ when $\theta \leq 5^\circ$, the period T can be expressed as:

$$T \approx 2\pi\sqrt{L/g} = 1.0000s \quad (5.29)$$

where $L = 0.218237 \text{ m} + 0.03 \text{ m}$ is the distance between the hinged point and center of the circle; $g = 9.8 \text{ m/s}^2$ is the gravitational acceleration.

In numerical model, the upper part is divided into two units, which include one flat-top element and one PU element e_1 , as shown in Figure 5.12 (b). The lower part is discretized into nine flat-top PU units with nine flat-top elements and sixteen PU elements. These two parts are connected by the PU element e_1 to ensure the continuity. After intersected by boundary, the numerical model is generated in Figure 5.12 (c), where the curved boundary is approximated by several straight lines.

Figure 5.12 Generation of numerical model for a pendulum

Table 5.3 Maximum and minimum percentages of effective area and period T for different cases

l_r		0.17	0.38	1.00	2.25	6.00
Percentage of effective area (%)	maximum	100.0	100.0	100.0	100.0	91.7
	minimum	38.6	35.1	27.4	18.0	15.1
$T / (s)$	$\Delta t = 0.001s$	1.004	1.004	1.004	1.004	1.004
	$\Delta t = 0.0005s$	1.0035	1.0040	1.0040	1.0040	1.0040

For the lower part with curved boundary, the influence of length ratio l_r is studied. The values for l_r are selected as 0.17, 0.38, 1.00, 2.25 and 6.00, respectively. The corresponding maximum and minimum percentages of effective area are listed in Table 5.3, which shows that no optimization is needed in this numerical example. For each l_r , two different situations with time step $\Delta t = 0.001s$ and $0.0005s$ are simulated. The period T obtained for every case is also presented in Table 5.3. It is observed that the numerical results are almost the same with each other and the maximum error is only 0.4% compared with the analytical solution. Therefore, it

can be concluded that the influence of length ratio l_r is negligible, and the dynamic analysis with flat-top PU mesh for problem with large aspect ratio and curved boundary is efficient and stable.

The displacements at the center of the circle within first $2s$ for two cases, i.e. $l_r = 6.00$, $\Delta t = 0.001s$ and $l_r = 0.17$, $\Delta t = 0.0005s$, are chosen to be depicted in Figure 5.13 (a) and 5.13 (b), respectively. Two similar smooth curves with period $T \approx 1.00s$ can be observed. These results further verify the accuracy and stability of flat-top PU mesh in dynamic analysis with large aspect ratio and curved boundary.

(a) Displacement for the pendulum when $l_r = 6.00$ and $\Delta t = 0.001s$

(b) Displacement for the pendulum when $l_r = 0.17$ and $\Delta t = 0.0005s$

Figure 5.13 Displacement for the pendulum with different length ratio l_r and time step Δt

5.4.3 Collision

Two blocks with the distance of $1m$ apart are simulated as a collision example. The dimensions of these two blocks are illustrated in Figure 5.14. For both blocks, the density ρ , Young's modulus E and Poisson's ratio ν are $2.65 \times 10^3 kg/m^2$, $66.25 GPa$ and 0.3 , respectively, and the acceleration a is $10m/s^2$ towards each other, while the initial velocity is equal to $0m/s$.

Figure 5.14 Problem domain for two collision blocks

For easy derivation of analytical solutions for the displacement U_x and velocity V_x , it is assumed that these two blocks are non-deformable rigid bodies. Therefore, according to the conservation of total momentum and total kinetic energy before and after collision, the analytical solutions for displacement U_x and velocity V_x before collision can be expressed as follows:

$$\begin{cases} U_x = at^2/2 \\ V_x = at \end{cases} \quad (\text{for Block 1}) \\ \begin{cases} U_x = -at^2/2 \\ V_x = -at \end{cases} \quad (\text{for Block 2}) \quad t \leq \sqrt{d/a} \quad (5.30)$$

where the direction from Block 1 towards Block 2 is defined as the positive direction; $d = 1m$ is the distance between two blocks. While the analytical solutions for displacement U_x and velocity V_x after collision can be expressed as:

$$\begin{cases} U_x = C_1 \cdot \sqrt{d \cdot a} (t - \sqrt{d/a}) + a(t - \sqrt{d/a})^2/2 \\ V_x = C_1 \cdot \sqrt{d \cdot a} + a(t - \sqrt{d/a}) \end{cases} \quad (\text{for Block 1}) \\ \begin{cases} U_x = C_2 \cdot \sqrt{d \cdot a} (t - \sqrt{d/a}) - a(t - \sqrt{d/a})^2/2 \\ V_x = C_2 \cdot \sqrt{d \cdot a} - a(t - \sqrt{d/a}) \end{cases} \quad (\text{for Block 2}) \quad t > \sqrt{d/a} \quad (5.31)$$

where C_1 and C_2 are coefficients determined by the mass of Block 1 (m_1) and Block 2 (m_2) as follows:

$$C_1 = \frac{m_1 - 3m_2}{m_1 + m_2} \quad C_2 = \frac{3m_1 - m_2}{m_1 + m_2} \quad (5.32)$$

As for the numerical analysis, the results obtained from both flat-top PU based DDA and original DDA are presented and analyzed. In flat-top PU based DDA method, a flat-top PU mesh is covered over the problem domain, as shown in Figure 5.15 (a). After intersected by physical boundary, the numerical model is generated as in Figure 5.15 (b), where the maximum and minimum percentages of effective area are 95.2% and 38.1%, respectively. It can be seen that there are two flat-top PU units within Block 1, including two flat-top elements and one PU element; while four flat-top PU units within Block 2, containing four flat-top elements and five PU elements. The eight red-colored vertices and six green-colored sides in Figure 5.15 (b) are recognized as potential penetration nodes and potential penetration lines, respectively. While in original DDA method, the numerical model is composed of two individual blocks as shown in Figure 5.14. The time step Δt is set as $0.0005s$ in both numerical methods.

(a) Two collision blocks covered by a flat-top PU mesh

(b) Numerical model in flat-top PU based DDA method

Figure 5.15 Generation of numerical model for two collision blocks

In Figure 5.16, the displacement U_x and velocity V_x at the center of each block within first 0.5s are demonstrated and compared with the analytical solutions in Eqs. (5.30) and (5.31), respectively.

It can be observed that, before the collision of two blocks, the numerical results from both methods are almost identical to the corresponding analytical solutions. The maximum error for displacements in original DDA method is 2.22%, except in the first time step, which is 20% for both blocks. The error of velocities in original DDA method is negligible. While in flat-top PU based DDA method, before collision, the maximum error for both displacements and velocities is less than 0.00002% in all the time steps. Therefore, in summary, these two numerical methods are accurate and the difference between them is negligible before collision.

(a) Displacement U_x for Block 1

(b) Velocity V_x for Block 1

(c) Displacement U_x for Block 2 (d) Velocity V_x for Block 2Figure 5.16 Displacement U_x and velocity V_x for Block 1 and Block 2

However, after collision, the difference between two numerical methods is obvious, which can be evidently observed in Figure 5.16. In flat-top PU based DDA method, for both blocks, the displacement difference between numerical results and analytical solutions increases along with time; while the velocity difference between them keeps more or less the same, and the average differences for velocity are 0.368m/s and -0.092m/s for Block 1 and Block 2, respectively. Therefore, it is easy to calculate that, after the moment of collision, the error for momentum is less than 0.01% ; while the loss of kinetic energy is about 8.97% . It is because that, unlike the analytical analysis, the blocks in numerical model are deformable. Therefore, a portion of the kinetic energy is transferred into elastic potential energy. Hence,

further study is carried out to enhance the rigidity of two blocks in numerical model. The results show that, if the Young's modulus E increases to $100.00GPa$, the loss of momentum and kinetic energy can be reduced to 0.006% and 6.12% , respectively. However, in the original DDA method, two blocks move towards left side with almost the same velocity after collision. The average difference of velocity between Block 1 and Block 2 is $-0.0008m/s$. It can be calculated that, after the moment of collision, the error of momentum is negligible; but the loss of kinetic energy is up to 63.72% , which is much more than the result obtained in flat-top PU based DDA method. The reason is that, without determining the exact position for contacting in original DDA method, the stiff spring between two blocks is added before the time contact happens. As a result, besides the kinetic energy transferred into elastic potential energy, a large part of kinetic energy is used to overcome the high resistance force between two blocks before collision.

Therefore, it can be concluded that, when simulating collision, the improved contact algorithm in flat-top PU based DDA method is more accurate than that in original DDA method.

5.4.4 Sliding of slope

A slope with the dimensions shown in Figure 5.17 (a) is analyzed in this section. The density ρ , Young's modulus E , Poisson's ratio ν and gravitational acceleration g are $2.65 \times 10^3 kg/m^2$, $66.25GPa$, 0.3 and $9.8m/s^2$, respectively. Block 2 is fixed and Block 1 slides along the boundary of Block 2 under the effect of gravity. It is assumed that the friction between two blocks is equal to zero. Therefore, the analytical solutions of displacement (U_x , U_y) and velocity (V_x , V_y) for Block 1 can be expressed as follows:

$$\begin{cases} U_x = -gt^2/5 \\ U_y = -gt^2/10 \end{cases} \quad \begin{cases} V_x = -2gt/5 \\ V_y = -gt/5 \end{cases} \quad (5.33)$$

As for the numerical analysis, a flat-top PU mesh is utilized to discretize the problem domain, as demonstrated in Figure 5.17 (b). However, after intersected by boundary and discontinuity, the maximum and minimum percentages of effective area are 99.9% and 0.2% , respectively, which will generate an ill-conditioned

(a) Slope with a square-shaped block and a right-triangle-shaped block

(b) Flat-top PU mesh for the slope

(c) Numerical model

Figure 5.17 Generation of numerical model for slope analysis

stiffness matrix. Therefore, in this example, optimization is carried out regarding to the flat-top PU units with the percentages of effective area as 0.2% and 0.4%. Correspondingly, an optimized numerical model is generated as illustrated in Figure 5.17 (c). At this time, the maximum and minimum percentages of effective area are 99.9% and 3.0%, respectively. There are four flat-top PU units within Block 1, including four flat-top elements and five PU elements; while twelve flat-top PU units within Block 2, containing twelve flat-top elements and 21 PU elements. The fifteen red-colored vertices and thirteen green-colored sides are recognized as potential penetration nodes and potential penetration lines, respectively. In Figure 5.17 (c), it is observed that some potential penetration nodes and potential penetration lines for Block 1 overlap with some of them for Block 2. In this numerical simulation, during the sliding of Block 1, all the nine types of penetration, as listed in Table 5.2, are involved.

In Figure 5.18, the numerical results of displacement (U_x , U_y) and velocity (V_x , V_y) at the center of Block 1 within first 0.9s are demonstrated and compared with analytical solution. It can be observed that the numerical results are almost identical with the corresponding analytical solutions. The errors for horizontal displacement U_x , horizontal velocity V_x , vertical displacement U_y and vertical velocity V_y are 0.00%, 0.00%, 0.03% and 0.16%, respectively, at the moment of 0.9s. Therefore, it can be concluded that the contact algorithm for all nine types of penetration is efficient and accurate.

(a) Horizontal displacement U_x

(b) Vertical displacement U_y

(c) Horizontal velocity V_x

(d) Vertical velocity V_y

Figure 5.18 Displacement (U_x , U_y) and velocity (V_x , V_y) for Block 1

5.5 Conclusions

In this chapter, the flat-top PU based high-order approximation is extended to the numerical simulation for discontinuous problem domain through coupling with the contact algorithm in original DDA method. The generation of numerical model with flat-top PU mesh for discontinuous problem domain is introduced. The formula for improved contact algorithm is derived. Moreover, several numerical examples are presented and analyzed to verify the accuracy, efficiency and stability of flat-top PU mesh and improved contact algorithm. Therefore, the following conclusions can be obtained for this chapter:

1. Based on the introduction of flat-top PU mesh in Chapter 3, the generation of numerical model with flat-top PU mesh for discontinuous problem domain is easy and straightforward.
2. In order to avoid ill-conditioned stiffness matrix in global equilibrium equations, optimization of flat-top PU mesh is carried out based on the calculation of percentage of effective area for each flat-top PU unit.
3. The formula for improved contact algorithm within the framework of flat-top PU mesh is derived in detail, which can improve the numerical accuracy by determining the exact contact position, especially for the simulation of collision.
4. The influence of length ratio l_r in dynamic analysis is negligible, which further verifies the construction flexibility and numerical stability of flat-top PU mesh.
5. The flat-top PU based DDA method is accurate and stable in simulating dynamic problem with large aspect ratio and curved boundary.

CHAPTER 6 FLAT-TOP PU BASED HIGH-ORDER DDA FOR STRESS WAVE PROPAGATION

6.1 Introduction

The study and simulation of stress wave are significantly important, considering the situations like the stability of buildings and underground structures under earthquake and blasting (Kutter and Fairhurst, 1971; Veletsos and Meek, 1974), the strength test and mechanical behavior study of materials under dynamic loading (Taylor et al., 1986; Guo et al., 2005), the discontinuity and material layer detection with elastic wave transmission and reflection (Popovics et al., 2000; Zerwer et al., 2003). Therefore, in this chapter, the flat-top PU based high-order DDA method is extended to the simulation of stress wave propagation.

In order to simulate stress wave propagation within infinite domain, non-reflective boundary is added to damp the reflection of stress wave in numerical model with limited dimensions, such as the unified boundary (White et al., 1977), the absorbing boundary (Clayton and Engquist, 1977) and the perfectly matched absorbing layer model (Collino and Tsogka, 2001). The classical standard non-reflective boundary is proposed by Lysmer and Kuhlemeyer (1969), where dashpots are added along the corresponding boundary to provide normal and tangential stresses in the direction opposite to the incident wave. Bao et al. (2012) improved its accuracy by deriving the analytical velocity of dashpots, instead of calculating the velocity with finite difference method. So that, the dashpots can provide higher absorbing efficiency. Therefore, in this chapter, the formula derived by Bao et al. (2012) is modified and incorporated into the flat-top PU based high-order DDA method.

In this chapter, the improved formula of classical standard non-reflective boundary is modified and coupled with two-dimensional flat-top PU based high-order DDA method in Section 6.2. Several numerical examples of stress wave propagation are presented and analyzed with detailed parametric study in Section 6.3. And the conclusions for this chapter are drawn in Section 6.4.

6.2 Non-reflective boundary

For the problems such as earthquake and blasting in underground rock cavern, stress wave propagates within an infinite domain. However, the dimensions of

numerical model are finite. Therefore, in this section, non-reflective boundary is incorporated into the flat-top PU based high-order DDA method. So that the influence of stress wave reflected by the numerical boundary can be minimized.

As introduced in Section 6.1, the improved formula for non-reflective boundary, which is derived by Bao et al. (2012), is modified and incorporated. As illustrated in Figure 6.1, the red line represents the boundary in numerical model, which actually should be infinitely far away. A pair of blue-colored dashpots are added along the boundary, which are oriented normal and tangential to the numerical boundary, respectively. The global and local coordinate systems are set as (x, y) and (x', y') , respectively. The relationship between these two coordinate systems can be expressed as:

$$\begin{cases} x' = x \sin \alpha - y \cos \alpha \\ y' = x \cos \alpha + y \sin \alpha \end{cases} \quad (6.1)$$

where α is the angle between x -axis and the boundary concerned, as shown in Figure 6.1. For simplification, Eq. (6.1) can be expressed in matrix form as:

$$\begin{pmatrix} x' \\ y' \end{pmatrix} = \begin{pmatrix} \sin \alpha & -\cos \alpha \\ \cos \alpha & \sin \alpha \end{pmatrix} \begin{pmatrix} x \\ y \end{pmatrix} = \begin{pmatrix} x' \\ y' \end{pmatrix} = [L] \begin{pmatrix} x \\ y \end{pmatrix} \quad (6.2)$$

where $[L]$ represents the transformation matrix for two coordinate systems.

Figure 6.1 Non-reflective boundary in numerical model

According to Lysmer and Kuhlemeyer (1969), the normal stress $\sigma_{x'}(x', y', t)$ and shear stress $\sigma_{y'}(x', y', t)$ for the element at the boundary can be expressed as:

$$\begin{cases} \sigma_{x'}(x', y', t) = a\rho V_p \cdot V_{x'}(x', y', t) = c_n \cdot V_{x'}(x', y', t) \\ \sigma_{y'}(x', y', t) = b\rho V_s \cdot V_{y'}(x', y', t) = c_t \cdot V_{y'}(x', y', t) \end{cases} \quad (6.3)$$

where a and b are dimensionless parameters, and when $a = b = 1$, Eq. (6.3) will be most effective in absorbing energy; ρ is the density; V_P and V_S represent the velocities for P -wave and S -wave, respectively, which can be expressed by the Young's modulus E , Poisson's ratio ν and density ρ as follows:

$$V_P = \sqrt{\frac{E(1-\nu)}{\rho(1+\nu)(1-2\nu)}} \quad V_S = \sqrt{\frac{E}{2\rho(1+\nu)}} \quad (6.4)$$

$V_x(x', y', t)$ and $V_y(x', y', t)$ are the normal and tangential velocities of the element, respectively; while $c_n = a\rho V_P$ and $c_t = b\rho V_S$ are the normal damping coefficient and tangential damping coefficient, respectively.

Within each time step Δt , the following equilibrium equations are satisfied by the element at the boundary:

$$\begin{cases} mA_{x'}(x', y', t) + c_n V_{x'}(x', y', t) = 0 \\ mA_{y'}(x', y', t) + c_t V_{y'}(x', y', t) = 0 \end{cases} \quad (6.5)$$

where m is the mass of the element, $A_{x'}(x', y', t)$ and $A_{y'}(x', y', t)$ are the normal and tangential accelerations of the element, respectively. When $t = 0$, the following initial conditions are satisfied by the boundary element:

$$\begin{cases} V_{x'}(x', y', 0) = V_{x'0}(x', y') & U_{x'}(x', y', 0) = 0 \\ V_{y'}(x', y', 0) = V_{y'0}(x', y') & U_{y'}(x', y', 0) = 0 \end{cases} \quad (6.6)$$

where $U_{x'}(x', y', t)$ and $U_{y'}(x', y', t)$ are the normal and tangential displacements of the element, respectively. Based on Eq. (6.6), the analytical solution for Eq. (6.5) can be obtained as:

$$\begin{cases} V_{x'}(x', y', t) = V_{x'0}(x', y') e^{-\frac{c_n t}{m}} & U_{x'}(x', y', t) = \frac{m}{c_n} V_{x'0}(x', y') \left(1 - e^{-\frac{c_n t}{m}}\right) \\ V_{y'}(x', y', t) = V_{y'0}(x', y') e^{-\frac{c_t t}{m}} & U_{y'}(x', y', t) = \frac{m}{c_t} V_{y'0}(x', y') \left(1 - e^{-\frac{c_t t}{m}}\right) \end{cases} \quad (6.7)$$

Combining the four equations in Eq. (6.7), the velocities $V_{x'}(x', y', t)$ and $V_{y'}(x', y', t)$ for the boundary element can be obtained as:

$$\begin{cases} V_{x'}(x', y', t) = V_{x'0}(x', y') - \frac{c_n}{m} U_{x'}(x', y', t) \\ V_{y'}(x', y', t) = V_{y'0}(x', y') - \frac{c_t}{m} U_{y'}(x', y', t) \end{cases} \quad (6.8)$$

According to Eq. (6.7), at the end of each time step, i.e. when $t = \Delta t$, the velocities $V_{x'\Delta t}(x', y')$ and $V_{y'\Delta t}(x', y')$ can be expressed as:

$$\begin{cases} V_{x'\Delta t}(x', y') = V_{x'0}(x', y')e^{-\frac{c_n\Delta t}{m}} \\ V_{y'\Delta t}(x', y') = V_{y'0}(x', y')e^{-\frac{c_t\Delta t}{m}} \end{cases} \quad (6.9)$$

Since the time step Δt is small, the total displacement $U_{x'}(x', y')$ and $U_{y'}(x', y')$ for current time step can be approximated as:

$$\begin{cases} U_{x'}(x', y') \approx \frac{V_{x'0}(x', y') + V_{x'\Delta t}(x', y')}{2} \Delta t \\ U_{y'}(x', y') \approx \frac{V_{y'0}(x', y') + V_{y'\Delta t}(x', y')}{2} \Delta t \end{cases} \quad (6.10)$$

Substituting Eq. (6.9) into Eq. (6.10), it can be derived that:

$$\begin{cases} V_{x'0}(x', y') = \frac{2U_{x'}(x', y')}{\Delta t \left(1 + e^{-\frac{c_n\Delta t}{m}}\right)} \\ V_{y'0}(x', y') = \frac{2U_{y'}(x', y')}{\Delta t \left(1 + e^{-\frac{c_t\Delta t}{m}}\right)} \end{cases} \quad (6.11)$$

Therefore, according to Eqs. (6.3), (6.8) and (6.6), the potential energy Π_{ck} for the k th element e_k induced by non-reflective boundary can be calculated as:

$$\begin{aligned} \Pi_{ck} &= \iint \left[\int \sigma_{x'}(x', y', t) dU_{x'}(x', y', t) + \int \sigma_{y'}(x', y', t) dU_{y'}(x', y', t) \right] dx dy \\ &= \iint \left[\int c_n \cdot V_{x'}(x', y', t) dU_{x'}(x', y', t) + \int c_t \cdot V_{y'}(x', y', t) dU_{y'}(x', y', t) \right] dx dy \\ &= \iint \left\{ c_n \int \left[V_{x'0}(x', y') - \frac{c_n}{m} U_{x'}(x', y', t) \right] dU_{x'}(x', y', t) + \right. \\ &\quad \left. c_t \int \left[V_{y'0}(x', y') - \frac{c_t}{m} U_{y'}(x', y', t) \right] dU_{y'}(x', y', t) \right\} dx dy \\ &= \iint \left\{ c_n \left[V_{x'0}(x', y') U_{x'}(x', y') - \frac{c_n}{2m} U_{x'}^2(x', y') \right] + \right. \\ &\quad \left. c_t \left[V_{y'0}(x', y') U_{y'}(x', y') - \frac{c_t}{2m} U_{y'}^2(x', y') \right] \right\} dx dy \end{aligned} \quad (6.12)$$

where $(x, y) \in e_k$. Reorganizing the terms in Eq. (6.12), it can be obtained that:

$$\Pi_{ck} = \iint \left\{ \begin{aligned} & (V_{x'0}(x', y') \quad V_{y'0}(x', y')) \begin{pmatrix} c_n & 0 \\ 0 & c_t \end{pmatrix} \begin{pmatrix} U_{x'}(x', y') \\ U_{y'}(x', y') \end{pmatrix} - \\ & \frac{1}{2m} (U_{x'}(x', y') \quad U_{y'}(x', y')) \begin{pmatrix} c_n & 0 \\ 0 & c_t \end{pmatrix} \begin{pmatrix} c_n & 0 \\ 0 & c_t \end{pmatrix} \begin{pmatrix} U_{x'}(x', y') \\ U_{y'}(x', y') \end{pmatrix} \end{aligned} \right\} dx dy \quad (6.13)$$

According to Eqs. (6.2) and (3.24), Eq. (6.13) can be derived as:

$$\begin{aligned} \Pi_{ck} &= \iint \left\{ \begin{aligned} & (V_{x'0}(x', y') \quad V_{y'0}(x', y')) [C] [L] \begin{pmatrix} U_x(x, y) \\ U_y(x, y) \end{pmatrix} - \\ & \frac{1}{2m} (U_x(x, y) \quad U_y(x, y)) [L]^T [C]^T [C] [L] \begin{pmatrix} U_x(x, y) \\ U_y(x, y) \end{pmatrix} \end{aligned} \right\} dx dy \\ &= \iint \left\{ \begin{aligned} & (V_{x'0}(x', y') \quad V_{y'0}(x', y')) [C] [L] \left(\sum_{jk} [T'_{jk}(x, y)] [D_{jk}] \right) - \\ & \frac{1}{2m} \left(\sum_{ik} [T'_{ik}(x, y)] [D_{ik}] \right)^T [L]^T [C]^T [C] [L] \left(\sum_{jk} [T'_{jk}(x, y)] [D_{jk}] \right) \end{aligned} \right\} dx dy \end{aligned} \quad (6.14)$$

where

$$[C] = \begin{pmatrix} c_n & 0 \\ 0 & c_t \end{pmatrix} \quad (6.15)$$

and ik, jk refer to the i th/ j th flat-top PU unit that the element e_k pertains to, in two-dimensional space, $i, j = 1$ for a flat-top element, $i, j = 1, 2$ for a blue-colored PU element, and $i, j = 1 \sim 4$ for a white-colored PU element.

According to the principle of virtual work, the derivative of Π_{ck} with respect to the unknown $[D_{jk}]$ can be obtained as:

$$\frac{\partial \Pi_{ck}}{\partial [D_{jk}]} = \iint \left\{ \begin{aligned} & (V_{x'0}(x', y') \quad V_{y'0}(x', y')) [C] [L] [T'_{jk}(x, y)] - \\ & \frac{1}{m} \left(\sum_{ik} [T'_{ik}(x, y)] [D_{ik}] \right)^T [L]^T [C]^T [C] [L] [T'_{jk}(x, y)] \end{aligned} \right\} dx dy \quad (6.16)$$

According to Eqs. (6.11), (6.2) and (3.24), Eq. (6.16) can be derived as:

$$\frac{\partial \Pi_{ck}}{\partial [D_{jk}]} = \iint \left\{ \left(\sum_{ik} [T'_{ik}(x, y)] [D_{ik}] \right)^T [L]^T [A]^T [C] [L] [T'_{jk}(x, y)] - \frac{1}{m} \left(\sum_{ik} [T'_{ik}(x, y)] [D_{ik}] \right)^T [L]^T [C]^T [C] [L] [T'_{jk}(x, y)] \right\} dxdy \quad (6.17)$$

where

$$[A] = \begin{pmatrix} \frac{2}{\Delta t \left(1 + e^{-\frac{c_n \Delta t}{m}} \right)} & 0 \\ 0 & \frac{2}{\Delta t \left(1 + e^{-\frac{c_t \Delta t}{m}} \right)} \end{pmatrix} \quad (6.18)$$

According to the principle of virtual work and Eq. (6.17), the derivative of $\partial \Pi_{ck} / \partial [D_{jk}]$ with respect to the unknown $[D_{ik}]$ can be obtained as:

$$\frac{\partial \Pi_{ck}}{\partial [D_{ik}] \partial [D_{jk}]} = \iint \left\{ [T'_{ik}(x, y)]^T [L]^T [A]^T [C] [L] [T'_{jk}(x, y)] - \frac{1}{m} [T'_{ik}(x, y)]^T [L]^T [C]^T [C] [L] [T'_{jk}(x, y)] \right\} dxdy \quad (6.19)$$

Therefore, the following sub-matrix is obtained and added into the global stiffness matrix:

$$\iint \left\{ [T'_{ik}(x, y)]^T [L]^T [A]^T [C] [L] [T'_{jk}(x, y)] - \frac{1}{m} [T'_{ik}(x, y)]^T [L]^T [C]^T [C] [L] [T'_{jk}(x, y)] \right\} dxdy \rightarrow [K_{(ik)(jk)}] \quad (6.20)$$

where $[K_{(ik)(jk)}]$ represents the sub-matrix in global stiffness matrix, which is positioned at the intersection of the row for i th flat-top PU unit and the column for j th unit that the element e_k pertains to.

According to Eq. (6.20), it is observed that one, four and sixteen sub-matrices are derived for a flat-top element, a blue-colored PU element and a white-colored PU element, respectively.

6.3 Numerical examples

In this section, three examples are analyzed to verify the numerical accuracy of flat-top PU based high-order DDA method in simulating stress wave propagation.

6.3.1 Long elastic bar

As illustrated in Figure 6.2, the dimensions for this elastic bar are $100m \times 1m$. Firstly, a standard numerical model is constructed. The problem domain is discretized into 100 flat-top PU units by the flat-top PU mesh, which includes 100 flat-top elements and 99 PU elements. For each flat-top PU unit, a complete polynomial with degree $p = 2$ is assigned as local approximation. It is assumed that the horizontal length for a flat-top element is denoted as l_{ft} , while it is l_{pu} for a PU element. Thus, the length ratio l_r takes the value of l_{pu}/l_{ft} . In this standard numerical model, l_r is equal to $0.2/0.8 = 0.25$. The material density ρ , Young's modulus E and Poisson's ratio ν are $2.65 \times 10^3 kg/m^3$, $66.25 GPa$ and 0.3 , respectively. A one cycle sinusoidal P -wave is applied as incident stress pulse at the left end of the numerical model, which is expressed as:

$$F = \begin{cases} A \sin(2\pi t / T) & (0 \leq t \leq T) \\ 0 & (t > T) \end{cases} \quad (6.21)$$

where F represents the incident stress pulse; $A = 1 MPa$ is the amplitude; T is the period, and it is equal to $0.004s$ in the standard numerical model. The time step Δt is set as $0.00001s$. The right end of numerical model is recognized as non-reflective boundary.

In the following analysis, besides the simulation results with standard numerical model, the influence of different parameters is studied, which includes the number of flat-top PU units, degree p of local approximation polynomial, time step Δt , length ratio l_r , Young's modulus E , period T , and boundary condition. During the study of each parameter, all the other parameters are kept the same with standard numerical model. In order to analyze the numerical results, two observation points are set at $30m$ away from left end and right end, which are Point A and Point B, respectively.

Figure 6.2 Numerical model of stress wave propagation within an elastic bar

Wave propagation with standard numerical model

For the standard numerical model, as shown in Figure 6.3, the stress waveforms at Point A and Point B within the first 0.04s are depicted and compared with analytical solution, respectively. It can be observed that, when the incident *P*-wave arrives at Point A and Point B, the waveforms generated by numerical simulation are almost identical with the corresponding analytical solutions, and the errors of peak pressure are 2.8% and 7.4% for Point A and Point B, respectively. Generally, in numerical

(a) Stress waveform at Point A

(b) Stress waveform at Point B

Figure 6.3 Stress waveforms at Point A and Point B in standard numerical model

Figure 6.4 Attenuation of peak pressure in standard numerical model

simulation, although the material is defined as linear elastic, the peak pressure decreases along with the propagation of stress wave, which is caused by the accumulation of numerical error. The attenuation of peak pressure within the first 80m is presented in Figure 6.4, where the largest error of peak pressure is 8.5%.

It can be calculated that, at 0.02s, the incident *P*-wave arrives at the right end of elastic bar. Meanwhile, the stress wave will be reflected by the right end. The reflected *P*-wave arrives at Point B at 0.026s and Point A at 0.034s, respectively. In this standard numerical model, a pair of dashpots are attached at the right end to absorb the stress wave. Therefore, theoretically, the peak pressure for reflected *P*-wave should be zero. From Figure 6.3, it is observed that the reflected stress wave in numerical simulation is negligible, where the peak pressure is only 1.2% of the incident wave. Therefore, it can be concluded that, based on Lysmer and Kuhlemeyer (1969) and Bao et al. (2012), the formula of non-reflective boundary derived for flat-top PU based high-order DDA method in Section 6.2 is accurate and effective.

Influence of unit number and local approximation degree p

The influence of unit number is studied by discretizing the long elastic bar into 40, 60, 80, 100 and 120 flat-top PU units, respectively. For each case, local approximation polynomial with degree p equal to one and two are employed, respectively.

Firstly, the numerical results with different numbers of flat-top PU units are compared under the condition of same degree p . Therefore, the attenuation of peak pressure along the propagation of stress wave within first 80m is presented in Figure 6.5 (a) for $p = 1$ and Figure 6.5 (b) for $p = 2$, respectively. From Figure 6.5 (a), it can be seen that, for $p = 1$, the numerical results tend to be more stable during wave propagation when increasing the number of flat-top PU units. Meanwhile, the attenuation of peak pressure, which is caused by accumulation of numerical error, becomes less. In Figure 6.5 (b), it can be observed that, when $p = 2$, the numerical results are stable during wave propagation for almost all conditions, except the case with only 40 flat-top PU units. Moreover, the difference between these four cases with 60 to 120 units is small, especially for the three cases with 80, 100 and 120 units, of which the difference is less than 0.42%.

(a) Peak pressure attenuation for different numbers of units when $p = 1$

(b) Peak pressure attenuation for different numbers of units when $p = 2$

Figure 6.5 Peak pressure attenuation for different local approximation degree p and numbers of flat-top PU units

Next, the numerical results with p equal to 1 and 2 are compared under the same number of flat-top PU units. The peak pressure attenuation and waveform with different degree p are presented in Figure 6.6 for four cases with 60, 80, 100 and 120 units, respectively. Firstly, the attenuation of peak pressure within first 80m is illustrated in Figure 6.6 (a) and Figure 6.6 (b) for 60 units and 100 units, respectively. It can be seen that the numerical results for $p = 2$ are more stable compared with that for $p = 1$. However, when the number of flat-top PU units is increased, the difference between these results with different degree p becomes smaller. Secondly, the waveform at Point A within first 0.02s is presented in Figure 6.6 (c) and Figure 6.6 (d) for 80 and 120 units, respectively. It is observed that, when $p = 2$, the numerical results are almost identical with the analytical solution for both cases. However, difference exists between the numerical results with $p = 1$ and analytical solution. Specifically, fluctuation appears before the stress wave arrives at Point A, which is the same as the observation from original DDA (Gu and Zhao, 2009); and after the arrival of stress wave, the waveform for 80 flat-top PU units does not consistent with the analytical solution completely when $p = 1$, but the difference is negligible.

The number of DOFs for the case with $p = 1$ and 120 units is equal to that for the case with $p = 2$ and 60 units, i.e. 360, but the errors of peak pressure at 80m away from wave source are 9.03% and 8.24%, respectively, and the numerical results are more stable in the latter one. It means that, even with the same number of DOFs, the numerical results for $p = 2$ are more accurate and stable than those for $p = 1$.

(a) Peak pressure attenuation for 60 flat-top PU units

(b) Peak pressure attenuation for 100 flat-top PU units

(c) Waveform at Point A for 80 flat-top PU units

(d) Waveform at Point A for 120 flat-top PU units

Figure 6.6 Peak pressure attenuation and waveform at Point A for different numbers of flat-top PU units and local approximation degree p

In summary, it can be concluded that: 1) for the case of $p = 1$, the loss of energy caused by accumulation of numerical error decreases with the increase of number of flat-top PU units; 2) for the case of $p = 2$, the numerical results tend to be similar with each other when increasing the number of units; 3) from the view of points on both peak pressure attenuation and waveform, with the same number of flat-top PU units, the numerical results obtained from $p = 2$ are more accurate and stable than those from $p = 1$; 4) When the degree p is increased from 1 to 2, while the number of DOFs is kept the same by reducing the number of flat-top PU units, the error of peak pressure can still be reduced and the numerical stability can also be ensured.

Influence of time step Δt

The influence of time step Δt is studied by setting its values ranging from 0.000005s to 0.0001s.

Firstly, for five cases with different time step Δt , the attenuation of peak pressure within the first 80m is presented in Figure 6.7. It can be observed that the loss of energy increases when the time step Δt becomes larger, which is caused by the accumulation of numerical error in time integration. Then, the waveform at Point A within first 0.02s with time step Δt equal to 0.000005s, 0.00002s and 0.0001s, respectively, is compared with analytical solution in Figure 6.8. It can be seen that the difference between numerical results and analytical solution becomes smaller and smaller when decreasing the value of time step Δt ; especially when Δt is equal to 0.000005s, the numerical results are almost identical with the analytical solution. It means that decreasing time step Δt can effectively reduce the error caused by time integration. Moreover, the phenomenon of fluctuation before the arrival of stress wave does not appear even for the case of $\Delta t = 0.0001s$. Together with the numerical results presented in Figure 6.6 (c) and Figure 6.6 (d), it can be concluded that the fluctuation is caused by inaccurate integration of energy in physical domain rather than numerical error of time integration, which is because that the time integration only influences the movement of stress field between steps rather than the distribution of stress values within each step.

Figure 6.7 Peak pressure attenuation with different time step Δt Figure 6.8 Waveform at Point A for different time step Δt

Therefore, it can be concluded that: 1) decreasing the value of time step Δt can reduce the numerical error caused by time integration, so that the attenuation of peak pressure during wave propagation can be decreased, and this is the same as the observation from original DDA (Gu and Zhao, 2009); 2) the fluctuation of stress before the arrival of wave is caused by numerical error in integration of energy within physical domain rather than integration of time.

Influence of length ratio l_r

The influence of length ratio l_r is analyzed with seven different values ranging from 0.11 to 9.00.

As illustrated in Figure 6.9, the attenuation of peak pressure during stress wave propagation within first 80m is stable and almost identical with each other, except

the case with a value extremely small, i.e. $l_r = 0.11$. The largest difference between all these seven cases is 0.61%; while for the six cases with l_r ranging from 0.25 to 9.00, the largest difference is only 0.25%.

In Figure 6.10, the waveform within first 0.02s with l_r equal to 0.11, 1.00 and 9.00 is depicted and compared with analytical solution, where Figure 6.10 (a) and Figure 6.10 (b) are the results for Point A and Point B, respectively. It can be observed that the waveforms obtained by numerical simulation are almost identical with each other; and there is no fluctuation before the arrival of stress wave. The difference between numerical results and analytical solution in Figure 6.10 (a) is negligible; while in Figure 6.10 (b), the amplitude of numerical results is smaller than analytical solution due to the attenuation of peak pressure.

Figure 6.9 Peak pressure attenuation with different length ratio l_r

(a) Waveform at Point A with different length ratio l_r

(b) Waveform at Point B with different length ratio l_r Figure 6.10 Waveform at Point A and Point B for different length ratio l_r

Therefore, it can be concluded that, as long as the number of flat-top PU units is fixed, the influence of length ratio l_r is negligible, which further verifies the construction flexibility and numerical stability of flat-top PU mesh.

Relationship between Young's modulus E and P -wave velocity V_P

The value of Young's modulus E is selected from $6.625GPa$ to $106.0GPa$ to study its relationship with P -wave velocity V_P .

In the numerical model in Figure 6.2, since the elastic bar is very long, which is 100 times of the height, it can be treated as a one-dimensional example. Hence, the influence of Poisson's ratio ν in Eq. (6.4) can be eliminated. Therefore, the analytical solution for P -wave velocity V_P can be expressed as:

$$V_P = \sqrt{\frac{E}{\rho}} \quad (6.22)$$

While in the numerical simulation, the P -wave velocity V_P is calculated as:

$$V_P = \frac{d_P}{t_P} \quad (6.23)$$

where d_P represents the distance the peak pressure of P -wave propagates, t_P is the corresponding time spent. The numerical results of P -wave velocity V_P for different Young's modulus E are compared with analytical solution in Figure 6.11. It can be

observed that the numerical results are accurate with different Young's modulus E , and the largest error is less than 0.7%.

Figure 6.11 Relationship between Young's modulus E and P -wave velocity V_P

Therefore, it can be concluded that the flat-top PU based high-order DDA method is capable of simulating P -wave propagation with a highly-accurate velocity.

Influence of wave length

Here, the influence of wave length l_w is analyzed. Since the wave length l_w is determined by velocity V_P and period T as:

$$l_w = V_P \cdot T \quad (6.24)$$

the study is carried out for the following two cases: 1) the period T is fixed as 0.004s, while the velocity V_P changes from 1581m/s to 6325m/s through setting the value of Young's modulus E from 6.625GPa to 106.0GPa; 2) the velocity V_P is set as 5000m/s, but the period T is chosen from 0.00125s to 0.005s. For both cases, the time step Δt is kept as 0.00001s.

The error of peak pressure at Point B is illustrated in Figure 6.12, in which the blue-colored line is corresponding to the first case, while the yellow-colored line represents the second case. It is observed that, for both cases, the error of peak pressure increases with the decrease of wave length l_w . It is because that, when the wave length l_w becomes shorter, the whole stress wave will be distributed within fewer flat-top PU units. Therefore, it is more difficult to accurately describe the stress field with the same degree p . As a result, the accumulation of numerical error

turns into larger. It is also observed that the numerical results are more sensitive to the period T than the velocity V_P . This is because, since the time step Δt is fixed as $0.00001s$, the change of period T affects not only the description of stress field but also the accuracy of time integration. That is, when the period T becomes smaller, the accuracy of time integration for a cycle of stress wave decreases. Therefore, in the second case, the accuracy for both integrations of time and physical problem domain is affected; but in the first case, only the accuracy for integration of physical domain is influenced. Consequently, the numerical results are more sensitive to period T compared with the velocity V_P .

According to the observation in Kuhlemeyer and Lysmer (1973), in order to obtain accurate numerical results in one-dimensional case, the wave length l_w is suggested to be 8 to 12 times greater than the largest element length along the direction of wave propagation. In this numerical model, the largest element length along the direction of wave propagation is equal to $0.9m$. Therefore, the wave length l_w should be greater than $7.2m \sim 10.8m$. It can be calculated that, at the end of first wave length in each case, the error of peak pressure is less than 2.0%, except when velocity $V_P = 1581m/s$ and period $T = 0.004s$. For all the cases with error less than 2.0%, the wave length l_w ranges from $6.3m$ to $25.3m$, which shows a wider application scope than the suggestion in Kuhlemeyer and Lysmer (1973). Moreover, the numerical results obtained by flat-top PU based high-order DDA method are more accurate than the results obtained by original DDA method in Gu and Zhao (2009), whose error of peak pressure at the end of first wave length is around 5%.

Hence, it can be concluded that: 1) the error of numerical results will be greater if the wave length l_w becomes shorter, 2) the numerical simulation is more sensitive to period T compared with velocity V_P , 3) considering the requirement of wave length l_w with respect to the largest element length, the flat-top PU based high-order DDA method has a wider application scope than the suggestion in Kuhlemeyer and Lysmer (1973).

Figure 6.12 Error of peak pressure at Point B with different wave length l_w

Wave reflection and transmission at material interface

The elastic bar in Figure 6.2 is divided into two parts with equal length, i.e. 50m, and the two parts are assigned with different material properties. The first 50m, i.e. left half of the bar, has the same material properties as the standard numerical model, i.e. Young's modulus $E = 66.25\text{GPa}$ and density $\rho = 2.65 \times 10^3 \text{kg/m}^3$. While the Young's modulus E of the last 50m is changed into 16.56GPa , which is a quarter of the E -value for the first 50m; the density ρ of the last 50m is kept as $2.65 \times 10^3 \text{kg/m}^3$. According to Bedford and Drumheller (1994), at the interface of two different materials, the relationship between incident stress σ_i , reflected stress σ_r and transmitted stress σ_t satisfies the following requirements:

$$\begin{cases} \sigma_i + \sigma_r = \sigma_t \\ \frac{\sigma_i}{\sqrt{E_1 \rho_1}} - \frac{\sigma_r}{\sqrt{E_1 \rho_1}} = \frac{\sigma_t}{\sqrt{E_2 \rho_2}} \end{cases} \quad (6.25)$$

where E_1, ρ_1 and E_2, ρ_2 are the Young's modulus and density for first 50m and last 50m of the elastic bar, respectively. By solving Eq. (6.25), the reflected stress σ_r and transmitted stress σ_t can be obtained as follows:

$$\begin{cases} \sigma_r = -\frac{1}{3}\sigma_i \\ \sigma_t = \frac{2}{3}\sigma_i \end{cases} \quad (6.26)$$

The waveform within first 0.025s for Point A and Point B is presented in Figure 6.13 (a) and Figure 6.13 (b), respectively. It can be observed that the waveform is

stable for both cases. In order to assess the accuracy of wave reflection and transmission at the material interface, the peak pressure at Point B obtained in the standard numerical model is selected as σ_i , so that the influence of energy loss during wave propagation can be reduced. According to the values of peak pressure, it can be calculated that the error of reflection and transmission at the material interface are 0.1% and 0.01%, respectively.

(a) Waveform at Point A

(b) Waveform at Point B

Figure 6.13 Reflection and transmission of stress wave at the interface of two different materials

Therefore, it can be concluded that the flat-top PU based high-order DDA method is accurate and stable in simulating stress wave reflection and transmission between different materials.

Wave reflection at different boundary conditions

Different boundary conditions are applied at the right end of elastic bar in Figure 6.2. In the standard numerical model, the non-reflective boundary is simulated and only 1.2% energy is reflected, which proves the accuracy and efficiency of the formula derived in Section 6.2. Therefore, only the conditions of fixed boundary and free boundary are analyzed here. The reflected stress for fixed boundary σ_{fx} and free boundary σ_{fr} can be expressed as (Bedford and Drumheller, 1994):

$$\begin{cases} \sigma_{fx} = \sigma_i \\ \sigma_{fr} = -\sigma_i \end{cases} \quad (6.27)$$

In Figure 6.14, the waveform for Point A and Point B within first 0.04s are presented. Specifically, the numerical results in Figure 6.14 (a) and Figure 6.14 (b) are for Point A and Point B with fixed boundary, respectively; while Figure 6.14 (c) and Figure 6.14 (d) for Point A and Point B with free boundary, respectively. It is observed that the waveform is stable for all the four cases. In order to assess the accuracy of wave reflection under different boundary conditions, the peak pressures of incident wave and reflected wave at 20m away from the right end are selected. Thus, the influence of energy loss during wave propagation can be minimized. Therefore, it can be calculated that the errors of reflection for fixed boundary and free boundary are less than 4.9% and 4.8%, respectively, which include the error caused by energy loss when the wave propagates from the observation point to the right end as incident wave and from the right end back to the observation point as reflected wave.

Therefore, it can be concluded that the flat-top PU based high-order DDA method is efficient and stable in simulating wave reflection under different boundary conditions, including non-reflective boundary, fixed boundary and free boundary.

(a) Waveform at Point A with fixed boundary

(b) Waveform at Point B with fixed boundary

(c) Waveform at Point A with free boundary

(d) Waveform at Point B with free boundary

Figure 6.14 Reflection of stress wave under different boundary conditions at the right end

6.3.2 Infinite plate with a circular hole

For the stress wave propagation in infinite plate with a circular hole, due to its property of symmetry, only a quarter of the domain will be simulated. As illustrated in Figure 6.15, a $10m \times 10m$ plate with a quarter circular hole located at its lower left corner is modelled, and the radius for the circle is $R = 1m$. In the numerical model, the problem domain is discretized into 288 flat-top PU units, which includes 288 flat-top elements and 797 PU elements. The left boundary and lower boundary are fixed in x direction and y direction, respectively. The right boundary and upper boundary are applied with dashpots to prevent stress reflection. The uniformly distributed incident P -wave with peak pressure equal to $3MPa$ is applied perpendicular to the boundary of the circular hole. As shown in Figure 6.16, the period T for this incident P -wave is equal to $0.001s$. The density ρ , Young's modulus E and Poisson's ratio ν for this example are $2.65 \times 10^3 kg/m^3$, $66.25 GPa$ and 0.3 , respectively. In numerical simulation, the time step Δt is set as $0.00001s$, and the local approximation degree p is equal to 2 . The green-colored dots with coordinates $x = 2m, 3m, \dots, 8m$ and $y = 0m$ are selected as observation points.

Figure 6.15 Numerical model for stress wave propagation within infinite plate

Figure 6.16 Incident P-wave with peak pressure equal to 3MPa

In order to assess the numerical results, the peak pressure ratio is defined as the value of peak pressure at observation points divided by 3MPa, i.e. the peak pressure of incident *P*-wave. Based on the analytical solution in Graff (1979) and Zhao et al. (2014), the numerical results of peak pressure ratio are compared with analytical solution in Figure 6.17. It is observed that, the numerical results agree well with the analytical solution. Moreover, based on the analytical solution in Eq. (6.4), the error of *P*-wave velocity in this two-dimensional numerical example is only 3.3%.

Figure 6.17 Peak pressure ratio of numerical results and analytical solution

6.3.3 Half infinite plate

Due to the property of symmetry for stress wave propagation in half infinite plate, only the right half of the problem domain is simulated. The numerical model is illustrated in Figure 6.18, where the $10m \times 10m$ plate is discretized into 289 flat-top PU units, including 289 flat-top elements and 800 PU elements. The coordinate system is defined as follows: the origin is located at the lower left corner of the plate, the x and y axes are pointing rightward and upward, respectively. The left boundary is fixed in x direction, while the right boundary and lower boundary are applied with dashpots to prevent stress reflection. The plate is subjected to point loading $P = 3kN$ at the point $(0m, 10m)$ from the moment $t = 0s$ onwards. The density ρ , Young's modulus E and Poisson's ratio ν for this example are $2.65 \times 10^3 kg/m^3$, $66.25 GPa$ and 0.3 , respectively. In numerical simulation, the time step Δt is set as $0.00001s$, and the local approximation degree p is equal to 2. The Point B with coordinates $(0m, 5m)$ is selected as observation point, and the line AC with end nodes $(0m, 8m)$ and $(0m, 2m)$ is recognized as observation line.

Figure 6.18 Numerical model for stress wave propagation within half infinite plate

According to Fung (1965), when $x = 0m$, the analytical solution for stress σ_y can be expressed as:

$$\sigma_y = -\frac{Q}{2\pi} \mathbf{1}(t - v_1 r) \left[M_1(k_+^{(1)}) \frac{\partial k_+^{(1)}}{\partial t} - M_1(k_-^{(1)}) \frac{\partial k_-^{(1)}}{\partial t} \right] + \frac{2Q}{\pi} \mathbf{1}(t - v_2 r) \left[M_2(k_+^{(2)}) \frac{\partial k_+^{(2)}}{\partial t} - M_2(k_-^{(2)}) \frac{\partial k_-^{(2)}}{\partial t} \right] \quad (6.28)$$

where $Q = 2P = 6kN$ is the point loading applied in the whole problem domain; the parameter v_1 and v_2 can be calculated as:

$$v_1 = \frac{1}{V_p} \quad v_2 = \frac{1}{V_s} \quad (6.29)$$

r represents the distance from wave source $(0m, 10m)$; $\mathbf{1}(t)$ is the unit step function defined as:

$$\mathbf{1}(t) = \begin{cases} 0 & (t < 0) \\ 1 & (t \geq 0) \end{cases} \quad (6.30)$$

when $x = 0m$, $k_{\pm}^{(1)}$ and $k_{\pm}^{(2)}$ are calculated as:

$$k_{\pm}^{(1)} = \pm \sqrt{\frac{t^2}{r^2} - v_1^2} \quad k_{\pm}^{(2)} = \pm \sqrt{\frac{t^2}{r^2} - v_2^2} \quad (6.31)$$

and $M_1(k)$ and $M_2(k)$ are expressed as:

$$\begin{cases} M_1(k) = \frac{(2k^2 + v_2^2)^2}{(2k^2 + v_2^2)^2 - 4k^2 \sqrt{(k^2 + v_1^2)(k^2 + v_2^2)}} \\ M_2(k) = \frac{k^2 \sqrt{(k^2 + v_1^2)(k^2 + v_2^2)}}{(2k^2 + v_2^2)^2 - 4k^2 \sqrt{(k^2 + v_1^2)(k^2 + v_2^2)}} \end{cases} \quad (6.32)$$

From Eq. (6.4), it is easy to obtain that $V_P > V_S$, which means that $v_1 < v_2$ according to Eq. (6.29). Therefore, in Eq. (6.28), it can be observed that, for an arbitrary point D located over line $x = 0m$ and with distance r away from the wave source: 1) when $0 \leq t < v_1 r$, the stress $\sigma_y = 0kPa$; 2) when $t = v_1 r$, the leading front of P -wave arrives at point D and the stress $\sigma_y \rightarrow \infty$; 3) when $v_1 r < t < v_2 r$, the value of stress σ_y is a finite number, which is caused by the arrival of P -wave; 4) when $t = v_2 r$, the leading front of S -wave arrives at point D, meanwhile the value of stress σ_y increases to a finite number; 5) when $t > v_2 r$, the value of stress σ_y is a finite number, which is caused by the P -wave and S -wave together.

The numerical results for waveform at Point B within first $0.003s$ are compared with analytical solution in Figure 6.19. It is observed that the trend for stress σ_y in numerical results is similar with the analytical solution. Based on Eq. (6.28), it can be calculated that, when the P -wave reaches Point B at $t = 0.00086s$, the analytical solution increases from $0kPa$ to negative infinity suddenly; and it becomes to $-0.97kPa$ within $0.00001s$. However, it is impossible for the numerical method to demonstrate this singularity exactly. It can be observed that the numerical results gradually increase to $-1.14kPa$ from $t = 0.00082s$ to $t = 0.000105s$. After that, both the numerical results and analytical solution tend to decrease for a while and then increase, gradually, before the arrival of S -wave. In analytical solution, the S -wave reaches point B at $t = 0.00161s$, meanwhile the second peak pressure $\sigma_y = -1.24kPa$. While in the numerical simulation, the stress σ_y reaches its second peak pressure as $-1.13kPa$ when $t = 0.00149s$. Then, both the numerical results and analytical solution decrease slowly. In summary, for the numerical results in Figure 6.19, the

overall trend of stress σ_y , the arrival time of P -wave and S -wave and the value of second peak pressure agree well with the analytical solution. Although the negative infinity for peak pressure of P -wave cannot be reached by numerical simulation, the corresponding peak pressure in numerical results can be up to 17.5% higher than the stress σ_y obtained 0.00001s after the arrival of P -wave in analytical solution.

Figure 6.19 Waveform at Point B within first 0.003s

Figure 6.20 Attenuation of second peak pressure along the observation line AC

When the S -wave arrives, the stress σ_y reaches its second peak pressure. The attenuation of second peak pressure along the observation line AC in numerical results and analytical solution is compared in Figure 6.20. It is observed that the numerical results agree well with the analytical solution. Moreover, in general, the difference between them becomes smaller when the distance from wave source is longer. It is because that, according to Eq. (6.28), at the location of wave source, the

stress σ_y is negative infinite, which is a singular point. Therefore, when the observation point is located further away from the wave source, the numerical approximation will be more accurate.

Furthermore, the numerical results and analytical solution for the distribution of stress σ_y along the observation line AC when $t = 0.003s$ are compared in Figure 6.21. It can be observed that the numerical results agree well with the analytical solution. At this time, the second peak pressure has passed through all the points located over line AC. Therefore, through further comparison with Figure 6.20, it can be concluded that the difference between numerical results and analytical solution becomes smaller with the increase of time, especially for the points located closer to the wave source.

Figure 6.21 Distribution of stress σ_y along the observation line AC when $t = 0.003s$

6.4 Conclusions

In this chapter, the flat-top PU based high-order DDA method is extended to simulate stress wave propagation. Moreover, the non-reflective boundary is added to simulate infinite problem domain. The one- and two-dimensional numerical examples are simulated and analyzed in detail. Accordingly, several conclusions can be drawn for this chapter as follows:

1. In order to simulate the infinite problem domain, the improved formula for non-reflective boundary is modified and incorporated with flat-top PU mesh within the framework of high-order local approximation.

2. The numerical accuracy of stress wave propagation can be improved in three ways, which are increasing the number of flat-top PU units, improving local approximation degree p and decreasing time step Δt , respectively.
3. The instability of numerical results, which includes the fluctuation of stress before the arrival of wave and the unstable attenuation of peak pressure, can be reduced by improving the local approximation degree p .
4. The numerical accuracy is more sensitive to period T than velocity V_P ; because the former influences the accuracy of integration for both time and physical domain, while the latter only affects the accuracy of integration for physical domain.
5. Considering the requirement of wave length l_w with respect to the largest element length, the flat-top PU based high-order DDA method has a wider application scope than the suggestion in Kuhlemeyer and Lysmer (1973).
6. Since the influence of length ratio l_r is negligible, the construction flexibility and numerical stability of flat-top PU mesh can be ensured.
7. The flat-top PU based high-order DDA method is capable of simulating stress wave propagation with a highly-accurate velocity.
8. The numerical results for stress wave reflection and transmission at the interface of two different materials are accurate and stable.
9. The simulation of stress reflection under different boundary conditions, i.e. non-reflective boundary, fixed boundary and free boundary, is accurate and efficient.
10. The numerical results in simulating stress wave propagation with singular points are quite acceptable and reasonable.

CHAPTER 7 PRELIMINARY STUDY OF CRACK PROPAGATION WITH FLAT-TOP PU BASED HIGH-ORDER DDA

7.1 Introduction

It has been widely recognized that cracks play an important role in the mechanical behavior analysis. When a crack is initiated in the problem domain, the stress at the crack tip is infinite theoretically, and engineering failure occurs as the crack propagates (Hatzor et al., 1997; An, 2010). Therefore, in this chapter, the flat-top PU based high-order DDA method is extended to simulate crack propagation by establishing the crack initiation criteria and the crack propagation algorithm for flat-top PU mesh. Thus, with the crack propagation path provided by numerical simulation, reinforcement can be implemented in a more cost-efficient way.

The criteria for crack initiation can be mainly classified into two groups, which are the stress-based criteria and the energy-based criteria, respectively. For the stress-based criteria, there are the Mohr-Coulomb failure criterion (An, 2010), the triaxial failure criterion (Menetrey and Willam, 1995) and the maximum circumferential stress failure criterion (Carloni and Nobile, 2005) and so on. While for the energy-based criteria, there are the maximum strain energy release rate failure criterion (Shen and Stephansson, 1994), the strain energy density failure criterion (Kipp and Sih, 1975) and the energy-based failure criterion for peridynamic (Foster et al., 2011) and so on. The energy-based criteria are mainly limited in simulating problem with predefined cracks. Therefore, similar with the original DDA method, the stress-based criteria, i.e. the tension failure criterion and the Mohr-Coulomb failure criterion, are incorporated with the flat-top PU mesh to determine whether the critical values are reached, which is easy to understand and implement.

In this chapter, after a brief review of the crack propagation modes and the failure criteria, the crack initiation within flat-top PU mesh is introduced in Section 7.2. Then, the algorithm for crack propagation within flat-top PU mesh is discussed in detail in Section 7.3. After that, the numerical analysis procedure for crack propagation is summarized in Section 7.4. Several numerical examples are presented in Section 7.5. Finally, the conclusions for this chapter is addressed in Section 7.6.

7.2 Criteria for crack initiation

In this section, the three basic crack propagation modes are firstly reviewed, but only two of them need to be considered in the two-dimensional flat-top PU based high-order DDA method. Then, the tension failure criterion and the Mohr-Coulomb failure criterion are adopted to determine whether the critical values of tension stress and shear stress are reached. Based on the two basic modes and the failure criteria, the initiation of crack within flat-top PU mesh is introduced.

7.2.1 Crack propagation mode

In fracture mechanics, the crack can propagate under three different modes, which are Mode I: opening, Mode II: in-plane shear and Mode III: out-of-plane shear, respectively, as shown in Figure 7.1.

Figure 7.1 Three basic crack propagation modes

However, in the two-dimensional flat-top PU based high-order DDA method, only the first two modes, i.e. Mode I and Mode II, are considered due to the dimensional limitation. It is observed that the tension failure criterion and the Mohr-Coulomb failure criterion are perfectly suitable for determining crack initiation by providing tension stress inequality for Mode I and shear stress inequality for Mode II.

7.2.2 Failure Criteria

For the purpose of completeness, the tension failure criterion and the Mohr-Coulomb failure criterion are briefly reviewed here.

Tension failure criterion

For the Mode I crack, its initiation occurs when the following inequality is satisfied:

$$\sigma_n > \sigma_{n\max} \quad (7.1)$$

where σ_n represents the normal stress, which is positive for tension and negative for compression; and $\sigma_{n\max}$ is the tensile strength, which can be easily obtained through conventional uniaxial compression test.

Mohr-Coulomb failure criterion

As for the Mode II crack, its initiation occurs when the shear stress σ_t exceeds the shear strength $\sigma_{t\max}$. However, the value of shear strength $\sigma_{t\max}$ changes along with the normal stress σ_n , and the relationship between them can be depicted as in Figure 7.2. The intercept of the σ_t -axis c is the cohesion, and the slope of the failure envelop φ is the internal friction angle. Both parameters c and φ can be obtained in the shear test.

Figure 7.2 Sketch for Mohr-Coulomb failure criterion

Therefore, the Mode II crack initiates when the following inequality is satisfied:

$$\sigma_t > \sigma_{t\max} = c - \sigma_n \tan \varphi \quad (7.2)$$

7.2.3 Initiation of crack within flat-top PU mesh

In flat-top PU mesh, the normal stress and shear stress for each node are calculated along two sides. According to the construction of flat-top PU mesh in Chapters 3 and 4, it is observed that all the nodes in flat-top PU mesh are distributed as vertices of flat-top elements. Therefore, the calculation of normal stress and shear stress is carried out based on each flat-top element. As shown in Figure 7.3, a flat-top

element with four nodes P_i ($i = 1 \sim 4$) is taken as an example. The coordinates for node P_i are expressed as (x_i, y_i) . Moreover, it is stipulated that the coordinates (x_0, y_0) for P_0 and (x_5, y_5) for P_5 are equal to the coordinates for nodes P_4 and P_1 , respectively. Thus, the two sides, which share the common node P_i , can be referred as $P_{i-1}P_i$ and P_iP_{i+1} , respectively. Therefore, based on the stress state at node P_i , the normal stress and shear stress are calculated for both sides, i.e. $P_{i-1}P_i$ and P_iP_{i+1} .

Figure 7.3 A flat-top element with four nodes

For easy understanding, both sides for node P_i are referred as $P_{i1}P_{i2}$, where for line $P_{i-1}P_i$, $i_1 = i - 1$ and $i_2 = i$; while for line P_iP_{i+1} , $i_1 = i$ and $i_2 = i + 1$. As illustrated in Figure 7.4, the global coordinate system for the problem domain is set as (x, y) , while the local coordinate system for the red-colored line $P_{i1}P_{i2}$ is defined as (x', y') . Thus, the relationship between these two coordinate systems can be expressed as:

$$\begin{cases} x = x' \sin \alpha + y' \cos \alpha \\ y = -x' \cos \alpha + y' \sin \alpha \end{cases} \quad (7.3)$$

where α is the angle between x -axis and the line $P_{i1}P_{i2}$, thus $\sin \alpha$ and $\cos \alpha$ can be calculated as:

$$\begin{cases} \sin \alpha = \frac{y_{i2} - y_{i1}}{\sqrt{(x_{i2} - x_{i1})^2 + (y_{i2} - y_{i1})^2}} \\ \cos \alpha = \frac{x_{i2} - x_{i1}}{\sqrt{(x_{i2} - x_{i1})^2 + (y_{i2} - y_{i1})^2}} \end{cases} \quad (7.4)$$

Figure 7.4 Global and local coordinate systems

Therefore, the derivative of x and y with respect to x' and y' , respectively, can be obtained as:

$$\begin{cases} \frac{\partial x}{\partial x'} = \sin \alpha \\ \frac{\partial x}{\partial y'} = \cos \alpha \end{cases} \quad \begin{cases} \frac{\partial y}{\partial x'} = -\cos \alpha \\ \frac{\partial y}{\partial y'} = \sin \alpha \end{cases} \quad (7.5)$$

In linear elasticity, the normal strain ε_n and tangential strain ε_t for line $P_{i1}P_{i2}$ at node P_i are calculated as:

$$\begin{cases} \varepsilon_n = \frac{\partial U_{x'i}}{\partial x'} = \frac{\partial U_{x'i}}{\partial x} \cdot \frac{\partial x}{\partial x'} + \frac{\partial U_{x'i}}{\partial y} \cdot \frac{\partial y}{\partial x'} \\ \varepsilon_t = \frac{\partial U_{y'i}}{\partial y'} = \frac{\partial U_{y'i}}{\partial x} \cdot \frac{\partial x}{\partial y'} + \frac{\partial U_{y'i}}{\partial y} \cdot \frac{\partial y}{\partial y'} \end{cases} \quad (7.6)$$

where $(U_{x'i}, U_{y'i})$ are the global displacement at node P_i in the x' and y' directions, respectively, which can be calculated as:

$$\begin{cases} U_{x'i} = U_{xi} \sin \alpha - U_{yi} \cos \alpha \\ U_{y'i} = U_{xi} \cos \alpha + U_{yi} \sin \alpha \end{cases} \quad (7.7)$$

where (U_{xi}, U_{yi}) are the global displacement at node P_i in the x and y directions, respectively.

Substituting Eqs. (7.5) and (7.7) into Eq. (7.6), it can be obtained that:

$$\begin{cases} \varepsilon_n = \frac{\partial U_{xi}}{\partial x} \cdot \sin^2 \alpha + \frac{\partial U_{yi}}{\partial y} \cdot \cos^2 \alpha - \left(\frac{\partial U_{yi}}{\partial x} + \frac{\partial U_{xi}}{\partial y} \right) \cdot \sin \alpha \cos \alpha \\ \varepsilon_t = \frac{\partial U_{yi}}{\partial y} \cdot \sin^2 \alpha + \frac{\partial U_{xi}}{\partial x} \cdot \cos^2 \alpha + \left(\frac{\partial U_{yi}}{\partial x} + \frac{\partial U_{xi}}{\partial y} \right) \cdot \sin \alpha \cos \alpha \end{cases} \quad (7.8)$$

Furthermore, according to the constitutive equations, the normal stress σ_n for line $P_{i1}P_{i2}$ at node P_i can be derived as:

$$\sigma_n = a\varepsilon_n + b\varepsilon_t \quad (7.9)$$

where for plane-stress problem, the coefficients a and b are calculated as:

$$a = \frac{E}{1-\nu^2} \quad b = \frac{E\nu}{1-\nu^2} \quad (7.10)$$

while for plane-strain problem, the coefficients a and b are expressed as:

$$a = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \quad b = \frac{E\nu}{(1+\nu)(1-2\nu)} \quad (7.11)$$

Substituting Eq. (7.8) into Eq. (7.9), it can be obtained that:

$$\begin{aligned} \sigma_n = & \left(a \frac{\partial U_{xi}}{\partial x} + b \frac{\partial U_{yi}}{\partial y} \right) \sin^2 \alpha + \left(a \frac{\partial U_{yi}}{\partial y} + b \frac{\partial U_{xi}}{\partial x} \right) \cos^2 \alpha + \\ & (b-a) \left(\frac{\partial U_{yi}}{\partial x} + \frac{\partial U_{xi}}{\partial y} \right) \sin \alpha \cos \alpha \end{aligned} \quad (7.12)$$

Given the following conditions:

$$\begin{cases} \sigma_{xi} = a\varepsilon_{xi} + b\varepsilon_{yi} = a \frac{\partial U_{xi}}{\partial x} + b \frac{\partial U_{yi}}{\partial y} \\ \sigma_{yi} = a\varepsilon_{yi} + b\varepsilon_{xi} = a \frac{\partial U_{yi}}{\partial y} + b \frac{\partial U_{xi}}{\partial x} \\ \sigma_{xyi} = \frac{a-b}{2} \varepsilon_{xyi} = \frac{a-b}{2} \left(\frac{\partial U_{yi}}{\partial x} + \frac{\partial U_{xi}}{\partial y} \right) \end{cases} \quad (7.13)$$

Eq. (7.12) can be written as:

$$\sigma_n = \sigma_{xi} \sin^2 \alpha + \sigma_{yi} \cos^2 \alpha - 2\sigma_{xyi} \sin \alpha \cos \alpha \quad (7.14)$$

Substituting Eq. (7.14) into Eq. (7.1), if the inequality is true, a tension crack will propagate along the direction parallel to line $P_{i1}P_{i2}$ from node P_i .

The shear stress σ_t for line $P_{i1}P_{i2}$ at node P_i can be derived in the same way, and the result is given as:

$$\sigma_t = (\sigma_{xi} - \sigma_{yi}) \sin \alpha \cos \alpha + \sigma_{xyi} (\sin^2 \alpha - \cos^2 \alpha) \quad (7.15)$$

Substituting Eqs. (7.14) and (7.15) into Eq. (7.2), if the inequality is true, a shear crack will propagate along the direction parallel to line $P_{i1}P_{i2}$ from node P_i .

7.3 Algorithm for crack propagation

In order to avoid generating new flat-top PU units and artificially determining the initial values for newly-generated unknowns, it is assumed that the crack propagates along the sides of elements. In this section, the two basic cases for crack

propagation within flat-top PU mesh are firstly introduced, and the treatment of blue-colored PU elements is presented at the same time. Then, based on the different combinations of these two basic cases, the treatment for white-colored PU element is discussed in detail.

7.3.1 Two basic cases for crack propagation within flat-top PU mesh

Based on the definition of flat-top PU unit in Chapter 3, it is stipulated that, when a crack propagates along the direction parallel to line $P_{i1}P_{i2}$, the blue-colored PU element taking line $P_{i1}P_{i2}$ as one of its sides will be taken over by its left- or upper-side flat-top element, which are defined as the first basic case and second basic case, respectively. So that, the number of unknowns in global equilibrium equations can be kept the same before and after crack propagation occurs.

Figure 7.5 shows the first basic case when the blue-colored PU element is taken over by its left-side flat-top element. As illustrated in Figure 7.5 (a), if it is determined that the crack propagates along the direction parallel to line $P_{(j)2}P_{(j)3}$ in flat-top element e_j or line $P_{(k)4}P_{(k)1}$ in flat-top element e_k , the blue-colored PU element e_n will be taken over by its left-side flat-top element e_j , and the red line represents the crack propagation path. Figure 7.5 (b) demonstrates the elements generated after crack propagation, where the discontinuity between elements $e_{j'}$ and e_k is ensured.

Figure 7.5 The first basic case of crack propagation

(a) Before crack propagation

(b) After crack propagation

Figure 7.6 The second basic case of crack propagation

The second basic case is illustrated in Figure 7.6, where the blue-colored PU element is taken over by its upper-side flat-top element. As shown in Figure 7.6 (a), it is determined that the crack propagates along the direction parallel to line $P_{(j)1}P_{(j)2}$ in flat-top element e_j or line $P_{(k')3}P_{(k')4}$ in flat-top element $e_{k'}$. Under this condition, the blue-colored PU element e_n' is taken over by its upper-side flat-top element e_j , and the crack propagates along the red line. Thus, the elements after crack propagation can be depicted as in Figure 7.6 (b), where the discontinuity between elements $e_{j'}$ and $e_{k'}$ is ensured.

It is worth to note that the position relationship between the flat-top elements and the PU elements is recognized and reserved when constructing the flat-top PU mesh and will not change during simulation, even though the coordinates for nodes are updated at the end of each time step.

7.3.2 Different combinations of two basic cases

For the treatment of white-colored PU elements, it is determined by the combinations of the two basic case in Section 7.3.1. Before a white-colored PU element is linked to any crack, it is connected with four elemental supports. As illustrated in Figure 7.7, the nodes for the white-colored PU element are numbered from 1 to 4 in sequence, and the elemental support corresponding to the i th node is recognized as e_i . If the white-colored PU element is linked with crack paths, the number of its elemental supports may be decreased. Therefore, according to the number of elemental supports that the white-colored PU element is connected with, the treatment for these PU elements can be classified into four groups.

Figure 7.7 Elemental supports for a white-colored PU element not linked to any crack

Group 1

It is easy to observe that each white-colored PU element is linked to four lines that may be split into crack paths. Therefore, four different cases are included in Group 1, as illustrated in Figure 7.8. For all these four cases, the elemental support for the i th node of white-colored PU element is kept as flat-top element e_i or e_i' .

In each case of Group 1, two new potential penetration lines are generated, which are the two crack boundaries. Take the Case 1 in Figure 7.8 as an example, after the crack propagation, the lower side of element e_4' and the upper side of element e_1 are recognized as potential penetration lines. However, no potential penetration node is generated in these four cases.

(a) Case 1

(b) Case 2

Figure 7.8 Four cases in Group 1

Group 2

Three cases are included in Group 2, where the white-colored PU element is connected with three elemental supports and transferred into orange-colored PU element, as shown in Figure 7.9. In Case 1 and Case 3, the elemental supports for node 1 and node 3 are changed into flat-top element $e_{4'}$. In Case 2, the elemental support for node 2 is determined by the sequence the two crack paths generated. If the element $e_{3'}$ is generated before $e_{1'}$, the elemental support for node 2 is changed into element e_3 ; otherwise, it is changed into $e_{1'}$. The red nodes and lines in Figure 7.9 are the potential penetration nodes and potential penetration lines, respectively.

(c) Case 3

Figure 7.9 Three cases in Group 2

Group 3

Group 3 contains four cases, where the white-colored PU element is connected to two elemental supports, and it is changed into blue-colored PU element, correspondingly. As shown in Figure 7.10, in Case 1 and Case 2, the elemental supports for node 1 and node 2 are changed into flat-top elements $e_{4'}$ and $e_{3'}$, respectively; while in Case 3 and Case 4, the elemental supports for node 2 and node 3 are substituted by elements $e_{1'}$ and $e_{4'}$, respectively. The red nodes and red lines in each case in Figure 7.10 are the potential penetration nodes and potential penetration lines, respectively.

(a) Case 1

(b) Case 2

Figure 7.10 Four cases in Group 3

Group 4

The Group 4 is composed of four cases, where the white-colored PU element is connected with only one elemental supports, and thus it is taken over by the flat-top element e_4 in Figure 7.7. The potential penetration nodes and potential penetration lines generated in each case are marked as red nodes and red lines, respectively, in Figure 7.11.

Figure 7.11 Four cases in Group 4

7.4 Numerical analysis procedure

In summary, the simulation of crack propagation with flat-top PU based high-order DDA method is implemented as follows:

Step 1: The numerical model is generated through intersecting the flat-top PU mesh with physical boundary and initial crack of the problem domain, as introduced in Section 5.2.2.

Step 2: The loading is increased gradually. Within each time step, the normal and tangential stresses are calculated at each node and checked against the tensile strength and shear strength, respectively, as introduced in Section 7.2.

Step 3: The increase of loading will be stopped, when crack initiation is observed at a node P_i along the line $P_{i1}P_{i2}$ according to the criteria introduced in Section 7.2.

Step 4: With the assumption that the crack propagates along the sides of elements, the treatment of blue- and white-colored PU elements is implemented according the algorithm introduced in Section 7.3, which means that the increment of crack length within each time step is determined by the element size.

Step 5: With the newly generated numerical model according to the algorithm in Section 7.3, the normal and tangential stresses at each node are recalculated and checked against the tensile and shear strength, respectively. If Eq. (7.1) and/or Eq. (7.2) is satisfied, the crack will propagate along the line $P_{i1}P_{i2}$ with largest value of normal and/or tangential stress, and the simulation goes to Step 4. Otherwise, the loading will be increased gradually again, and the simulation goes to Step 2. The numerical analysis will be stopped when the crack intersects with the physical boundary.

7.5 Numerical examples

Based on the crack initiation criteria and the crack propagation algorithm introduced in Sections 7.2 and 7.3, several numerical examples are presented to verify the accuracy of flat-top PU based high-order DDA method in simulating crack propagation.

7.5.1 Spalling of long elastic bar

In this example, the long elastic bar in Section 6.3.1 is applied with a half cycle sinusoidal compressive P -wave at the left end, which can be expressed as:

$$F = \begin{cases} A \sin(\omega\pi t) & (0 \leq t \leq 1/\omega) \\ 0 & (t > 1/\omega) \end{cases} \quad (7.16)$$

where the amplitude $A = 80\text{MPa}$; ω is the frequency of the incident P -wave. The right end is designed as free boundary, as illustrated in Figure 7.12. The material density ρ , Young's modulus E , Poisson's ratio ν and tensile strength $\sigma_{\max}$ are $2.65 \times 10^3 \text{kg/m}^3$, 66.25GPa , 0.3 and 20MPa , respectively. In the numerical model, the elastic bar is discretized into 1000 flat-top PU units, the local approximation degree p is equal to 1, and the time step Δt is set as 0.00001s .

Figure 7.12 A long elastic bar subjected to compressive P -wave

Under the condition of free boundary at the right end, the reflected stress becomes tensile wave, and thus crack initiates when the tensile strength $\sigma_{\max}$ is reached. The numerical results for spalling of the elastic bar are presented in Figure 7.13, where the incident P -wave is designed with three different frequencies, i.e. 500Hz, 250Hz and 125Hz, respectively. Each green-colored part represents an intact block, while the red-colored parts are the spalling area with numbers of small blocks. It is observed that, when the frequency of incident wave decreases, the lengths for the right-side intact block and the spalling area increase, which is similar with the results obtained in Gu and Zhao (2009). It is because that, when the frequency decreases, the wave length increases. Therefore, 1) with the peak pressure kept the same, the reflected wave needs to travel a longer distance to reach the tensile strength $\sigma_{\max}$ after superposed with the incident wave, so the length for the right-side intact block increases; 2) the length for the area, where the value of reflected stress or superposed stress exceeds the tensile strength $\sigma_{\max}$, is also increased, so the length for the spalling area increases.

Moreover, it is obtained that, in the spalling area, the length for the small blocks ranges from $0.1m$ to $0.4m$, from $0.1m$ to $0.5m$ and from $0.1m$ to $0.6m$, when the frequency is equal to $500Hz$, $250Hz$ and $125Hz$, respectively. This is similar with the results obtained in Brara et al. (2001). It is because that, with the peak pressure kept as $80MPa$, the elastic energy is more concentrated when the frequency is higher. Correspondingly, the sizes of blocks in spalling area are smaller.

Figure 7.13 Spalling of elastic bar with different frequencies of incident P -wave

The numerical results for the velocity at the right end of the elastic bar within first $0.04s$ are presented in Figure 7.14, which includes three cases with different frequencies. It is observed that the velocity increases to its peak value quickly from the moment when the incident wave arrives at the right end, i.e. $t = 0.02s$. Then, after detaching from the elastic bar, the velocity fluctuates to a stable value gradually. In Figure 7.14, the duration for the velocity increasing to its peak value becomes shorter, when the frequency increases. This phenomenon is also caused by the length of incident wave. Moreover, the peak value for velocity tends to be lower, when the increasing duration is shorter. It can also be observed from Figure 7.14 that the duration for the velocity fluctuating to its stable value becomes longer, when the frequency decreases. It is because that, as shown in Figure 7.13, the length of the right-side intact block increases with the decrease of frequency. As a result, it takes a longer time to accomplish one wave reflection cycle within the right-side intact block. Correspondingly, the velocity takes a longer time to fluctuate to its stable value.

Figure 7.14 Velocity at the right end of the elastic bar

Figure 7.15 Velocity at the right end of the left-side intact block

For the left-side intact block, the numerical results for the velocity at its right end within first 0.04s are demonstrated in Figure 7.15. It is observed that the moment, at which the velocity starts to increase, becomes earlier, when the frequency decreases. The reason is that, as illustrated in Figure 7.13, the length for the left-side intact block decreases together with the frequency. As a result, it takes a shorter time for the incident wave to arrive at the right end of the left-side intact block. The duration for the velocity to increase and decrease is longer, when the wave length becomes longer. When the velocity decreases to a specific value, i.e. around 1.15m/s, the reflected tensile wave reaches the critical value σ_{nmax} and spalling occurs. Meanwhile, for the left-side intact block, its right end becomes the new free boundary. After a short fluctuation caused by the detachment, the velocity decreases to zero suddenly after spalling. Moreover, it can be also observed that the peak value for velocity increases with the decrease of frequency. This observation

coincides with the numerical results in Section 6.3.1, where the attenuation of peak pressure decreases with the increase of period T .

7.5.2 Square plate with internal crack

A $0.6m \times 0.6m$ square plate with an internal crack is subjected to uniaxial uniformly distributed loading q along the upper and lower boundary, as shown in Figure 7.16 (a). The center of the internal crack coincides with the center of the square plate, and the angle θ between the horizontal line and the crack is designed as 45° and 30° , respectively. In both cases, the horizontal length of the crack is kept as $0.2m$. The same flat-top PU mesh is utilized for each case as shown in Figure 7.16 (b), where the original regularly patterned flat-top elements, which are expected to contain the crack propagation path, are locally refined. After intersected by the internal crack, different numerical models are generated in different cases. In numerical simulation, the Young's modulus E , Poisson's ratio ν , tensile strength σ_{max} , cohesion c and internal friction angle φ are $50GPa$, 0.11 , $1.15MPa$, $2.5MPa$ and 30° , respectively; the local approximation degree $p = 2$; and the uniformly distributed loading q increases gradually until crack initiation occurs within the problem domain.

(a) A square plate with an internal crack subjected to uniformly distributed loading

(b) Flat-top PU mesh

Figure 7.16 Basic information for square plate and flat-top PU mesh

The numerical results for crack propagation path are illustrated as red lines in Figure 7.17 (a) for $\theta = 45^\circ$ and Figure 7.17 (b) for $\theta = 30^\circ$, respectively. It is observed that, under vertical uniaxial tension, the crack propagates horizontally without considering the value for angle θ , as stated in Pan and Huang (1986).

(a) Crack propagation path for $\theta = 45^\circ$

(b) Crack propagation path for $\theta = 30^\circ$

Figure 7.17 Crack propagation path under uniaxial loading with different angle θ

Figure 7.18 Numerical model for a square plate with an internal crack subjected to biaxial uniformly distributed loading

Next, a square plate with internal crack is subjected to biaxial uniformly distributed loading q as shown in Figure 7.18, where a new locally refined flat-top PU mesh is utilized to generate numerical model by intersecting with the internal crack. In this case, the angle θ is fixed as 45° . The material properties, local approximation

degree p and loading pattern for the problem domain are kept the same as in uniaxial loading.

Figure 7.19 Crack propagation path under biaxial uniformly distributed loading

Under the biaxial uniformly distributed loading q , theoretically, the crack will propagate as a straight line with the inclination equal to 45° , due to the property of symmetry. The numerical results are presented in Figure 7.19 as red lines, where the crack propagation path is approximated with horizontal and vertical segments. It is observed that the overall crack propagation tendency agrees well with the theoretical prediction.

7.5.3 Cantilever beam with edge crack

As illustrated in Figure 7.20 (a), a $6m \times 2m$ cantilever beam with a $2m$ edge crack is subjected to a pair of point loading. The right end of the cantilever beam is set as fixed boundary. If the edge crack is located over the horizontal centerline, based on the property of symmetry, it should propagate in the horizontal direction as a straight line. However, the corresponding crack propagation path is unstable (An, 2010). Therefore, in this example, the crack is designed $0.05m$ away from the centerline. The numerical model is generated as shown in Figure 7.20 (b), where the red line represents the edge crack, and the original regularly patterned flat-top elements, which are expected to contain the crack propagation path, are locally refined. The Young's modulus E , Poisson's ratio ν , tensile strength σ_{max} , cohesion c and internal friction angle φ are $10GPa$, 0.3 , $1.2MPa$, $2.0MPa$ and 30° ,

respectively. In numerical simulation, the local approximation degree $p = 2$; the point loading P increases gradually until crack initiation occurs within the problem domain.

(a) A cantilever beam with an edge crack subjected to a pair of point loading

(b) Numerical model for the cantilever beam

Figure 7.20 Basic information for cantilever beam and numerical model

The numerical results for crack propagation path are illustrated in Figure 7.21 as red lines. It is observed that the crack propagates upwards with a curvature, which is similar with the result in An (2010).

Figure 7.21 Crack propagation path

7.6 Conclusions

In this chapter, the flat-top PU based high-order DDA method is extended to simulate crack propagation with the crack initiation criteria and crack propagation algorithm. Three numerical examples are presented to verify the accuracy of this method in predicting the crack propagation path. Considering the preliminary study in this chapter, several conclusions can be addressed as follows:

1. According to the tension failure criterion and the Mohr-Coulomb failure criterion, the crack initiation is determined by the stress state of nodes in flat-top PU mesh.
2. Based on the two basic cases for crack propagation within flat-top PU mesh, the treatment of PU elements is straightforward to ensure no new unknowns are generated after crack propagation.
3. For the spalling of long elastic bar, since no analytical solution is found, the numerical results are compared with the observations in other research work, and they agree well with each other. Qualitative analysis is presented in detail, which verifies the numerical efficiency.
4. For the crack propagation within square plate, different crack inclination angles and loading conditions are studied. The numerical results agree well with the theoretical prediction.
5. Since no analytical solution is found for the propagation of edge crack in cantilever beam, the numerical results are compared with other research work, and well agreement is observed.
6. The local refinement of flat-top PU mesh for the area, which is expected to contain the crack propagation path, can increase the numerical efficiency.
7. In the simulation of crack propagation, similar with the study in other chapters, it is more accurate to discretize the problem domain into more flat-top PU units. However, the relationship between the unit size, numerical accuracy and cost-efficiency need to be determined by carrying out more studies in the future.
8. In current study, in order to avoid generating new unknowns and determining their initial values artificially, the crack is designed to propagate along the sides of elements. Therefore, the propagation path, which does not coincide

with the element sides, is approximated with a series of serrations, as shown in Sections 7.4.2 and 7.4.3. However, in the future, the simulation of crack propagating through elements is worth to be studied.

CHAPTER 8 CONCLUSIONS AND RECOMMENDATIONS

8.1 Conclusions

This research work mainly focuses on the development and application of flat-top PU based high-order DDA method.

The development work can be summarized as follows:

1. Based on the traditional PU mesh, a new construction method for the linearly independent and regularly patterned flat-top PU mesh is proposed for one- to three-dimensional spaces, which is easier to understand and straightforward to implement, compared with other construction methods. The establishment of global equilibrium equations for the flat-top PU mesh with high-order approximation is derived in detail for three-dimensional case, and the derivation for one- and two-dimensional cases can be easily obtained by eliminating the corresponding dimensions.
2. Based on the regularly patterned flat-top PU mesh, the construction of locally refined flat-top PU mesh is proposed, which is easy to implement by ensuring continuity between original and new elements. According to the equivalent p -version element and rank deficiency counting approach, the linear independence of locally refined flat-top PU mesh can be proven from element level. In this study, the construction and proof procedures are presented for one- and two-dimensional cases in detail. As for the three-dimensional case with hexahedral element, the construction and proof can be implemented in exactly the same way.
3. The flat-top PU based high-order approximation is coupled with the DDA method. So that, the problem with discontinuities can be simulated, where the continuous blocks are discretized with flat-top PU mesh and the interaction between blocks is simulated with the contact algorithm. Moreover, the contact algorithm in original DDA method is improved by searching the exact contact position. Therefore, the energy consumption is more accurate, especially for the collision simulation. In flat-top PU based high-order DDA method, since the flat-top PU mesh and high-order approximation are utilized instead of discretizing the problem domain into numbers of small blocks, only the main

discontinuities need to be considered. Such that, the time-consuming iteration of contact detection, adding and removing stiff springs can be reduced. During the generation of numerical model for discontinuous problem domain, in order to avoid ill-conditioned stiffness matrix in global equilibrium equations, modification for the numerical model is implemented based on the concept of percentage of effective area.

4. The improved formula for non-reflective boundary is coupled with flat-top PU based high-order DDA method to simulate stress wave propagation within infinite problem domain, where the numerical accuracy can be improved by deriving the analytical velocity of dashpots, instead of calculating the velocity with finite difference method.
5. Based on the two crack propagation modes in two-dimensional space, i.e. Mode I opening and Mode II in-plane shear, the tension failure criterion and the Mohr-Coulomb failure criterion are adopted to determine crack initiation in flat-top PU based high-order DDA method. Within each time step, the criteria are applied at every node in the flat-top PU mesh. In order to avoid generating new unknowns and determining their initial values artificially, it is assumed that the crack propagates along the element sides. Correspondingly, the treatment of PU elements after crack propagation is introduced based on the two basic crack propagation cases.

Based on the numerical examples presented in Chapters 3 to 7, the following conclusions can be drawn:

1. The influence of relative size between flat-top element and PU element is studied in different chapters with different problems, and it has been verified that the flat-top PU mesh possesses the advantages of construction flexibility and numerical stability.
2. The numerical accuracy of flat-top PU based high-order DDA method can be improved in three ways, namely by increasing the local approximation degree p , decreasing the size for flat-top PU unit, and reducing the time step Δt .
3. With the construction of flat-top PU mesh and employment of high-order approximation, the numerical results with high accuracy can be obtained for problems such as slender beam or pendulum with large aspect ratio, curved

and concave structure, non-polynomial solution, and stress wave with singularity under point loading.

4. With the construction of locally refined flat-top PU mesh, the cost-efficiency can be significantly improved for the problems such as stress concentration, stress singularity and crack propagation.
5. In the study of influence of wave length, the numerical accuracy is more sensitive to incident wave period T than velocity V_P ; because the former influences the accuracy of integration for both time and physical domain, while the latter only affects the accuracy of integration for physical domain.
6. Considering the requirement of wave length l_w with respect to the largest element length, the flat-top PU based high-order DDA method has a wider application scope than the suggestion in Kuhlemeyer and Lysmer (1973).
7. The flat-top PU based high-order DDA method is capable of simulating wave reflection and transmission at the interface of two different materials and under different boundary conditions.
8. The numerical results of crack propagation path obtained in flat-top PU based high-order DDA method agree well with the analytical prediction and the observations from other research work.

8.2 Recommendations for future work

Compared with other numerical methods, such as FEM, XFEM and DEM, the development and application of flat-top PU based high-order DDA method are still under preliminary stage. Much more work needs to be carried out in the future to extend the application scope. Based on the current study, the future work is recommended as follows:

1. The extension from two-dimensional quadrilateral mesh to three-dimensional hexahedral mesh is straightforward, but more challenging considering the added z -axis. The generation of numerical model and the crack propagation algorithm will be more complicated. The size of global equilibrium equations will be larger, and it will require a larger data storage space. The time for computing will also increase. But it is worth to be taken into consideration in the future in order to simulate the three-dimensional structures.

2. For the development of flat-top PU mesh, in order to establish a strict local refinement algorithm, quadrilateral element and hexahedral element are adopted for two- and three-dimensional cases, respectively. The development of triangular element and tetrahedral element is worth to be considered in the future. However, regarding to the cost-efficiency in problems like stress concentration, stress singularity and crack propagation, a more complicated local refinement algorithm needs to be developed, which is able to ensure the continuities between the original elements and newly generated elements.
3. In current study, the flat-top PU based high-order DDA has been applied in simulating stress wave propagation within continuous media. However, considering the existence of discontinuities in real projects, the application scope can be extended to the stress wave propagation within discontinuous problem domain.
4. For the simulation of crack propagation, in order to avoid generating new unknowns and determining their initial values artificially, the crack is designed to propagate along the sides of elements. Therefore, the propagation path, which does not coincide with the element sides, is approximated with a series of serrations. It will be more accurate if the crack is allowed to propagate through elements. Correspondingly, new flat-top PU units will be generated by splitting the original elements with crack. Meanwhile, the determination of the unknowns for newly generated flat-top PU units will be a challenging work, which is necessary for the calculation in next time step.
5. In current study, the geometrical nonlinearity is achieved by dividing the deformation into finite time steps, but the material nonlinearity is absent, which can be considered in the future by setting different constitutive models in different time steps.

REFERENCES

- Amiri, F., Anitescu, C., Arroyo, M., Bordas, S. P. A. and Rabczuk, T. (2014), “XLME interpolants, a seamless bridge between XFEM and enriched meshless methods”, Computational Mechanics, Vol. 53, pp. 45-57.
- An, X. M. (2010), “Extended numerical manifold method for engineering failure analysis”, Ph.D. Thesis, Nanyang Technological University, Singapore.
- An, X. M., Li, L. X., Ma, G. W. and Zhang, H. H. (2011), “Prediction of rank deficiency in partition of unity-based methods with plane triangular or quadrilateral meshes”, Computer Methods in Applied Mechanics and Engineering, Vol. 200, pp. 665-674.
- An, X. M., Liu, X. Y., Zhao, Z. Y. and He, L. (2014), “Proof of linear independence of flat-top PU-based high-order approximation”, Engineering Analysis with Boundary Elements, Vol. 44, pp. 104-111.
- An, X. M., Zhao, Z. Y., Zhang, H. H. and Li, L. X. (2012), “Investigation of linear dependence problem of three-dimensional partition of unity-based finite element methods”, Computer Methods in Applied Mechanics and Engineering, Vol. 233, pp. 137-151.
- Babuška, I. and Melenk, J. M. (1997), “Partition of unity method”, International Journal for Numerical Methods in Engineering, Vol. 40, pp. 727-758.
- Bao, H., Hatzor, Y. H. and Huang, X. (2012), “A new viscous boundary condition in the two-dimensional discontinuous deformation analysis method for wave propagation problems”. Rock Mechanics and Rock Engineering, Vol. 45, pp. 919-928.
- Bedford, A. and Drumheller, D. S. (1994), Introduction to elastic wave propagation, John Wiley & Sons: New Jersey.
- Beer, G. (1983), “Finite element, boundary element and coupled analysis of unbounded problems in elastostatics”, International Journal for Numerical Methods in Engineering, Vol. 19, pp. 567-580.

- Belytschko, T. and Black, T. (1999), "Elastic crack growth in finite elements with minimal re-meshing", International Journal for Numerical Methods in Engineering, Vol. 45, pp. 601-620.
- Belytschko, T. and Fleming, M. (1999), "Smoothing, enrichment and contact in the element-free Galerkin method", Computers and Structures, Vol. 71, pp. 173-195.
- Bergara, A., Dorado, J. I., Martin-Meizoso, A. and Martínez-Esnaola, J. M. (2017), "Fatigue crack propagation in complex stress fields: experiments and numerical simulations using the extended finite element method (XFEM)", International Journal of Fatigue, Vol. 103, pp. 112-121.
- Beyabanaki, S. A. R., Jafari, A. and Yeung, M. R. (2010), "High-order three-dimensional discontinuous deformation analysis (3-D DDA)", International Journal for Numerical Methods in Biomedical Engineering, Vol. 26, pp. 1522-1547.
- Beyabanaki, S. A. R., Jafari, A., Biabanaki, S. O. R. and Yeung, M. R. (2009), "Nodal-based three-dimensional discontinuous deformation analysis (3-D DDA)", Computers and Geotechnics, Vol. 36, pp. 359-372.
- Bieniawski, Z. T., Denkhaus, H. G. and Vogler, U. W. (1969), "Failure of fractured rock", International Journal of Rock Mechanics and Mining Sciences, Vol. 6, pp. 323-341.
- Blandford, G. E., Ingraffea, A. R. and Liggett, J. A. (1981), "Two-dimensional stress intensity factor computations using the boundary element method", International Journal for Numerical Methods in Engineering, Vol. 17, pp. 387-404.
- Bohlen, T. (2002), "Parallel 3-D viscoelastic finite difference seismic modelling", Computers and Geosciences, Vol. 28, pp. 887-899.
- Brady, B. H. G. and Wassyng, A. (1981), "A coupled finite element-boundary element method of stress analysis", International Journal of Rock Mechanics and Mining Sciences and Geomechanics Abstracts, Vol. 18, pp. 475-485.
- Brara, A., Camborde, F., Klepaczko, J. R. and Mariotti, C. (2001), "Experimental and numerical study of concrete at high strain rates in tension", Mechanics of Materials, Vol. 33, pp. 33-45.

- Brian, P. L. T. (1961), "A finite-difference method of high-order accuracy for the solution of three-dimensional transient heat conduction problems", AIChE Journal, Vol. 7, pp. 367-370.
- Brighi, B., Chipot, M. and Gut, E. (1998), "Finite differences on triangular grids", Numerical Methods for Partial Differential Equations, Vol. 14, pp. 567-579.
- Bushnell, D., Almroth, B. O. and Brogan, F. (1971), "Finite difference energy method for nonlinear shell analysis", Computer and Structures, Vol. 1, pp. 361-387.
- Caendish, J. C., Field, D. A. and Frey, W. H. (1985), "An approach to automatic three-dimensional finite element mesh generation", International Journal for Numerical Methods in Engineering, Vol. 21, pp. 329-347.
- Cai, J. G. and Zhao, J. (2000), "Effects of multiple parallel fractures on apparent attenuation of stress waves in rock masses", International Journal of Rock Mechanics and Mining Sciences, Vol. 37, pp. 661-682.
- Cai, M. and Liu, D. (2009), "Study of failure mechanisms of rock under compressive-shear loading using real-time laser holography", International Journal of Rock Mechanics and Mining Sciences, Vol. 46, pp. 59-68.
- Cai, Y., Zhu, H. and Zhuang, X. (2013), "A continuous/discontinuous deformation analysis (CDDA) method based on deformable blocks for fracture modeling", Frontiers of Structural and Civil Engineering, Vol. 7, pp. 369-378.
- Cai, Y. C., Zhuang, X. Y. and Augarde, C. (2010), "A new partition of unity finite element free from the linear dependence problem and possessing the delta property", Computer Methods in Applied Mechanics and Engineering, Vol. 199, pp. 1036-1043.
- Carlioni, C. and Nobile, L. (2005), "Maximum circumferential stress criterion applied to orthotropic materials", Fatigue and Fracture of Engineering Materials and Structures, Vol. 28, pp. 825-833.
- Casulli, V. (1999), "A semi-implicit finite difference method for non-hydrostatic, free-surface flows", International Journal for Numerical Methods in Fluids, Vol. 30, pp. 425-440.

- Cavendish, J. C. (1974), "Automatic triangulation of arbitrary planar domains for the finite element method", International Journal for Numerical Methods in Engineering, Vol. 8, pp. 679-696.
- Chang, Q. T. (1994), "Nonlinear dynamic discontinuous deformation analysis with finite element meshed block systems", PhD thesis, University of California, Berkeley.
- Chen, G. Q., Ohnishi, Y. and Ito, T. (1998), "Development of high-order manifold method", International Journal for Numerical Methods in Engineering, Vol. 43, pp. 685-712.
- Chen, Y. M. (1975), "Numerical computation of dynamic stress intensity factors by a Lagrangian finite-difference method (the HEMP code)", Engineering Fracture Mechanics, Vol. 7, pp. 653-660.
- Cheng, Y. M., Zhang, Y. H. and Chen, W. S. (2002), "Wilson non-confirming element in numerical manifold method", Communications in Numerical Methods in Engineering, Vol. 18, pp. 877-884.
- Chiou, Y. J., Lee, Y. M. and Tsay, R. J. (2002), "Mixed fracture propagation by manifold method", International Journal of Fracture, Vol. 114, pp. 327-347.
- Christian, J. T. and Boehmer, J. W. (1970), "Plane strain consolidation by finite elements", Journal of Soil Mechanics and Foundations Div, Vol. 96, pp. 1435-1457.
- Chryssanthakis, P., Barton, N., Lorig, L. and Christanson, M. (1997), "Numerical simulation of fiber reinforced shotcrete in a tunnel using the discrete element method", International Journal of Rock Mechanics and Mining Sciences, Vol. 34, pp. 54.e1-54.e14.
- Clayton, R. and Engquist, B. (1977), "Absorbing boundary conditions for acoustic and elastic wave equations", Bulletin of the Seismological Society of America, Vol. 67, pp. 1529-1540.
- Coates, R. T. and Schoenberg, M. (1995), "Finite-difference modeling of faults and fractures", Geophysics, Vol. 60, pp. 1514-1526.

-
- Collino, F. and Tsogka, C. (2001), “Application of the perfectly matched absorbing layer model to the linear elastodynamic problem in anisotropic heterogeneous media”, Geophysics, Vol. 66, pp. 294-307.
- Cook, N. G. W. (1970), “An experiment proving that dilatancy is a pervasive volumetric property of brittle rock loaded to failure”, Rock Mechanics, Vol. 2, pp. 181-188.
- Cook, R. D. (1974), “Improved two-dimensional finite element”, Journal of the Structural Division-ASCE, Vol. 100, pp. 1851-1863.
- Cook, R. D., Malkus, D. S. and Plesha, M. E. (1989), Concepts and applications of finite element analysis, 3rd ed., John Wiley: New York.
- Courant, R., Isaacson, E. and Rees, M. (1952), “On the solution of nonlinear hyperbolic differential equations by finite differences”, Communications on Pure and Applied Mathematics, Vol. 5, pp. 243-255.
- Crouch, S. L. and Tian, Y. (1988), “A two-dimensional direct boundary integral method for elastodynamics”, International Journal of Rock Mechanics and Mining Sciences and Geomechanics, Vol. 25, pp. 149-158.
- Cui, M. (2009), “Compact finite difference method for the fractional diffusion equation”, Journal of Computational Physics, Vol. 228, pp. 7792-7804.
- Cundall, P. A. (1971), “A computer model for simulating progressive, large scale movements in blocky rock systems”, Proceedings of the International Symposium Rock Fracture, ISRM, Nancy, Vol. 1, pp. II-8.
- Cundall, P. A. (1980), “UDEC – a generalized distinct element program for modelling jointed rock”, Report PCAR-1-80, Peter Cundall Associates, European Research Office, US Army Corps of Engineers.
- Cundall, P. A. (1988), “Formulation of a three-dimensional distinct element model – part I: a scheme to detect and represent contacts in a system composed of many polyhedral blocks”, International Journal of Rock Mechanics and Mining Sciences, Vol. 25, pp. 107-116.

- Dai, N., Vafidis, A. and Kanasewich, E. R. (1995), “Wave propagation in heterogeneous, porous media: a velocity-stress finite-difference method”, Geophysics, Vol. 60, pp. 327-340.
- Dieterich, J. H. (1979), “Modeling of rock friction: 1. Experimental results and constitutive equations”, Journal of Geophysical Research, Vol. 84, pp. 2161-2168.
- Dolbow, J., Moës, N. and Belytschko, T. (2000), “Discontinuous enrichment in finite elements with a partition of unity method”, Finite Elements in Analysis and Design, Vol. 36, pp. 235-260.
- Donea, J., Giuliani, S. and Halleux, J. P. (1982), “An arbitrary Lagrangian-Eulerian finite element method for transient dynamic fluid-structure interaction”, Computer Methods in Applied Mechanics and Engineering, Vol. 33, pp. 689-723.
- Duarte, C. A., Babuška, I. and Oden, J. T. (2000), “Generalized finite element methods for three-dimensional structural mechanics problems”, Computer and structure, Vol. 77, pp. 215-232.
- Duarte, C. A. and Oden, J. T. (1996), “H-p clouds—an h-p meshless method”, Numerical methods for partial differential equations, Vol. 12, pp. 673-706.
- Dvorkin, E. N., Cuitiño, A. M. and Gioia, G. (1990), “Finite elements with displacement interpolated embedded localization lines insensitive to mesh size and distortions”, International Journal for Numerical Methods in Engineering, Vol. 30, pp. 541-564.
- Fan, L. F., Yi, X. W. and Ma, G. W. (2013), “Numerical manifold method (NMM) simulation of stress wave propagation through fractured rock mass”, International Journal of Applied Mechanics, Vol. 5, pp. 1350022.
- Foster, J. T., Silling, S. A. and Chen, W. (2011), “An energy based failure criterion for use with peridynamic states”, International Journal for Multiscale Computational Engineering, Vol. 9, pp. 675-688.
- Fung, Y. C. (1965), Foundations of solid mechanics, Prentice-Hall, Inc., Englewood Cliffs, New Jersey.

- Gentier, S., Riss, J., Archambault, G., Flamand, R. and Hopkins, D. (2000), "Influence of fracture geometry on shear behavior", International Journal of Rock Mechanics and Mining Sciences, Vol. 37, pp. 161-174.
- Ghorashi, S. S., Valizadeh, N., Mohammadi, S. and Rabczuk, T. (2015), "T-spline based XIGA for fracture analysis of orthotropic media", Computers and Structures, Vol. 147, pp. 138-146.
- Goodman, R. E., Taylor, R. L. and Brekke, T. L. (1968), "A model for the mechanics of jointed rock", ASCE Journal of Soil Mechanics, Vol. 99, pp. 637-659.
- Graff, K. F. (1975), Wave Motion in Elastic Solids, Ohio State University Press, Columbus.
- Grayeli, R. and Mortazavi, A. (2006), "Discontinuous deformation analysis with second-order finite element meshed block", International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 30, pp. 1545-1561.
- Griffiths, D. V. and Lane, P. A. (1999), "Slope stability analysis by finite elements", Géotechnique, Vol. 49, pp. 387-403.
- Grigoriev, M. M. and Dargush, G. F. (1999), "A poly-region boundary element method for incompressible viscous fluid flow", International Journal for Numerical Methods in Engineering, Vol. 46, pp. 1127-1158.
- Grilli, S. T., Skourup, J. and Svendsen, I. A. (1989), "An efficient boundary element method for nonlinear water waves", Engineering Analysis with Boundary Elements, Vol. 6, pp. 97-107.
- Gu, J. and Zhao, Z. Y. (2009), "Consideration of the discontinuous deformation analysis on wave propagation problems", International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 33, pp. 1449-1465.
- Guo, Y. B., Wen, Q. and Horstemeyer, M. F. (2005), "An internal state variable plasticity-based approach to determine dynamic loading history effects on material property in manufacturing processes", International Journal of Mechanical Sciences, Vol. 47, pp. 1423-1441.

Harper, T. R. and Last, N. C. (1989), “Interpretation by numerical modeling of changes of fracture system hydraulic conductivity induced by fluid injection”, Géotechnique, Vol. 39, pp. 1-11.

Hart, R. D., Cundall, P. A. and Lemos, J. V. (1988), “Formulation of a three-dimensional distinct element method – part II: mechanical calculations for motion and interaction of a system composed of many polyhedral blocks”, International Journal of Rock Mechanics and Mining Sciences, Vol. 25, pp. 117-125.

Hatzor, Y. H., Zur, A. and Mimran, Y. (1997), “Microstructure effects on microcracking and brittle failure of dolomites”, Tectonophysics, Vol. 281, pp. 141-161.

He, L., An, X. M., Liu, X. Y., Zhao, Z. Y. and Yang, S. Q. (2015). “Augmented Numerical Manifold Method with implementation of flat-top partition of unity”, Engineering Analysis with Boundary Elements, Vol. 61, pp. 153-171.

Hong, W. T. and Lee, P. S. (2013a), “Coupling flat-top partition of unity method and finite element method”, Finite Elements in Analysis and Design, Vol. 67, pp. 43-55.

Hong, W. T. and Lee, P. S. (2013b), “Mesh based construction of flat-top partition of unity functions”, Applied Mathematics and Computation, Vol. 219, pp. 8687-8704.

Hsiung, S. M. (2001), “Discontinuous deformation analysis (DDA) with n-th order polynomial displacement functions”, In: Elworth D, Tinucci JP, Heasley KA, editors, Rock mechanics in the national interest: Swets and Zeitlinger Lisse, pp. 1437-1444.

Hughes, T. J. R., Franca, L. P. and Hulbert, G. M. (1989), “A new finite element formulation for computational fluid dynamics: VIII. the galerkin/least-squares method for advective-diffusive equations, “ Computer Methods in Applied Mechanics and Engineering, Vol. 73, pp. 173-189.

Jiang, Q. H., Deng, S. S., Zhou, C. B. and Lu, W. B. (2010), “Modeling unconfined seepage flow using three-dimensional numerical manifold method”, Journal of Hydrodynamics, Vol. 22, pp. 554-561.

- Jiang, Q. H. and Yeung, M. R. (2004), "A model of point-to-face contact for three dimensional discontinuous deformation analysis", Rock Mechanics and Rock Engineering, Vol. 37, pp. 95-116.
- Jiang, Q. H., Zhou, C. B. and Li, D. Q. (2009), "A three-dimensional numerical manifold method based on tetrahedral meshes", Computers and Structures, Vol. 87, pp. 880-889.
- Jiang, Q. H., Zhou, C. B., Luo, X. Q. and Zheng, H. (2006), "Coupling method of three-dimensional DDA-FEM and its application", Chinese Journal of Geotechnical Engineering, Vol. 28, pp. 998-1001.
- Jing, L. (2003), "A review of techniques, advances and outstanding issues in numerical modelling for rock mechanics and rock engineering", International Journal of Rock Mechanics and Mining Sciences, Vol. 40, pp. 283-353.
- Jing, L., Ma, Y. and Fang Z. (2001), "Modeling of fluid flow and solid deformation for fractured rocks with discontinuous deformation analysis (DDA) method", International Journal of Rock Mechanics and Mining Sciences, Vol. 38, pp. 343-355.
- Johnson, C. and Hansbo, P. (1992), "Adaptive finite element methods in computational mechanics", Computer Methods in Applied Mechanics and Engineering, Vol. 101, pp. 143-181.
- Ke, T. (1996), "Artificial joint-based DDA", Proceedings of the first International Forum on Discontinuous Deformation Analysis (DDA) and Simulations of Discontinuous Media, Berkeley, California, USA, pp. 326-333.
- Kim, Y., Amadei, B. and Pan, E. (1999), "Modeling the effect of water, excavation sequence and rock reinforcement with discontinuous deformation analysis", International Journal of Rock Mechanics and Mining Sciences, Vol. 36, pp. 949-970.
- Kipp, M. E. and Sih, G. C. (1975), "The strain energy density failure criterion applied to notched elastic solids", International Journal of Solids and Structures, Vol. 11, pp. 153-173.

- Kuhlemeyer, R. L. and Lysmer, J. (1973), "Finite element method accuracy for wave propagation problems", Journal of Soil Mechanics and Foundations Div, Vol. 99, pp. 421-427.
- Kurowski, P. M. (2016), Engineering analysis with SOLIDWORKS simulation 2016, SDC Publications, Mission.
- Kutter, H. K. and Fairhurst, C. (1971), "On the fracture process in blasting", International Journal of Rock Mechanics and Mining Sciences, Vol. 8, pp. 181-202.
- Lafhaj, Z. and Shahrour, I. (2000), "Use of the boundary element method for the analysis of permeability tests in boreholes", Engineering Analysis with Boundary Elements, Vol. 24, pp. 695-698.
- Li, H. B., Han, G. M. and Mang, H. A. (1985), "A new method for evaluating singular integrals in stress analysis of solids by the direct boundary element method", International Journal for Numerical Methods in Engineering, Vol. 21, pp. 2071-2098.
- Li, Q. Y., Wang, N. C. and Yi, D. Y. (2001), Numerical analysis, Tsinghua University Press, Beijing. (In Chinese)
- Liao, S. J. (1997), "Boundary element method for general nonlinear differential operators", Engineering Analysis with Boundary Elements, Vol. 20, pp. 91-99.
- Lin, S. Z. (2005), "Recursive formula for simplex integration", Journal of Yangtze River Scientific Research Institute, Vol. 22, pp. 32-34.
- Lindholm, D. A. (1983), "Automatic triangular mesh generation on surfaces of polyhedral", IEEE Transactions on Magnetic, Vol. 19, pp. 2539-2542.
- Liszka, T. and Orkisz, J. (1980), "The finite difference method at arbitrary irregular grids and its application in applied mechanics", Computers and Structures, Vol. 11, pp. 83-95.
- Liu, G. R., Nguyen, T. T., Nguyen, X. H. and Lam, K. Y. (2009), "A node-based smoothed finite element method (NS-FEM) for upper bound solutions to solid mechanics problems", Computer and Structures, Vol. 87, pp. 14-26.

- Liu, L., Wang, G. M., Chen, J. H. and Yang S. (2013), “Creep experiment and rheological model of deep saturated rock”, Transactions of Nonferrous Metals Society of China, Vol. 23, pp. 478-483.
- Liu, X. Y. (2013), “A DDA based complete and high order polynomial displacement approximation method in elastic mechanics and its cases verification”, Master thesis, Changjiang River Scientific Research Institute, Wuhan, China. (In Chinese)
- Lo, S. H. (1985), “A new mesh generation scheme for arbitrary planar domains”, International Journal for Numerical Methods in Engineering, Vol. 21, pp. 1403-1426.
- Lysmer, J. and Kuhlemeyer, R. L. (1969), “Finite dynamic model for infinite media”, Journal of the Engineering Mechanics Division, Vol. 95, pp. 859-877.
- Ma, G. W., An, X. M., Zhang, H. H. and Li, L. X. (2009), “Modeling complex crack problems using the numerical manifold method”, International Journal of Fracture, Vol. 156, pp. 21-35.
- MacNeal, R. H. and Harder, R. L. (1985), “A proposed standard set of problems to test finite element accuracy”, Finite Elements in Analysis and Design, Vol. 1, pp. 3-20.
- Meguro, K. and Hakuno, M. (1989), “Fracture analyses of concrete structures by the modified distinct element method”, Structural Engineering/Earthquake Engineering, Vol. 6, pp. 283-294.
- Melenk, J. M. and Babuška, I. (1996), “The partition of unity finite element method: basic theory and applications”, Computer Methods in Applied Mechanics and Engineering, Vol. 139, pp. 289-314.
- Menetrey, P. and Willam, K. J. (1995), “Triaxial failure criterion for concrete and its generalization”, ACI Structural Journal, Vol. 92, pp. 311-318.
- Moës, N., Dolbow, J. and Belytschko, T. (1999), “A finite element method for crack growth without remeshing”, International Journal for Numerical Methods in Engineering, Vol. 46, pp. 131-150.

- Morinishi, Y., Lund, T. S., Vasilyev, O. V. and Moin, P. (1998), "Fully conservative higher order finite difference schemes for incompressible flow", Journal of Computational Physics, Vol. 143, pp. 90-124.
- Nakayama, T. and Washizu, K. (1981), "The boundary element method applied to the analysis of two-dimensional nonlinear sloshing problems", International Journal for Numerical Methods in Engineering, Vol. 17, pp. 1631-1646.
- Narasimhan, T. N. and Witherspoon, P. A. (1976), "An integrated finite difference method for analyzing fluid flow in porous media", Water Resources Research, Vol. 12, pp. 57-64.
- Ning, Y. J., An, X. M. and Ma, G. W. (2011), "Footwall slope stability analysis with the numerical manifold method", International Journal of Rock Mechanics and Mining Sciences, Vol. 48, pp. 964-975.
- Ning, Y. J., Yang, J., An, X. M. and Ma, G. W. (2011a), "Modelling rock fracturing and blast-induced rock mass failure via advanced discretization within the discontinuous deformation analysis framework", Computers and Geotechnics, Vol. 38, pp. 40-49.
- Ning, Y. J., Yang, J., Ma, G. W. and Chen, P. W. (2011b), "Modelling rock blasting considering explosion gas penetration using discontinuous deformation analysis", Rock Mechanics and Rock Engineering, Vol. 44, pp. 483-490.
- Oden, J. T., Duarte, C. A. M. and Zienkiewicz, O. C. (1998), "A new cloud-based hp finite element method", Computer Methods in Applied Mechanics and Engineering, Vol. 153, pp. 117-126.
- Onate, E., Idelsohn, S. R., Celigueta, M. A. and Rossi, R. (2008), "Advances in the particle finite element method for the analysis of fluid-multibody interaction and bed erosion in free surface flows", Computer Methods in Applied Mechanics and Engineering, Vol. 197, pp. 1777-1800.
- Ong, K. S. (1974), "A finite-difference method to evaluate the thermal performance of a solar water heater", Solar Energy, Vol. 16, pp. 137-147.

- Pan, J. and Huang, K. (1986), "Further studies of mixed mode crack propagation in Pressure Vessels", Symposium of Fatigue and Fracture Assessment by Analysis and Testing, ASME PVP, Vol. 103, pp. 59-66.
- Papantonopoulos, C., Psycharis, I. N., Papastamatiou, D. Y., Lemos, J. V. and Mouzakis, H. P. (2002), "Numerical prediction of the earthquake response of classical columns using the distinct element method", Earthquake Engineering and Structural Dynamics, Vol. 31, pp. 1699-1717.
- Perrone, N. and Kao, R. (1975), "A general finite difference method for arbitrary meshes", Computer and Structures, Vol. 5, pp. 45-58.
- Pietruszczak, S. and Mróz, Z. (1981), "Finite element analysis of deformation of strain-softening materials", International Journal for Numerical Methods in Engineering, Vol. 17, pp. 327-334.
- Pitarka, A. (1999), "3D elastic finite-difference modeling of seismic motion using staggered grids with nonuniform spacing", Bulletin of the Seismological Society of America, Vol. 89, pp. 54-68.
- Popovics, J. S., Song, W. J., Ghandehari, M., Subramaniam, K. V., Achenbach, J. D. and Shah, S. P. (2000), "Application of surface wave transmission measurements for crack depth determination in concrete", ACI Structural Journal, Vol. 9, pp. 127-135.
- Portela, A., Aliabadi, M. H. and Rooke, D. P. (1992), "The dual boundary element method: effective implementation for crack problems", International Journal for Numerical Methods in Engineering, Vol. 33, pp. 1269-1287.
- Portela, A., Aliabadi, M. H. and Rooke, D. P. (1993), "Dual boundary element method incremental analysis of crack propagation", Computer and Structures, Vol. 46, pp. 237-247.
- Rabczuk, T. (2012), "Computational methods for fracture in brittle and quasi-brittle solids: state-of-the-art review and future perspectives", ISRN Applied Mathematics, Vol. 2013, pp. 1-38.
- Rabczuk, T. and Zi, G. (2007), "A meshfree method based on the local partition of unity for cohesive cracks," Computational Mechanics, Vol. 39, pp. 743-760.

- Raynaud, M. and Bransier, J. (1986), "A new finite-difference method for the nonlinear inverse heat conduction problem", Numerical Heat Transfer, Vol. 9, pp. 27-42.
- Riker, C. and Holzer, S. M. (2009), "The mixed-cell-complex partition-of-unity method", Computer Methods in Applied Mechanics and Engineering, Vol. 198, pp. 1235-1248.
- Schweitzer, M. A. (2012), "Generalizations of the finite element method", Central European Journal of Mathematics, Vol. 10, pp. 3-24.
- Shen, B. and Stephansson, O. (1994), "Modification of the G-criterion for crack propagation subjected to compression", Engineering Fracture Mechanics, Vol. 47, pp. 177-189.
- Shi, G. H. (1988), "Discontinuous deformation analysis: a new numerical method for the statics and dynamics of block systems", PhD Thesis, University of California, Berkeley.
- Shi, G. H. (1995), "Numerical manifold method", Proceedings of the First International Conferences on Analysis of Discontinuous Deformation (ICADD-1), Chungli, Taiwan, pp. 187-222.
- Shi, G. H. (1996), "Simplex integration for manifold method, FEM, DDA and analytical analysis", Proceeding of first International Forum on Discontinuous Deformation Analysis (DDA) and Simulations of Discontinuous Media, TSI Press, Albuquerque, NM, Mexico, pp. 205-262.
- Shi, G. H. (2001), "Three dimensional discontinuous deformation analysis", The 38th US Symposium on Rock Mechanics (USRMS), American Rock Mechanics Association.
- Shi, G. H. and Goodman, R. E. (1985), "Two dimensional discontinuous deformation analysis", International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 9, pp. 541-556.
- Shyu, K. (1993), "Nodal-based discontinuous deformation analysis", PhD thesis, University of California, Berkeley.

Stead, D., Eberhardt, E. and Coggan, J. S. (2006), “Developments in the characterization of complex rock slope deformation and failure using numerical modelling techniques”, Engineering Geology, Vol. 83, pp. 217-235.

Strouboulis, T., Babuška, I. and Copps, K. (2000), “The design and analysis of the generalized finite element method”, Computer Methods in Applied Mechanics and Engineering, Vol. 181, pp. 43-69.

Strouboulis, T., Copps, K. and Babuška, I. (2001), “The generalized finite element method”, Computer Methods in Applied Mechanics and Engineering, Vol. 190, pp. 4081-4193.

Sukumar, N. (2003), “Voronoi cell finite difference method for the diffusion operator on arbitrary unstructured grids”, International Journal for Numerical Methods in Engineering, Vol. 57, pp. 1-34.

Sukumar, N., Moës, N., Moran, B. and Belytschko, T. (2000), “Extended finite element method for three-dimensional crack modelling”, International Journal for Numerical Methods in Engineering, Vol. 48, pp. 1549-1570.

Sutradhar, A. and Paulino G. H. (2004), “A simple boundary element method for problems of potential in non-homogeneous media”, International Journal for Numerical Methods in Engineering, Vol. 60, pp. 2203-2230.

Tal, Y., Hatzor, Y. H. and Feng, X. T. (2014), “An improved numerical manifold method for simulation of sequential excavation in fractured rocks”, International Journal of Rock Mechanics and Mining Sciences, Vol. 65, pp. 116-128.

Taylor, L. M., Chen, E. P. and Kuszmaul, J. S. (1986), “Microcrack-induced damage accumulation in brittle rock under dynamic loading”, Computer Methods in Applied Mechanics and Engineering, Vol. 55, pp. 301-320.

Tian, R., Mastubara, H. and Yagawa, G. (2006), “Advanced 4-node tetrahedrons”, International Journal for Numerical Methods in Engineering, Vol. 68, pp. 1209-1231.

Tian, R. and Yagawa, G. (2005), “Generalized nodes and high-performance elements”, International Journal for Numerical Methods in Engineering, Vol. 64, pp. 2039-2071.

- Tian, R., Yagawa, G. and Terasaka, H. (2006), "Linear dependence problems of partition of unity-based generalized FEMs", Computer Methods in Applied Mechanics and Engineering, Vol. 195, pp. 4768-4782.
- Timoshenko, S. and Goodier, J. N. (1951), Theory of Elasticity, McGraw-Hill, Inc., New York.
- Tsesarsky, M. and Hatzor, Y. H. (2006), "Tunnel roof deflection in blocky rock masses as a function of joint spacing and friction – a parametric study using discontinuous deformation analysis (DDA)", Tunnelling and Underground Space Technology, Vol. 21, pp. 29-45.
- Tsesarsky, M., Hatzor, Y. H. and Sitar, N. (2005), "Dynamic displacement of a block on an inclined plane: analytical, experimental and DDA results", Rock Mechanics and Rock Engineering, Vol. 38, pp. 153-167.
- Veletsos, A. S. and Meek, J. W. (1974), "Dynamic behavior of building-foundation systems", Earthquake Engineering and Structural Dynamics, Vol. 3, pp. 121-138.
- Virieux, J. (1984), "SH-wave propagation in heterogeneous media: velocity-stress finite-difference method", Geophysics, Vol. 49, pp. 1933-1942.
- Virieux, J. (1986), "P-SV wave propagation in heterogeneous media: velocity-stress finite-difference method", Geophysics, Vol. 51, pp. 889-901.
- White, W., Vallipan, S. and Lee, I. K. (1977), "Unified boundary for finite dynamic models", Journal of the Engineering Mechanics Division, Vol. 103, pp. 949-964.
- Wu, A. Q., Ding, X. L., Li, H. Z., Chen, S. H. and Shi, G. H. (2006), "Numerical simulation of startup and whole failure process of Qianjiangping landslide using discontinuous deformation analysis method", Chinese Journal of Rock Mechanics and Engineering, Vol. 25, pp. 1297-1303. (In Chinese)
- Wu, A. Q., Liu, X. Y., Zhang, Y., Lu, B. and Lin, S. Z. (2014), "A DDA based complete and high order polynomial displacement approximation method in elastic mechanics and its cases verification", Chinese Journal of Solid Mechanics, Vol. 35, pp. 142-149. (In Chinese)

- Wu, J. H., Ohnishi, Y. and Nishiyama, S. (2004), "Simulation of the mechanical behavior of inclined jointed rock mass during tunnel construction using Discontinuous Deformation Analysis (DDA)", International Journal of Rock Mechanics and Mining Sciences, Vol. 41, pp. 731-743.
- Yamada, Y. and Yoshimura, N. (1968), "Plastic stress-strain matrix and its application for the solution of elastic-plastic problems by the finite element method", International Journal of Mechanical Sciences, Vol. 10, pp. 343-354.
- Yeung, M. R. and Leong, L. L. (1997), "Effects of joint attributes on tunnel stability", International Journal of Rock Mechanics and Mining Sciences, Vol. 34, pp. 348.e1-348.e18.
- Yeung, M. R., Jiang, Q. H. and Sun, N. (2007), "A model of edge-to-edge contact for three-dimensional discontinuous deformation analysis", Computers and Geotechnics, Vol. 34, pp. 175-186.
- Zerwer, A., Polak, M. A. and Santamarina, J. C. (2003), "Rayleigh wave propagation for the detection of near surface discontinuities: finite element modeling" Journal of Nondestructive Evaluation, Vol. 22, pp. 39-52.
- Zhang, C., Pekau, O. A., Jin, F. and Wang, G. (1997), "Application of distinct element method in dynamic analysis of high rock slopes and blocky structures", Soil Dynamics and Earthquake Engineering, Vol. 16, pp. 385-394.
- Zhang, H. H., Li, L. X., An, X. M. and Ma, G. W. (2010), "Numerical analysis of 2-D crack propagation problems using the numerical manifold method", Engineering Analysis with Boundary Elements, Vol. 34, pp. 41-50.
- Zhang, H. W. and Zhou, L. (2006), "Numerical manifold method for dynamic nonlinear analysis of saturated porous media", International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 30, pp. 927-951.
- Zhang, Y. (2009), "Theory and application for high order three dimensional numerical method", PhD Thesis, Hohai University, Nanjing, China. (In Chinese)
- Zhao, G. F., Zhao, X. B. and Zhu, J. B. (2014), "Application of the numerical manifold method for stress wave propagation across rock masses", International

Journal for Numerical and Analytical Methods in Geomechanics, Vol. 38, pp. 92-110.

Zheng, H. and Xu, D. (2014), “New strategies for some issues of numerical manifold method in simulation of crack propagation”, International Journal for Numerical Methods in Engineering, Vol. 97, pp. 986-1010.

Zienkiewicz, O. C., Best, B., Dullage, C. and Stagg, K. (1970), “Analysis of non-linear problems in rock mechanics with particular reference to jointed rock systems”, Proceedings of the Second International Congress on Rock Mechanics, Belgrade.

Zienkiewicz, O. C., Kelly, D. W. and Bettess, P. (1977), “The coupling of the finite element method and boundary solution procedures”, International Journal for Numerical Methods in Engineering, Vol. 11, pp. 355-375.

Zienkiewicz, O. C. and Taylor, R. L. (2000), The Finite Element Method, 5th ed., Butterworth-Heinemann, Oxford.