Seismic Analysis of Post-tensioned Gravity Dams using Scaled Boundary Finite Element Method

Author:
YA, Shukai

Publication Date:
2022

DOI:
https://doi.org/10.26190/unswworks/24464

License:
https://creativecommons.org/licenses/by/4.0/
Link to license to see what you are allowed to do with this resource.

Downloaded from http://hdl.handle.net/1959.4/100757 in https://unswworks.unsw.edu.au on 2022-11-15
Seismic Analysis of Post-tensioned Gravity Dams using Scaled Boundary Finite Element Method

Shukai Ya

A thesis submitted for the degree of
Doctor of Philosophy

School of Civil and Environmental Engineering
Faculty of Engineering
University of New South Wales
Sydney, Australia

August 2022
ORIGINALITY STATEMENT

☑️ I hereby declare that this submission is my own work and to the best of my knowledge it contains no materials previously published or written by another person, or substantial proportions of material which have been accepted for the award of any other degree or diploma at UNSW or any other educational institution, except where due acknowledgement is made in the thesis. Any contribution made to the research by others, with whom I have worked at UNSW or elsewhere, is explicitly acknowledged in the thesis. I also declare that the intellectual content of this thesis is the product of my own work, except to the extent that assistance from others in the project’s design and conception or in style, presentation and linguistic expression is acknowledged.

COPYRIGHT STATEMENT

☑️ I hereby grant the University of New South Wales or its agents a non-exclusive licence to archive and to make available (including to members of the public) my thesis or dissertation in whole or part in the University libraries in all forms of media, now or hereafter known. I acknowledge that I retain all intellectual property rights which subsist in my thesis or dissertation, such as copyright and patent rights, subject to applicable law. I also retain the right to use all or part of my thesis or dissertation in future works (such as articles or books).

For any substantial portions of copyright material used in this thesis, written permission for use has been obtained, or the copyright material is removed from the final public version of the thesis.

AUTHENTICITY STATEMENT

☑️ I certify that the Library deposit digital copy is a direct equivalent of the final officially approved version of my thesis.
UNSW is supportive of candidates publishing their research results during their candidature as detailed in the UNSW Thesis Examination Procedure.

Publications can be used in the candidate's thesis in lieu of a Chapter provided:

- The candidate contributed greater than 50% of the content in the publication and are the "primary author", i.e. they were responsible primarily for the planning, execution and preparation of the work for publication.
- The candidate has obtained approval to include the publication in their thesis in lieu of a Chapter from their Supervisor and Postgraduate Coordinator.
- The publication is not subject to any obligations or contractual agreements with a third party that would constrain its inclusion in the thesis.

☑️ The candidate has declared that some of the work described in their thesis has been published and has been documented in the relevant Chapters with acknowledgement.

A short statement on where this work appears in the thesis and how this work is acknowledged within chapter/s:

Chapter 4 of this thesis is partially comprised of a journal paper that I contributed and published in Computer Methods in Applied Mechanics and Engineering. Chapter 6 of this thesis is partially comprised of a journal paper that I contributed and accepted by Soil Dynamics and Earthquake Engineering. Acknowledgement of the work of other authors of this paper has been made at the beginning of the chapter as well as in my Acknowledgements section.

Candidate's Declaration

I declare that I have complied with the Thesis Examination Procedure.
Acknowledgments

Foremost, I would like to express my gratitude to my supervisor Prof. Chongmin Song, for his continuous support during the past four years. He provided expert guidance and valuable insights during the whole Ph.D studying period. His carefulness and preciseness have greatly inspired me to strive for excellence in research. I can not express enough thanks to him for all the time he has invested to ensure the best quality of research. Besides, his kindness and patience encouraged me to move forward in the most difficult situations. There are countless things I have learnt from him, both in research and in my life.

In addition, my sincere thanks go to my co-supervisor, Dr. Sascha Eisenträger for his help on revising papers. He provided many helpful suggestions on improving the statements in my manuscripts and unreservedly shared his experience on writing papers. Special thanks goes to Dr. Albert Saputra, Dr. Weiwei Xing, Dr. Junqi Zhang, Dr. Qian Huang, Dr. Kang Gao, Dr. Yanling Qu and my colleagues Hongzhe Chen, Yingda Zhang and Yifan Zhan for all help in research and accompany in life.

I would like to acknowledge the financial support I received including the Australian Research Council – Linkage Project (LP160101229) and China Scholarship Council (CSC).

Last but not least, words can not express my thanks to my mother, Yanhong Yu, for her support and sacrifice all these years. All my accomplishments would not be possible without her love and encouragement throughout my life.
Abstract

Dams are hydraulic structures built across rivers to create reservoirs, which provide essential services to society such as flood control, human water supply, and electricity generation. A dam shall be designed to ensure stability against overturning and sliding caused by the hydro-pressure of the reservoir. A common type of dam is the concrete gravity dam that mainly relies on its self-weight and resistance to sliding on the foundation to maintain its stability. Installing post-tensioned anchors (PTAs) is a practical and cost-effective technique in dam engineering. It provides an additional stabilizing force and improves the shear resistance at the dam-foundation interface. Seismic safety evaluation of post-tensioned concrete gravity dams is necessary for new dam designs or strengthened existing dams to guarantee that the structures will survive at specified seismic hazard levels.

This thesis presents the development of an efficient numerical framework for the seismic analysis of post-tensioned concrete gravity dam-reservoir-foundation systems. This framework is realized by implementing the scaled boundary finite element method (SBFEM) in the well-known commercial FEM software ABAQUS as user elements (UEL). Polytope elements (polygonal elements in 2D and polyhedral elements in 3D) are as versatile as standard FEM solid elements, while they provide greater flexibility in mesh generation for bounded domains. Unbounded user elements (UEL) are derived to model wave propagation in far-fields. An unbounded UEL only requires discretization with a small number of faces at the near-field/far-field interface and can rigorously satisfy the radiation condition at infinity.
The ABAQUS software enhanced with the UELs is employed for two-dimensional seismic analysis of gravity dams, overcoming the difficulties encountered in standard FEM, for example, local mesh refinement for geometrical features, generating matching interfacial meshes for weak joints, and simulation of anchor-structure interactions. The overall system consists of a near-field containing the dam body and its neighboring reservoir and foundation, and a far-field of the reservoir and foundation continua. The near-field dam and foundation are discretized as quadtree meshes assigned with polygonal UELs. Quadtree meshes allow rapid and smooth transitions in element size, which facilitates the local mesh refinement for dam lift joints, anchor boreholes, drainage systems, etc. An unbounded UEL represents the far-field foundation in terms of displacement unit-impulse response matrices. It captures free-field motions and transfers them as equivalent seismic inputs acting at the near-field/far-field interface. The reservoir is modeled by ABAQUS built-in acoustic elements. At the far end of the reservoir, a non-reflecting acoustic boundary embedded in ABAQUS is employed to satisfy the radiation condition of the unbounded reservoir.

Comprehensive considerations have been taken in the numeral simulation of post-tensioned gravity dams, such as weak joint behaviors, anchor-structure interaction, and concrete damage. Weak joints in a concrete gravity dam, such as the dam-foundation interface and the dam lift joints, are the most likely places where the sliding and cracking occur. A cohesive-frictional contact scheme is utilized to simulate the non-linear behaviors of these weak joints. A PTA is usually grouted with the structure along a portion of the length, called bond length. At the grouting interface, the bond stress develops with the slippage between the anchor and structure, and then transfers the prestressing in the anchor to the structure. Cohesive elements connected with the anchor and structure are generated along the bond length to simulate the bond-slip interaction. A Mazars’ damage evolution law for dynamic loading is applied to simulate the quasi-brittle behaviors of the concrete. To avoid mesh sensitivity, a partially regularized local damage model is introduced into this application.
Automatic re-meshing algorithms to generate conforming interfacial meshes are developed for the sake of the simulation of interfacial problems. For the weak joints, the domains in contact are allowed to be discretized individually, and then the existing meshes at the interfaces are re-meshed to be node-to-node matching. The anchor is embedded automatically in the structure by inserting additional nodes into the existing structural meshes along the anchor layout. By duplicating the inserted nodes and connecting the duplicated nodes, beam elements conforming with structural meshes are formed naturally. These re-meshing procedures are easily operated on the polygonal meshes allowing arbitrary numbers of nodes and edges. Cohesive elements can be generated with the matching nodes at interfaces, and no constraints are required to connect them with the surrounding elements.

The proposed approach is verified by performing seismic analysis of a post-tensioned gravity dam with simple geometry, and comparing the results obtained from the model using ABAQUS built-in elements. The advantages of the proposed approach in handling complex problems are demonstrated through dams with multiple inclined anchors. Applications of this method can be extended to three-dimensional cases, and composite materials with randomly spread fiber inclusions.
Contents

List of Figures xiii

List of Tables xix

Nomenclature xxi

1 Introduction 1

1.1 Post-tensioned anchors in dam engineering 4

1.1.1 Applications 4

1.1.2 Design of PTA system 6

1.1.2.1 Components and classifications 6

1.1.2.2 Prestressing force 9

1.1.2.3 Bond length design 11

1.1.2.4 Typical PTA layouts 11

1.2 Challenges 12

1.2.1 Dam-reservoir-foundation interaction system 13

1.2.2 Modeling of interfaces 14

1.2.2.1 Non-linearity of weak joints 15

1.2.2.2 Bond-slip phenomena 16

1.2.3 Mesh generation 18

1.2.3.1 Smooth transition mesh 18

1.2.3.2 Conforming mesh 20
1.2.4 Numerical solution method ........................................... 21

1.3 Research aims, methodology, objectives and scope ..................... 22
1.3.1 Research aims ....................................................... 22
1.3.2 Proposed methodology ............................................. 23
1.3.3 Research objectives .................................................. 25
1.3.4 Scope of the work .................................................... 28

1.4 Organization of the thesis ................................................. 29

1.5 List of publications ......................................................... 30
1.5.1 Journal papers ....................................................... 30
1.5.2 Conference proceedings ................................................ 30

2 Literature review .................................................................. 31

2.1 Seismic analysis of dam-reservoir-foundation systems ................. 31
2.1.1 Modeling procedures .................................................. 32
2.1.2 Fluid-structure interaction ............................................ 34
2.1.3 Soil-structure interaction ............................................. 36
2.1.4 Seismic input models .................................................. 38

2.2 Interfacial problems of post-tensioned gravity dam ..................... 39
2.2.1 Modeling of weak interfaces ........................................ 39
2.2.2 Modeling of PTAs .................................................... 41

2.3 Meshing techniques .......................................................... 43
2.3.1 Smooth transition meshes ........................................... 46
2.3.2 Constructing conforming meshes from non-conforming meshes . 48
2.3.3 Anchor inclusions ....................................................... 50

2.4 Damage analysis of concrete .............................................. 52
2.4.1 Damage constitutive models ......................................... 53
2.4.1.1 Evolution laws .................................................... 54
2.4.1.2 Equivalent strain .................................................. 56
2.4.2 Regularized damage models .......................................... 58
# 2.5 Conclusion

## 3 Scaled boundary finite element method

### 3.1 Polyhedral elements in the SBFEM

#### 3.1.1 Scaled boundary geometry transformation

#### 3.1.2 Geometrical properties in scaled boundary coordinates

#### 3.1.3 Governing equations of elastodynamics

#### 3.1.4 Displacement, strain and stress fields

#### 3.1.5 Scaled boundary finite element equation

#### 3.1.6 Solution of the scaled boundary finite element equation

### 3.2 Two-dimensional unbounded element

### 3.3 Near-field/far-field coupling system subjected to free-field motion

### 3.4 Conclusion

## 4 ABAQUS implementation of SBFEM

### 4.1 Introduction

### 4.2 Implementation of polyhedral elements

#### 4.2.1 General workflow of ABAQUS analysis involving user subroutines

#### 4.2.2 Input file formats

#### 4.2.2.1 ABAQUS input file format

#### 4.2.2.2 Text file format

### 4.2.3 Data structure of the mesh information

### 4.2.4 User element

### 4.2.5 Element-based surface definition

### 4.3 Implementation of two-dimensional unbounded elements

#### 4.3.1 Quasi-static analysis

#### 4.3.2 Dynamic analysis

#### 4.3.3 Equivalent seismic loads

### 4.4 Numerical examples
4.4.1 Benchmark tests ........................................... 116
  4.4.1.1 Patch test ........................................... 117
  4.4.1.2 Modal analysis of a cantilever beam ................... 118
  4.4.1.3 Transient analysis of a cantilever beam ................ 119
  4.4.1.4 Wave source problem in a semi-infinite half space .... 122
4.4.2 Examples for non-matching meshes .......................... 127
  4.4.2.1 Patch test for contact analysis ........................ 127
  4.4.2.2 Patch test of cohesive element in conjunction with contact131
  4.4.2.3 A cube with a spherical inclusion ....................... 137
4.4.3 Examples for automatic mesh generation .................... 143
  4.4.3.1 Analysis of a porous asphalt concrete specimen ....... 144
  4.4.3.2 Analysis of a knee joint ............................ 147
4.5 Conclusion .................................................. 152

5 Interfacial problems in post-tensioned gravity dams ............. 155
  5.1 Interface element .......................................... 156
    5.1.1 Stiffness matrix ...................................... 156
    5.1.2 Constitutive law of cohesive element in ABAQUS ....... 161
  5.2 Modeling schemes ........................................... 163
    5.2.1 Modeling of weak interfaces ........................... 164
    5.2.2 Modeling of PTAs considering bond-slip ................. 166
  5.3 Mesh-conforming techniques ................................ 168
    5.3.1 Surface conforming meshes ............................. 170
    5.3.2 Nodal conforming meshes ............................... 172
    5.3.3 Intersection situations ................................ 175
  5.4 Numerical examples ......................................... 176
    5.4.1 Direct shear test simulation ............................ 177
    5.4.2 Pull-out simulation ..................................... 177
  5.5 Conclusion ................................................ 181
6 Seismic analysis of post-tensioned gravity dams

6.1 Introduction ............................................. 186
6.2 Modeling of post-tensioned dam-reservoir-foundation system ............. 189
6.3 Numerical examples ........................................ 192
   6.3.1 Manly dam ........................................... 192
       6.3.1.1 Verification of unbounded S-element ............... 197
       6.3.1.2 Pseudo-static analysis ............................. 199
       6.3.1.3 Transient analysis ................................. 202
   6.3.2 A dam with three joints ................................. 213
6.4 Conclusion .............................................. 224

7 Implementation of a regularized damage model and application to damage
analysis of dams ............................................ 227

7.1 Constitutive law of \( \mu \) model ................................ 228
   7.1.1 Equivalent strains .................................... 228
   7.1.2 Triaxial effect ....................................... 229
   7.1.3 Damage evolution .................................... 230
7.2 Regularization of \( \mu \) model ................................ 231
   7.2.1 Fracture energy ..................................... 233
   7.2.2 Regularization using fracture energy ................... 235
7.3 Regularized \( \mu \) model in polygonal UEL .............................. 237
   7.3.1 Application in polygonal S-element ...................... 237
       7.3.1.1 Strains at the scaling center ...................... 238
       7.3.1.2 Characteristic length ............................. 239
   7.3.2 ABAQUS Implementation ............................... 240
7.4 Numerical examples ....................................... 241
   7.4.1 A single element under uniaxial cyclic loading ............... 242
   7.4.2 Plain Koyna dam .................................... 244
   7.4.3 Post-tensioned Koyna dam ............................. 253
7.5 Conclusion .......................................................... 256

8 Conclusions and recommendations .......................... 259
  8.1 Concluding statements ...................................... 259
  8.2 Recommendations for future work ..................... 263

Bibliography .......................................................... 267
List of Figures

1.1 Sketch of FEM modeling of a dam-reservoir-foundation system anchored by a vertical PTA ........................................ 3
1.2 Applications of PTAs in dam engineering ........................ 7
1.3 Components of a typical PTA [1, 2] .............................. 8
1.4 Forces acting on a post-tensioned gravity dam [3] ............. 10
1.5 Schematic illustration of the typical uses of PTAs ............... 12
1.6 Schematic of a dam-reservoir-foundation interaction system .. 13
1.8 Schematic of bond stress distribution along tendon bond length [5] .... 17
1.9 Local bond stress vs. slippage[6] .................................. 18
1.10 Zones needing fine meshes in post-tensioned gravity dams ...... 19
1.11 S-elements and their coupling ..................................... 24
1.12 Remeshing to obtain conforming meshes ........................ 25
2.1 Modeling procedures of dam-reservoir-foundation systems [7] .... 33
2.2 Residual displacements for gravity dams with weak interfaces [8] .... 40
2.3 Modeling of weak interfaces using joint element .................. 41
2.4 Modeling of bond-slip interaction .................................. 44
2.5 Bond-slip constitutive laws .......................................... 45
2.6 Strategies of generating smooth transition elements ............. 47
2.7 Modified quadtree meshes for the standard FEM ................. 48
2.8 An example of mesh matching on hexahedral meshes by pillowing sheet and dicing column [9] ................................................................. 49
2.9 An example of mesh matching on polyhedral meshes [10] ........... 50
2.10 Anchor inclusion by enriched elements based on XFEM [11] ....... 52
2.11 Anchor inclusion in polygonal elements [12] ............................ 53
2.12 Schematic representation of the tensile stress-strain relationship in three softening models ................................................................. 54

3.1 Schematic of a polyhedral mesh ................................................. 62
3.2 Triangular and quadrilateral reference elements in natural coordinates $\eta, \zeta$ .......................................................... 63
3.3 Scaling of the surface element using radial coordinate $\xi$ ($\xi = 0$ at the scaling center and $\xi = 1$ on the boundary) ......................... 64
3.4 Unbounded S-element ...................................................... 83
3.5 Coupling of near-field/far-field .................................................. 87
3.6 A free-field motion system subjected to a vertically propagated S-wave . . 88

4.1 General workflow for ABAQUS/Standard analysis involving SBFEM user subroutines ................................................................. 98
4.2 A polyhedron mesh with two elements ........................................ 99
4.3 An example of ABAQUS input format involving SBFEM user elements . 99
4.4 An example of text input format describing SBFEM polyhedral mesh .. 101
4.5 Flowchart of user subroutine UEL for SBFEM .......................... 105
4.6 Overlaying standard element on SBFEM user element for defining surface 111
4.7 Extraction of $u_b$ for the trapezoidal rule based on historical solutions 115
4.8 Modeling of a quadrangular prism .............................................. 117
4.9 Contours of a quadrangular prism under uniaxial tension .............. 118
4.10 Model of a cantilever beam ..................................................... 119
4.11 Modal analysis comparison of a cantilever beam ....................... 120
4.12 A cantilever beam subjected to dynamic loading ...................... 121
6.15 Verification of PTA behavior of Manly dam ............................................. 211
6.16 Seismic response comparison of Manly dam with perfect bond and bond-
slip ................................................................. 212
6.17 Geometry of a dam with three lift joints (unit: m) ................................. 213
6.18 Anchoring schemes for a dam with three lift joints ............................... 215
6.19 Mesh of the dam-reservoir-foundation system for a dam with three lift joints 217
6.20 El Centro wave .............................................................................. 218
6.21 Seismic response of a dam with three lift joints subjected to El Centro wave 219
6.22 Displacement contours of three models for a dam with three lift joints
(time: 5.00 s, unit: m, deformation scale factor: 25) .................................. 220
6.23 Lift joint behavior of a dam with three lift joints subjected to El Centro
wave ............................................................................. 221
6.24 First principal stresses of three models for a dam with three lift joints
(time: 1.50 s, unit: MPa) .................................................................. 222
6.25 PTA (Anchor 1) behavior of a dam with three lifts subjected to El Centro
wave ............................................................................. 223
7.1 Typical stress-strain curves of $\mu$ model under uniaxial loadings .......... 232
7.2 Crack modeling using fracture energy approach [13] ............................ 234
7.3 Energy dissipation of an element for concrete fracture .......................... 235
7.4 Tensile stress-strain relationship of $\mu$ model regularized by fracture energy 236
7.5 Flowchart of subroutine MAZARD incorporating concrete damage model
into UEL .............................................................................. 242
7.6 Uniaxial test of a simple square model .................................................. 243
7.7 Histories of $\epsilon_Y$, $Y$ and $\omega$ of an element under uniaxial cyclic loading .... 245
7.8 Stress-strain relationship of an element under uniaxial cyclic loading .... 246
7.9 Horizontal component of Koyna earthquake ....................................... 247
7.10 Geometry of Koyna dam (Unit: m) ...................................................... 247
7.11 Numerical models of Koyna dam ......................................................... 249
7.12 Cracking properties of Koyna dam concrete [14]  
7.13 Stress-strain curves of CDP model and $\mu$ model for Koyna dam concrete  
7.14 Meshes of Koyna dam for SBFEM models  
7.15 Koyna dam crest/heel horizontal relative displacement  
7.16 Tensile damage contour comparison of Koyna dam  
7.17 Post-tensioned Koyna dam  
7.18 Post-tensioned Koyna dam crest/heel horizontal relative displacement  
7.19 Tensile damage contour comparison of Post-tensioned Koyna dam models
List of Tables

4.1 Data structure stored in the COMMON blocks ........................................ 102
4.2 Data structures of relative nodal coordinates and element topology .......... 106
4.3 Maximum nodal $L^2$ error norm of a quadrangular prism under uniaxial
tension ........................................................................................................... 118
4.4 $L^2$ Error norm of $\sigma_z$ of different models with different curvatures .... 130
4.5 Material properties of the patch test inserting cohesive element .............. 134
4.6 Material properties of a cube with a spherical inclusion ............................ 138
4.7 Comparison of computational times (s) for non-linear analysis ................. 142
4.8 Material properties of a porous asphalt concrete specimen ...................... 145
4.9 Material properties of the knee joint [15] .............................................. 149

5.1 Cohesive-frictional contact parameters of concrete-sandstone contact [4] . 177
5.2 Parameters of quadrilinear model for sandstone-grout interface ............... 180

6.1 Material properties of Manly dam-reservoir-foundation system ............... 194
6.2 Cohesive-frictional contact parameters of concrete-sandstone joint [4] .... 194
6.3 Parameters of quadrilinear model for sandstone-grout interface ............... 194
6.4 Material properties of the dam-reservoir-foundation system for a dam
with three lift joints ......................................................................................... 214
6.5 Cohesive-frictional contact parameters for weak joints in a dam with three
lift joints [4] ..................................................................................................... 216
6.6 Parameters of quadrilinear models for bond length of different anchors for a dam with three lift joints [5, 16, 17] ............... 216

7.1 Material properties of uniaxial test ......................... 243
7.2 Characteristic values of strains for uniaxial test ............... 243
7.3 Material properties of Koyna dam concrete [14, 18] .......... 250
7.4 Parameters of $\mu$ model for Koyna dam concrete ............... 250
7.5 Parameters of quadrilinear model for anchor-structure grouting interface of Koyna dam [5] .................. 256
Nomenclature

$x, y, z$  Cartesian coordinates

$f_s^a$  dam safety factor against overturning

$f_s^s$  dam safety factor against sliding

$\langle \rangle$  MacAulay bracket

$\sigma$  stress tensor

$\varepsilon$  strain tensor

$D$  elasticity matrix

$\omega$  effective damage variable

$\omega_c$  compressive damage variable

$\omega_t$  tensile damage variable

$\tilde{\varepsilon}$  equivalent strain

$\lambda$  diagonal matrix of eigenvalues

$\phi$  eigenvector matrix

$\Delta\bar{t}$  time range of a subinterval in trapezoidal rule

$\eta, \zeta, \xi$  scaled boundary coordinates
Cartesian coordinates with respect to the scaling center
\[ x, y, z \]

strain-displacement matrices
\[ B_1, B_2 \]
vector of integration constants
\[ c \]
coefficient matrices
\[ E_0, E_1, E_2, M_0, Z_p \]
force vector due to free-field motion acting on far-field
\[ F^f_{\infty} \]
force vector due to free-field motion acting on near-field
\[ F^f_b \]
outward normal vectors
\[ g^\xi, g^\eta, g^\zeta \]
unit vectors in the Cartesian coordinates
\[ i, j, k \]
Jacobian matrix
\[ J \]
stiffness, mass, damping matrices of bounded element
\[ K, M, C \]
stiffness, damping matrices of unbounded element
\[ K_{\infty}, C_{\infty} \]
differential operator
\[ L \]
normalized outward normal vectors
\[ n^\xi, n^\eta, n^\zeta \]
nodal force functions along the radial coordinate
\[ q(\xi) \]
position vector
\[ r \]
ground interaction force
\[ R_b \]
dynamic stiffness matrix of an unbounded S-element
\[ S_{\infty}(\omega) \]
regular part of the dynamic stiffness matrix of an unbounded S-element
\[ S^r_{\infty} \]
tractions on surfaces
\[ t^\xi, t^\eta, t^\zeta \]
displacement functions along the radial coordinate
\[ u(\xi) \]
N  shape functions in matrix form

dS_ξ, dS_η, dS_ζ  infinitesimal surface area

nΔt  truncation time

α_R, β_R  Rayleigh damping coefficients

δ^o, δ^f, δ^{max}  separation/slippage of cohesive element for damage initiation, complete failure, and the maximum value during the loading

u^f_b  free field motion on near-field boundary

u^s_b  scattered motion on near-field boundary

μ_s, μ_k  static and kinetic friction coefficients

σ, τ  normal stress, shear stress

τ_u, τ_r  ultimate and residual bond stresses

D  damage variable of cohesive element

s_1, s_2, s_3  characteristic slippages in the quadrilinear bond-slip model

A_t, B_t, A_c, B_c, k  Parameters of Mazar’s concrete damage model

G_i  fracture energy

g_t  dissipated energy per unit of volume

l_{ch}  element characteristic length

r  triaxial factor of Mazar’s concrete damage model
Chapter 1

Introduction

Dams are hydraulic structures built across rivers to create reservoirs for impounding water. They are vital infrastructure to society due to the essential services, such as flood control, water supplying, irrigation, electricity generation, and recreation. Safety evaluation of dams is critical since dam failures are usually catastrophic. In general, a dam should be designed to ensure stability against overturning and sliding under three levels of loading conditions: usual, unusual, and extreme. Earthquakes are often involved in unusual and extreme loading conditions, and seismic analysis of dams is a mandatory requirement for new dam designs or strengthened existing dams to guarantee that the structures will survive at specified seismic hazard levels [19].

Time history dynamic analysis based on the finite element method (FEM) has been usually utilized for the seismic evaluation of dams, especially for large dams with high importance [19, 7, 20]. This problem is a challenging problem involving fluid-structure interaction (FSI) and soil-structure interaction (SSI) (see Section 1.2.1 for more details). A rigorous and efficient computational framework to model dam-reservoir-foundation interaction systems is usually employed for realistic considerations [21]. In this way, the fluid-structure and soil-structure interactions can be reflected through hydrodynamic pressures at the structure-water interface and mutual stresses at the foundation-structure interface, respectively. The coupled model is generally partitioned into near-field and far-field
domains. The near-field is a bounded domain that consists of the dam body and its adjacent reservoir and foundation. The far-field domain is unbounded and comprises the continua of the reservoir and foundation that extend to infinity. At the near-field/far-field interfaces, artificial boundary conditions (ABC) are usually constructed to satisfy the so-called radiation conditions that absorb outgoing waves.

The installation of post-tensioned anchors (PTA) in dam engineering is a practical and cost-effective technique to strengthen dams [3]. It has been widely used for various purposes, such as upgrading, rehabilitation, and new dam designs. The main application of installing PTAs is upgrading the old dam and thus, increasing the stability with respect to the extreme loads considered in modern safety standards, such as the probable maximum flood (PMF) and maximum credible earthquake (MCE). There are also existing dams, which have been damaged by severe earthquakes, where PTAs have been installed as remedial measures. Moreover, PTAs have been applied in the initial design of concrete dams, i.e., pre-stressed dams, and they are likely to significantly reduce the concrete volume and construction costs compared to unanchored dam designs.

Seismic analysis of post-tensioned dams involves the modeling of PTAs, which makes the problem more complicated. The embedment of PTAs into concrete blocks, especially inclined anchors, increases the meshing burdens in the framework of standard FEM. A PTA is usually grouted with the structure along a portion of its length, called bond length [5]. The prestressing of PTAs is transferred into the structure through the bond stress at the grouting interface. The bond stress develops with the slippage between the anchor and structure, and thus, the bond-slip interaction should be considered since the pull-out is a common type of PTA failure [2]. Besides, this thesis aims at a common type of dam, i.e., concrete gravity dams that mainly rely on their self-weight and sliding resistance on the foundation to maintain its stability. The concreting of a gravity dam is usually in stages, which introduces construction joints between dam lifts. Those joints are relatively weak interfaces where cracking and sliding mostly occur [4]. The interfacial behavior of the weak joints significantly affects the dam stability. Installing PTAs generates
additional compressive stresses at those joints and improves their resistance to cracking and sliding. To comprehensively evaluate the seismic performance of post-tensioned concrete gravity dams, sophisticated modeling with the features of weak joints and bond lengths should be established. A post-tensioned gravity dam-reservoir-foundation system with a vertical PTA is presented in Fig. 1.1.

In thesis, an efficient framework for two-dimensional seismic analysis of post-tensioned gravity dams are developed based on the scaled boundary finite element method [22]. This numerical method is capable of constructing polytope elements that offer greater flexibility regarding the mesh generation. The PTA modeling can be realized in an automatically manner by inserting nodes along the anchor layouts [12]. Besides, the dynamic properties of unbounded media can be rigorously captured using the SBFE approach [23].
The remaining of this chapter presents a brief background of installing post-tensioned anchors (PTAs) in dam engineering in Section 1.1, containing the typical applications and design considerations of PTAs. The current challenges in seismic analysis of post-tensioned concrete gravity dams are summarized in Section 1.2. Targeting on efficiently solving these challenges, the proposed approach, objectives and scopes of this thesis are presented in Section 1.3. Finally, Section 1.4 outlines the organization of this thesis.

1.1 Post-tensioned anchors in dam engineering

PTAs have been widely used to strengthen concrete gravity dams for either existing dams or new dams [24, 2]. The PTAs transmit the prestressing forces from the prestressing tendons to the ground, or the supported structures [25], thereby increase the resistance to sliding and overturning of a dam [26]. This section briefly summarizes the purposes of installing PTAs, the design considerations, and typical layouts of PTAs in dam engineering.

1.1.1 Applications

The installation of PTAs in dam engineering has more than 80 years of history with a wide range of applications [27, 3, 2]. The first recognized use of PTAs in dam engineering was back to 1934; it was applied in the Cherfas gravity dam in Algeria to raise and strengthen the 30 m high masonry structure [28]. The original Cheuras dam constructed in the 1880s did not consider the uplift force [29] and suffered flood damage on various occasions in the following years [3]. The ingenious method of anchoring the weak dam to foundation rock with post-tensioned vertical anchors was devised by a French civil engineer André Coyne; thus it is known as Coyne method is some literature [29].

Several review works ranging from 1965 to 2015 [29, 30, 3, 31, 2] identified hundreds of relevant cases and technical publications, from which the purposes of installing PTAs can be generally classified into four categories, as shown in Fig. 1.2: 1) improving the
overall dam stability, 2) stabilizing the appurtenant structures of dams or foundation, 3) repairing damaged dams due to strong earthquakes and 4) designing new dams:

- Among many applications, the primary purpose of installing PTAs is upgrading the overall dam stability. The probable maximum flood is a consideration in the modern standards, and it might be far higher than the historical flood level used in the design of many old dams [1]. Besides, the maximum credible earthquake determined under the modern standards differs greatly from the old standards, which leads to the existing old dams may not meet the stability criteria [3, 32]. The reasons for strengthening old dams are usually related to the PMF (such as the Great Falls [33], Goat Rock [34] and Shepaug dams [35]), MCE (such as the Minidoka [36] and Stewart Mountain dams [37]), deficiency in design (uplift) (such as the Elkhart dam [38]), or multiple purposes (such as the Cheuras [28], Condit [39] and Morgan Falls dams [34]).

- The appurtenant components of dams or foundation were anchored for the stabilization. Examples contain the Meadow-Bank [40] and Muda dams [41] (foundation); the Nepean dam [42, 43] (spillway, training wall and still basin); the Pacoima [44], Tekeze [45] and Steenbrass dams [46] (abutment), the St Michel [24] and Mont-Larron dams [47] (buttress), etc.

- Severe cracking has been observed in several concrete dams that experienced strong earthquakes. Placing PTAs in the cracked regions was employed as one of the retrofitting techniques in several examples. The Koyna Dam was damaged by a recorded earthquake with a peak ground acceleration (PGA) of 0.49 g [48], which taller monoliths were installed with PTAs passing through the horizontal cracks [49]. The SefidRud dam experienced an earthquake in 1990, and heavy damage was observed in the central monoliths at the concrete lift joints [50]. The rehabilitation techniques of the SefidRud Dam contain installing PTAs in the cracked areas and grouting cracks [51].
There are new concrete dams that were initially designed and constructed by installing PTAs, i.e., prestressed dams, as the prestressed concrete design has advantages from the economic aspect. The first prestressed concrete dam was the Emestina dam in Brazil, completed in 1955; it reduced the volume of concrete from 22,000 m$^3$ for the classical dam shape to 7,300 m$^3$ [24]. The Allt-na-Lairige dam in North of Scotland constructed in 1956 was prestressed by 28 PTAs, which scheme reduced the volume of concrete by 40% [52]. The construction cost of the Catabgunya dam, constructed in Australia by 1960, was reduced by 50% compared with an unanchored dam design [53].

1.1.2 Design of PTA system

The components and the classifications of typical PTA systems are introduced first. Moreover, the principles of the prestressing force design to guarantee the dam stability are explained. The design approach of bond length is presented since it is related to a common potential failure mechanism of PTA systems, i.e., pull-out failure. Finally, typical layouts of PTAs in dam engineering are summarized.

1.1.2.1 Components and classifications

The basic components of a PTA system include: i) an anchorage, and high strength prestressing tendon with ii) free (stressing) length and iii) bond length, as illustrated in Fig. 1.3. The anchorage is a combined system of anchor head, bearing plate, and stressing jack. The prestressing tendons are usually installed in grout-filled drill holes, i.e., boreholes, and encapsulated in double or triple flexible corrosion protection systems (sheathes).

The PTAs can be classified as fully bonded, partially bonded, or unbounded anchors based on the grouting condition of the free length [5]. A fully bonded PTA refers to the method in which the whole free length of the tendon is grouted with the surrounding concrete/rock. A partially bonded anchor provides a certain amount of bonded free length
Figure 1.2. Applications of PTAs in dam engineering

(a) Improving overall stability (Cheuras dam) [28]

(b) Stabilizing foundation (Meadow-Bank dam) [40]

(c) Rehabilitation of dam damaged by earthquake (Se-fidRud dam) [49]

(d) New dam design (Allt-na-Lairige dam) [52]
Figure 1.3. Components of a typical PTA [1, 2]
near the anchorage. For an unbounded anchor, the free length of the tendon is free to
elongate elastically in the sheath coated with grease. At the tendon bond length, the
tendon is bonded with the surrounding rock through cement grouting. Bond stress will
be generated along the grouting interfaces, which is capable of transmitting the applied
tensile load into the ground.

The focus of this thesis is on the unbonded anchors, since they are more frequently
used nowadays. Compared to the fully bonded PTAs, unbonded PTAs have the follow-
ing advantages: i) prestressed tendons can be monitored and re-tensioned, ii) ungrouted
tendons in corrosion protection systems are more durable [26].

1.1.2.2 Prestressing force

The prestressing force of PTAs should be designated to guarantee the dam stability. The
overall stability of the strengthened dams is accessed based on the safety factor, in the
same manner as for a normal gravity dam with the addition of extra stabilizing forces from
the PTAs [3]. Generally speaking, a gravity dam should be prevented from instability,
such as i) overturning at its toe and ii) sliding along the concrete-rock interface or any
weak plane within the foundation. Prestressing forces provided by the installed PTAs,
together with the dead weight, counteract the overturning and sliding of a gravity dam.
Figure 1.4 shows the possible forces acting on a typical post-tensioned gravity dam.

Regarding the stability against sliding along the dam-foundation interface, the dam
safety factor is calculated as

\[ f_s^s = \frac{AC + (V_t + F)f}{H_t}, \]

in which \( f_s^s \) is the shear friction factor of safety, \( A \) is the contact area, \( C \) is the unit shear
resistance, \( f \) is the frictional coefficient, \( V_t (\sum_{i=1}^{3} V_i) \) and \( H_t (\sum_{i=1}^{7} H_i) \) are the total vertical
and horizontal forces, respectively, as indicated in Fig. 1.4.

The stability of a post-tensioned gravity dam against overturning at its toe (“O” in
Fig. 1.4 ) has a safety factor expressed as

\[ f_s^0 = \frac{\Sigma MW + MF}{\Sigma MV}, \]  

(1.2)

where \( \Sigma MW \) is the summation of resisting moments (\( V_1 \) and \( H_5 \) about \( O \)), \( \Sigma MV \) is the summation of overturning moments (\( H_1, H_2, H_3, H_4, H_6, H_7, V_2 \) and \( V_3 \) about \( O \)), and \( MF \) is the moment of the prestressing force \( F \) about \( O \).

The load combinations in dam safety assessments are generally categorized as usual, unusual, and extreme conditions. A usual load combination includes the normal design reservoir level and the appropriate dead loads. An unusual load combination is the modification of the usual load combination, by replacing the normal design reservoir level with the PMF level. An extreme load combination is the usual load combination acting together with the MCE. The PTAs are usually installed to strengthen dams to resist unusual and extreme loadings. The designated prestressing forces should guarantee minimum allowable safety factors (in Eqs. (1.1) and (1.2)) of 2.0 and 1.3 under unusual and extreme loading combinations, respectively [54].
1.1.2.3 Bond length design

The design of the bond length deserves special mention among all design considerations. A PTA transmits the tensile load into the rock through the cement grouting at the bond length; thus the capacity of the PTA system highly depends on the shear strength at the grouting interfaces. In the literature, the principal failure mechanisms of a PTA system are summarized as: a) tensile failure of the tendon, b) shear failure of the surrounding rock, and c) bond failures at the tendon-grout and rock-grout interfaces [25, 5]. Among those failure modes, the interfacial failure of rock-grout bond, i.e., pull-out failure, is deemed to be critical to the capacity of a PTA system in many applications [55, 17, 2], especially for straight shaft anchors [30].

The current design approach of the bond length has little change since the review work of Littlejohn and Bruce [30], which simply assumes a uniform bond stress distribution along the bond length. The initial bond length design can be estimated based on the working bond stress for a specific rock type, expressed as [30, 25, 5]

\[ L_b = \frac{P}{\pi d \tau_w}, \]

where \( L_b \) is the designed bond length, \( P \) is the design load of the anchor, \( d \) is the diameter of the borehole, \( \tau_w \) is the working bond stress along a rock-grout interface. The ultimate/working bond stresses for common rock types have been recommended in Ref. [5]. Nonetheless, this empirical method relies heavily on the selection of the presumptive working bond stress. Therefore field pull-out tests should be conducted to verify the design during and before construction [19].

1.1.2.4 Typical PTA layouts

The classical layouts of PTAs in dam engineering for the purposes of strengthening and rehabilitation are presented in Fig. 1.5. The boreholes for installing PTAs can be drilled into the dam body or extended into the foundation, i.e., ground anchors. There are two
Figure 1.5. Schematic illustration of the typical uses of PTAs

typical ground anchors, as shown in Fig. 1.5a, in which Types A1 and A2 stabilize the concrete gravity dams against overturning and downstream sliding, respectively [56, 19, 2]. The main purpose of utilizing the ground anchors is to upgrade the dam stability with respect to higher standards of extreme loads (PMF and MCE) as preventive measures. Besides, PTAs have been installed in the dam body to control crack propagation, see A3 in Fig. 1.5b. This layout is commonly used in dams that have experienced severe seismic events as remedial measures, such as the widely known cases: the Menjil dam in Iran [57] and the Koyna dam in India [49].

1.2 Challenges

Post-tensioned gravity dams situated in seismic areas may be subjected to earthquakes. The seismic analysis of post-tensioned gravity dams is essential to the dam safety assessment. The finite element method (FEM), a numerical approach to undertake structural seismic analyses, has been highly recommended or regarded as a mandatory requirement in many engineering manuals [7, 20].

This section presents the challenges encountered in the seismic analysis of post-
tensioned gravity dams using the FEM approach. The challenges include the modeling of dam-reservoir-foundation interaction system, modeling of interracial problems, mesh generation, and solution issues of non-linear equations.

1.2.1 Dam-reservoir-foundation interaction system

In the FEM approach, a complete system that consists of the dam, reservoir, and foundation regions should be modeled and analyzed [7]. The analysis of this model is a complicated problem containing the fluid-structure (FSI) and the soil-structure interactions (SSI), which leads the model to a dam-reservoir-foundation interaction system. A schematic diagram of a dam-reservoir-foundation interaction system with a PTA is illustrated in Fig. 1.6. At the interfaces ($\Gamma_{fs}$) between reservoir and dam/foundation, the FSI should be established, i.e., the compatibility between hydrodynamic pressures $p$ and normal displacements should be fulfilled [59]. At the soil-structure ($\Gamma_{ss}$), the SSI should be considered, i.e., the kinematic and inertial interactions should be satisfied at the same time [60]. The near-field of foundation is treated as structure in this schematic.
The overall system is usually partitioned into the modelings of the near-field and far-field systems. The near-field system is a bounded domain comprised of the neighboring reservoir and foundation regions in conjunction with the dam. The meshes of near-field reservoir and foundation should be extended to a distance in the upstream, downstream, and downwards directions, to account for their interactions with the structure properly. Although there are simplified procedures to take into these interactions into consideration, such as Westergaard’s added mass approach [61] for FSI and massless foundation model [62] for SSI, rigorous computational finite element methodologies are usually utilized to model dam-reservoir-massed foundation systems for realistic simulations over the last decades [21, 63, 64, 65, 66]. As unbounded domains, the reservoir and foundation have one or more dimensions extending to infinity, which are modeled as far-fields.

The far-field system should satisfy the so-called radiation damping conditions absorbing outgoing waves [67, 68]. Earthquake sources are usually located far from the structure, i.e., the seismic excitation is generated in the far-field system. During an earthquake event, the released energy travels in the form of seismic waves into the near field system. If the incident waves encounter obstacles like structures and surface topography, they will be scattered and partially propagate towards infinity which can not be reflected to the near-field system, i.e., radiation damping conditions. If the spatial domains of the problem are unbounded, artificial boundary conditions (ABC) should be established in order to make the computational domain finite. The numerical solution of such wave problems is a challenging issue in seismic analysis, and great efforts have been made by scholars. At the interfaces (\(\Gamma_r^\infty\) and \(\Gamma_f^\infty\)) between the near-field and far-field systems, either local or global ABCs [69, 70, 65, 71] should be constructed to capture the dynamic properties of the unbounded domains.

1.2.2 Modeling of interfaces

There are interfacial problems existing in the seismic analysis of post-tensioned gravity dams, such as the contact and cohesive fracture at the dam-foundation interface and con-
struction joints, and the bond-slip interaction at the anchor-structure grouting interfaces.

1.2.2.1 Non-linearity of weak joints

Dam-foundation interface and construction joints are weak joints in a concrete gravity dam, since they are the most likely places for dams to fail post-earthquake by sliding [4, 20]. Besides, the lowest tensile strengths are commonly associated with such weak joints, and thus, cracking induced by seismic events tends to propagate along such features [4, 20]. This cracking may result in higher uplift pressures and further, reduce the dam stability against overturning. The influences of the weak joints on the seismic safety of concrete gravity dams have been investigated by several researches [72, 73, 74]. With the inclusion of weak joints, the stress contour shapes are changed and the maximum tensile stresses are reduced [74]; thus cracking in concrete blocks is avoided to some extent [73]. However, large sliding and cracking may occur at those weak interfaces, and the weak joint damage is the dominant concern for the dam stability [73, 74].

The interfacial behavior of the weak joints is complex. The construction joints are often sealed, which results in the cohesion to resist cracking and sliding. Cohesion exists in both the normal and tangential directions. In the normal direction, cracking and closure might occur during dynamic analysis. Before the cohesion of a weak joint is completely damaged, tensile stresses are generated when the interfaces at two sides are separated. Contact is activated when the interfaces are in closure and have a penetration tendency. In the shear direction, the resistance depends on both cohesion and friction. The degradation of cohesion occurs at small strain while the mobilization of friction requires larger slippage [75]. Figure 1.7 presents a typical shear stress-slippage curve for a joint. The curve exhibits a maximum shear stress (break bond strength) at small displacement. At this stage, both the cohesion and friction contribute to the shear resistance. With continued slippage, the cohesion is damaged, and thus, the shear stress decreases until reaching a constant, minimum value (residual strength) at large displacements.

Modeling these weak joints is a challenging task of dynamic contact problems with
adhesion. Non-linear kinematics are involved in a dynamic contact problem [76]. The kinematic constraints should be imposed on the interacting bodies during the crack and closure procedures. Kinetic friction should be considered as the sliding has velocities in a dynamic analysis [77]. The constitutive law of the cohesion should be set properly to reflect the degradation in the normal and tangential directions.

### 1.2.2.2 Bond-slip phenomena

The working capacity of a PTA system highly depends on the bond strength of the anchor-structure grouting interface, since the pull-out failure is deemed to be the most critical failure mechanism in many applications [55, 17, 2]. The bond resistance at the rock-grout interface comprises the adhesion, mechanical interlocking, and friction between the rock and grouting [78, 2], and it depends on many variables, such as grout mix, rock type, rock strength, borehole roughness, and diameters, etc [17]. The bond stress usually develops with the slippage between the anchor and structure, i.e., bond-slip interaction.

There is an obvious gap between the design approach and the knowledge about the bond stress distribution along the bond length. Due to the complexity of bond stress development, the current anchor design approaches simply assume a uniform bond stress distribution, and the initial bond length design can be estimated based on the ultimate bond stress for a specific rock type [30, 25, 5], as mentioned in Section 1.1.2.3. It has
been proved that bond stress distribution is non-uniform by experimental tests [17, 79], theoretical derivations [80], and numerical simulations [16, 81]. A schematic of the bond stress distribution along the bond length under different stages of loading is presented in Fig. 1.8. At the initial loading stage, the bond stress tends to be exponentially decayed from the proximal end to zero at the distal end. With a progressive loading, the bond stress exceeds the ultimate bond stress starting from the proximal end, and then it will reduce to a residual value and the peak bond stress moves towards the distal end. At the ultimate loading stage, the bond stress holds the residual value along a substantial portion of the bond length, and it has a peak value near the distal end. After the ultimate loading stage, the bond length only provides residual stress, and the full-out failure may occur. The design approach introduces efficiency factors to consider the bond stress non-uniformity. Nonetheless, this empirical method still relies heavily on the selection of the presumptive ultimate bond stress.

Bond-slip interaction at the bond length should be taken into consideration for realistic modeling of the PTAs. This consideration presents a challenge: interface elements should be established to connect the anchor and the structure. The constitutive law of these interface elements should reflect the local bond stress-slippage relationship in the
tangential direction, and prevent penetration between the anchor and structure in the normal direction. Figure 1.9 presents a typical local bond-stress slippage relationship. In general, the rock-grout bond failure process has a progressive nature, in which at least three distinct stages may be identified: i) pseudo-elastic behavior at small deformations, ii) de-bonding development at moderate displacements, and iii) residual behavior after larger displacements.

1.2.3 Mesh generation

To obtain analysis-ready meshes for seismic analysis of post-tensioned gravity dams, there are two considerations during the mesh generation. Firstly, local mesh refinement should be done for the regions where high-stress gradients exist. Secondly, conforming meshes are beneficial or necessary for interfacial problems. This section introduces the situations of these two requirements for mesh generation.

1.2.3.1 Smooth transition mesh

The high-stress gradients may exist in the regions of the i) anchor head, ii) bond length, iii) weak joints, and iv) geometries with sharp changes, as shown in Fig. 1.10. The pre-stressing force in a PTA is transmitted to the dam body through the bearing plate of the anchor head. Although the bear plate offers a contact area with the dam body, which reduces the pressure acting on the anchor head zone, the pre-stressing force is still an external force with a large magnitude (similar to a concentrated load). Subsequently, the
zone near the anchor head will experience stresses with high gradients. As to the bond length, the bond stress is obviously non-uniform and may change rapidly, see Fig. 1.8. To better reflect the local bond stress-slippage relationship, the meshes at the bond length should be fine enough. At the weak joints, there are non-linear interfacial behaviors as discussed in Section 1.2.2.1, thus sophisticated meshes should also be generated. There might be geometries with sharp changes in dam engineering, where stress concentration is easily occurring, such as dam slopes with sharp changes and corners of rectangular drainage.

Local mesh refinement is required for the regions mentioned above; by contrast, the rest of the domain with smooth stress variations should be discretized into coarse meshes for saving computational resources and improving computational efficiency. This difference may result in irregular shapes of meshes with unreasonable aspect ratios, especially for the transition regions between fine meshes and coarse meshes. Generating meshes
with smooth transitions in element size is a challenging topic in the framework of standard FEM [82, 83, 84, 85].

### 1.2.3.2 Conforming mesh

There are interfacial problems in the modeling of post-tensioned gravity dam, such as the weak joint modeling with cohesive contact and grouting interface modeling with bond-slip. Conforming meshes (matching nodes/edges) are favorable for improving the accuracy of contact modeling [86]. For the anchor-structure interaction, a FE mesh must be arranged so that an anchor is located along the edges of solid elements, and node pairs are required to reflect the relative displacement (slippage) between the anchor and the surrounding rock/concrete [87].

In contact mechanics, it is practical to discretize different domains in contact independently, resulting in non-matching meshes at the interfaces. Especially for curved interfaces, there might be gaps and penetrations at the interfacial meshes. Thus the contact modeling approach usually employs surface to surface (STS) scheme for non-conforming meshes, which may reduce the accuracy for contact modeling [86, 88, 89]. The contact modeling employing a node to node (NTN) scheme enhances the performance compared to the STS approach [86, 88], while the NTN scheme is realized on matching meshes.

The simulation of the bond-slip interaction at the rock-grout interface usually utilizes the linkage [90] or interface elements [91, 92, 93]. A linkage element connects one anchor node with a corresponding structural node. These two nodes have the same coordinates, and are referred as matching nodes. An interface element usually has zero thickness and is comprised of four nodes. Similar to the linkage element, an interface element also connects the nodes from the anchor and structure, but it connects two adjacent pairs of matching nodes. No matter using the linkage or interface elements, conforming meshes containing matching nodes are necessary for the numerical simulation of the bond-slip interaction between the anchor and the structure [87].

In the framework of standard FEM, the meshing procedure to generate conforming
meshes requires great human efforts. Firstly, special treatments should be taken to guarantee the meshes are matching at the interfaces, such as assigning mesh seeds on the two sides of interfaces. Besides, to fulfill the requirement that the anchor is located along the edges of structural elements, partitioning of the structure domain should be introduced. If there are multiple inclined anchors interacting with each other, the partitioning may result in irregular shapes of meshes with unreasonable aspect ratios during the mesh generation.

### 1.2.4 Numerical solution method

Seismic analysis of post-tensioned gravity dams using FEM approach involves numerical methods to perform time integration and to solve non-linear equations containing structural non-linearity (contact with friction) and material non-linearity (cohesive interfaces). Because of the approximate nature of numerical solutions, errors usually exist due to the truncation errors and round-off errors [94]. Especially for the problem at hand, significant accumulative errors may be obtained during many dynamic increments in solving non-linear solutions. Employing proper numerical solution methods is essential to get approximate but credible accurate solutions to the seismic analysis of post-tensioned gravity dams.

Many efficient and stable numerical methods have been proposed for dynamic and non-linear problems. In the time domain, implicit dynamic algorithms with unconditional stability are widely used for the time integration, to name a few, the well-known Newmark method [95], HHT-\(\alpha\) method [96], and Bathe method [97]. A wide range of non-linear solution schemes, such as Newton-Rapson method, BFGS (Broyden, Fletcher, Goldfarb, Shanno) method [98], arc-length method [99], have been proposed to solving the non-linear equilibrium equations by iterations.

Despite there are various numerical techniques, dynamic contact problems still present convergence challenges to the nonlinear solution methods. Two requirements should be fulfilled for the solution of dynamic contact problems: i) the contact conditions must be satisfied on the possible contact surface, and ii) the equations of motion should be
precisely time integrated [100]. Two main methods of solutions adopted on the contact modeling are the penalty approach [101] and the Lagrange multiplier method [102, 103]. In the analyses of static contact problems, the Lagrange multiplier method produces very accurate displacement fields [104]. However, for dynamic contact problems employing standard implicit time integrator of the Newmark family, undesirable oscillatory solutions in the kinematic fields (e.g., velocities and accelerations) along the contact surface might be obtained [76]. A significant research effort has been made to propose effective numerical methods for solving the dynamic contact problems efficiently and accurately [105, 106, 107].

1.3 Research aims, methodology, objectives and scope

1.3.1 Research aims

This thesis aims at proposing an efficient framework for the seismic analysis of post-tensioned gravity dams in the time domain, based on the scaled boundary finite element method (SBFEM). In this framework, a rigorous numerical simulation is performed with comprehensive considerations, and the current challenges discussed in Section 1.2 shall be efficiently addressed. A dam-reservoir-foundation system considering the fluid-structure (FSI) and soil-structure (SSI) interactions is essential. The radiation damping conditions of unbounded media should be satisfied. The crack opening and closure at weak structural interfaces in a dynamic analysis, such as dam-foundation interface and construction joints, should be realistically simulated. The sliding resistance of the weak interfaces depends on both cohesion and friction, which should be considered in the simulation. The bond-slip interaction should be taken into consideration in the modeling of post-tensioned anchors (PTA) since it affects the bearing capacity of a PTA system. The mesh generation of post-tensioned gravity dams requires local mesh refinement for the regions where high-stress gradients may exist. Interfacial meshes with matching nodes, i.e., conforming meshes, are preferred for better simulations of the weak interfaces and anchor-structure grouting
interfaces.

1.3.2 Proposed methodology

The scaled boundary finite element method (SBFEM) is utilized to model both the near-field and far-field. The near-field structural (both dam and foundation) domains are discretized as quadtree meshes, and the far-field foundation is represented by an unbounded element. Quadtree meshes offer smooth and rapid transition meshes, which is beneficial for local mesh refinement. The SBFEM is capable of generating polytope elements (S-element) with an arbitrary number of nodes/edges/surfaces [108]. Polygonal S-elements are complementary with the quadtree meshes, and thus the hanging node issue is avoided naturally. An unbounded S-element represents the far-field foundation in terms of displacement unit-impulse response matrices [109]. Radiation conditions of unbounded media can be rigorously satisfied by the unbounded S-element while this element only requires a small truncation area. It captures free-field motions and transfers them as equivalent seismic inputs acting at the near-field/far-field interface. Figure 1.11 shows the S-elements and the coupling of the near-field/far-field using polygonal and unbounded S-elements. The near-field/far-field coupling is reflected by interaction forces indicated as $R_b$ in Fig. 1.11c.

A cohesive-frictional contact modeling scheme is proposed for the modeling of the weak interfaces. This scheme inserts cohesive elements between contact surfaces, which enables that the cohesion and friction simultaneously contribute to the shear resistance. Conforming meshes are utilized for this scheme. Based on the polygonal S-elements, a mesh-conforming procedure converts non-conforming meshes to conforming ones by inserting/shift nodes and split edges at the interfaces only, as shown in Fig. 1.12a. Cohesive elements are generated through the matching node pairs to model the cohesion existing in the weak interfaces. No constraints are required to connect the cohesive elements with the surrounding elements as they are sharing nodes.

The modeling of post-tensioned anchors embedded in the structure is realized in
Figure 1.11. S-elements and their coupling
a fully automatic manner with the availability of polygonal S-elements. Figure 1.12b presents the remeshing procedure to embed two anchors in an existing quadtree mesh, in which the colored nodes are inserted or shifted nodes along the anchor layouts. Those nodes are duplicated, and then, the duplicated nodes are connected to form beam elements representing anchors. Cohesive elements are generated based on the node pairs to simulate the bond-slip interactions at the bond length.

This framework is realized by implementing the polytope and two-dimensional unbounded S-elements in the well-known commercial FEM software ABAQUS as user elements (UEL). ABAQUS offers powerful numerical solution techniques for non-linear and dynamic problems. ABAQUS can also benefit from the salient features of the SBFEM in the mesh generation, anchor modeling, and numerical simulation of the unbounded foundation.

1.3.3 Research objectives

The primary objectives of this research are set in three aspects: modeling, meshing and computer implementation. Comprehensive considerations should be taken for the numerical simulation of post-tensioned gravity dams. A rigorous dam-reservoir-foundation sys-
tem is required to consider the fluid-structure and soil-structure interactions. Interfacial phenomena existing in post-tensioned gravity dams should be simulated properly, such as the bond-slip interactions at the bond length of PTAs and the non-linearity of weak interfaces. Concrete may experience cracking when it is exposed to excessive tensions, and thus damage analysis should be performed. To facilitate the modeling of interfaces, conforming meshes are utilized at the interfaces (both anchor-structure interface and weak structural interface), and related mesh-conforming techniques should be developed. The proposed framework is implemented into commercial FEM software package ABAQUS to take advantages of dynamic nonlinear solver, contact modeling, etc. Detailed explanations of the research objectives are presented as follow:

- **Dam-reservoir-foundation system modeling**: A substructure method [21] is used for the modeling of the coupled system. In this approach, the dam, reservoir and foundation are modeled as three individual substructures. The interaction effects are represented by coupling the substructures at the fluid-structure interface and the soil-structure interface to satisfy the compatibility and equilibrium requirements [21]. Near-field domains of the dam and foundation are discretized as quadtree meshes derived by the SBFEM. At the boundary of the foundation (near-field/far-field interface), an unbounded S-element is constructed to satisfy the radiation conditions absorbing out-going waves. The reservoir domain can be idealized as a finite element system with an artificial boundary [110]. The fluid elements are formulated by the Eulerian approach that takes the water pressures (one DOF per node) as the variables [111].

- **Modeling of interfaces**: The non-linear behavior of the weak joints, such as the dam-foundation interface and construction joints, is simulated through a cohesive-frictional scheme. This scheme inserts cohesive elements at the contact interfaces. Kinetic friction is realized by employing an exponential decay friction model [18]. The bond-slip interaction at the anchor-structure interface is modeled by the cohesive elements connecting the anchor and structure. A quadrilinear model [112] is
chosen as the constitutive law of the cohesive elements in the tangential direction. This model has a relatively simple expression while it keeps the characteristics of the bond-slip relationship.

- **Damage analysis of concrete:** A new Mazars’ model ($\mu$ model) [113] is adopted to simulate the damage of concrete under multiaxial, cyclic and dynamic loadings. Isotropic damage is assumed in this model. A damage variable (scalar value) is evaluated based on the equivalent strain at the scaling center of a polygonal S-elements. The unilateral behavior of concrete (crack opening and closure) is considered, which is essential for the numerical simulations of concrete structures under seismic loadings.

- **Automatic mesh-conforming algorithms:** These algorithms are developed based on the polygonal S-elements. Conforming meshes can be obtained in a fully automatic manner. Surface conforming meshes, i.e., meshes with matching nodes and edges, are suitable for the cohesive-frictional scheme. They provide interface representations without initial gaps or penetrations for contact problems. The domains in contact can be discretized individually, and then the existing meshes at the interfaces are re-meshed to be conforming. Nodal conforming meshes, i.e., meshes with matching nodes only, are utilized for the PTA modeling. Again, the structural domain discretization is independent of the anchor layouts, and then they are re-meshed to insert the PTAs. It is achieved by inserting nodes into existing structural meshes along the anchor layouts, duplicating the inserted nodes, and then connecting the duplicated nodes to form beam elements representing anchors.

- **ABAQUS implementation:** The polytope and two-dimensional unbounded S-elements are implemented in the well-known commercial FEM software ABAQUS as user elements (UEL). ABAQUS offers powerful functions facilitating the seismic analysis, such as a robust non-linear dynamic solver (parallel computing), advanced element library (acoustic elements, cohesive elements, etc.), and comprehensive
contact modeling techniques. Applications of ABAQUS built-in functions/elements utilized in this work include: i) surface-to-surface contact modeling for weak interfaces; ii) cohesive elements for both weak structural interfaces and anchor-structure grouting interface; iii) beam elements for the anchor modeling; iv) acoustic elements and the non-reflecting acoustic boundary for the modeling of the fluid domain; v) dynamic, implicit solver for time integration. Furthermore, polytope S-elements reduces the meshing burdens of inserting anchors in standard finite elements and generating conforming meshes. The performance of ABAQUS’ contact modeling is improved with the availability of surface conforming meshes.

1.3.4 Scope of the work

The proposed framework is efficient to predict the seismic time-history responses of post-tensioned gravity dams. Firstly, it features the flexibility in mesh generation and the fidelity of unbounded domain modeling. Polygonal S-elements facilitate the local mesh refinement and generating interfacial conforming meshes. Unbounded S-elements only require a small truncation area to capture the dynamic properties of the unbounded foundation, which reduces the size of near-field foundation and further improves the computational efficiency. Moreover, the interfacial characteristics existing in post-tensioned gravity dams (as presented in Section 1.2.2) are rigorously simulated. The effects of dam-foundation interface and construction joints on the stability against overturning and sliding, and the effects of bond-slip phenomena on the bearing capacity of PTA systems and the seismic responses of dams can be studied. On the other hand, the permanent sliding occurring at the weak interfaces, and the possible pull-out failure of PTA systems can be monitored. Last but not least, the modeling of dams strengthened by multiple inclined post-tensioned anchors can be easily realized through the automatic mesh-conforming techniques. The mesh generation of dam and foundation is independent of the anchor layouts. The multiple inclined anchors are automatically inserted into the existing structural meshes through inserting nodes into polygonal S-elements along the anchor layouts.
The framework is implemented in ABAQUS to facilitate the use of this work to scholars. Three-dimensional seismic analysis of dams is not contained in this thesis although polyhedral S-elements have been implemented as well. The simulations are carried out on ABAQUS/Standard 2017. The utilized functions of ABAQUS are briefly explained. However, the detailed studies of those topics are beyond the scope of the present thesis.

1.4 Organization of the thesis

The remainder of this thesis is organized as follows.

- In Chapter 2, a literature review is presented, which targets on the topics related to the objectives of this thesis.

- In Chapter 3, the fundamental formulations of the scaled boundary finite element method in three-dimensional (polyhedral element) are derived, and the unbounded S-element in conjunction with its related seismic input model for the two-dimensional case is provided.

- In Chapter 4, the SBFEM user elements are implemented into ABAQUS, followed by several benchmarks to verify the implementation for both polyhedral and unbounded elements.

- In Chapter 5, the modeling of interfacial problems is studied, in which proper modeling schemes are proposed and the automatic mesh-conforming algorithms are developed.

- In Chapter 6, the seismic analysis of post-tensioned gravity dams is performed on the proposed framework based on the availability of SBFEM user elements and the ABAQUS platform.

- In Chapter 7, a regularized concrete damage model is introduced on top of the polygonal user elements, and damage analysis of post-tensioned gravity dams is
carried out.

• In Chapter 8, the conclusions drawn from this work are stated, and recommendations for future work are presented.

1.5 List of publications

1.5.1 Journal papers


2. Ya, S., Eisenträger, S., Qu, Y., Zhang, J., & Song, C. (2023), Seismic analysis of post-tensioned concrete gravity dams using scaled boundary finite elements implemented as ABAQUS UEL. (Accepted by Soil Dynamics and Earthquake Engineering)

1.5.2 Conference proceedings


Chapter 2

Literature review

This chapter presents a literature review on the numerical approaches for the seismic analysis of post-tensioned concrete gravity dams. In Section 2.1, the literature on seismic analysis of dam-reservoir-foundation interaction systems is presented, including the modeling procedures, the fluid-structure and soil-structure interactions, and the seismic input mechanisms. In Section 2.2, the modeling of the characteristics in post-tensioned concrete gravity dams is discussed. It contains the weak joint modeling and the post-tensioned anchor modeling. Interfacial problems are involved in these two features, as discussed in Section 1.2.2. Mesh generation techniques to obtain conforming meshes facilitating the interfacial modeling are reviewed in Section 2.3. In Section 2.4, damage model for concrete is summarized as the concrete cracking significantly affects the seismic performance and damage analysis is often involved in the numerical modeling of dams. Finally, conclusions of this chapter are drawn in Section 2.5.

2.1 Seismic analysis of dam-reservoir-foundation systems

As discussed in Section 1.2.1, in seismic analysis of dams, a system including the dam, reservoir, and foundation regions (dam-reservoir-foundation system) should be modeled to take the fluid-structure (FSI) and soil-structure (SSI) interactions into considerations.
2.1.1 Modeling procedures

The modeling procedures of dam-reservoir-foundation systems can be classified into two distinguishing methods: the substructure method [21] and the direct method [114]. Figure 2.1 depicts the finite element models of a gravity dam through these two procedures.

In the substructure method, the coupled system is modeled as three substructures, i.e., the dam, the reservoir, and the foundation domains [21]. The dynamic properties of each substructure is analyzed individually. The interaction effects are represented by coupling the substructures at the fluid-structure interface and the soil-structure interface to satisfy the compatibility and equilibrium requirements [21]. The substructure method essentially divides the coupled system into near-field and far-field systems. Near-fields contain bounded domains, such as the structure, while far-fields are simplified as unbounded domains, such as the reservoir and foundation that extend to infinity. The reservoir domain can be idealized as a simple prism to allow the use of analytical solutions [115], or a finite element system with an artificial boundary to model complex geometry [110]. The modeling of the foundation domain can also be realized by simplifying it as a homogeneous continuum and using simple solution procedures [116] or a combination of finite element and continuum systems to model non-homogeneous foundations with complex geometry [117]. The adjacent reservoir and foundation in finite elements belong to the near-field. In this thesis, the analysis of post-tensioned concrete gravity dams involves ground anchors that extend into the foundation. The adjacent foundation can be conveniently modeled as finite elements allowing the modeling of anchor-structure interaction. The supporting foundation (finite elements) is regarded as a part of the structure [60]. Earthquakes usually excite in the far-field of the foundation and transfer into the near-field in terms of seismic waves. The waves should be formulated as seismic forces acting on the soil-structure interface (near-field/far-field interface).

The direct method models the dam-reservoir-foundation system as a single composite system [7]. All three domains can be discretized as finite elements. The unbounded
Figure 2.1. Modeling procedures of dam-reservoir-foundation systems [7]
regions of the reservoir and foundation can be represented by viscous damper boundaries [65]. There are also simplified procedures readily for the modeling of the reservoir domain (such as Westergaard’s added mass approach [61]) and the foundation domain (such as the massless foundation model [62]). The foundation needs to be extended a distance in the upstream, downstream, and downward directions. The extended lengths should be at least equal to the dam height [7]. There are also researches that suggest a distance of three times the dam height for an accurate simulation [118]. Detailed explanations of the Westergaard’s added mass approach and massless foundation model will be explained in Section 2.1.2 and Section 2.1.3, respectively. The seismic input is applied at bottom of the foundation in terms of accelerations $a_x^g$ (horizontal) and $a_y^g$ (vertical).

In this thesis, the substructure method is utilized for the simulation since the effects of the FSI and SSI are more accurately simulated by this method [7]. The adjacent reservoir and foundation of the dam are modeled as finite elements (near-field). The near-field reservoir enables us to consider the effects of inclined upstream on the FSI. A near-field foundation is necessary for the modeling of PTAs. The far-field of the foundation is realized by a viscoelastic half space derived by a scaled boundary finite element approach. Artificial boundary is constructed at the far end of the reservoir.

### 2.1.2 Fluid-structure interaction

Hydrodynamic pressures are generated in the fluid during seismic events, which further affects the responses of the structure. The numerical simulation for the FSI can be generally classified into three approaches [59]: Westergaard, Lagrangian, and Eulerian.

The Westergaard technique assumes the water as in-compressible and the reservoir is of constant depth. The governing differential equations are solved analytically. This method simply represents the reservoir domain by means of added mass at the dam-reservoir interface [61], and thus, it is also called added mass approach. The added mass of a point at the dam-reservoir interface is derived on the basis of the analytical solution,
expressed as:

\[ M(y) = \frac{7}{8} \rho_w \sqrt{h_r(h_r - y)} A, \]  

(2.1)

in which \( \rho_w \) is the water density, \( h_r \) is the height of the reservoir, \( y \) is the distance of the point to the reservoir bottom, and \( A \) is the tributary area of the point from its immediate neighboring elements. Due to its simplicity, this technique has been widely employed for decades in dam engineering. However, the dam flexibility and water compressibility [119, 59] are neglected. This method only provides acceptable results in the range of restricted hypotheses [120, 119].

The Lagrangian approach and Eulerian approach involve the reservoir domain discretization. The main difference between these two methods is the unknown variables of the fluid domain. In the Lagrangian approach, displacements are the variables of fluid elements, while in the Eulerian approach, pressures or velocity potentials are the variables of fluid elements [111]. The Lagrangian approach formulates the fluid elements using the same shape functions as solid elements. Consequently, no special treatment is required to satisfy the compatibility at the fluid-structure interface, which makes the implementation of the Lagrangian approach convenient. However, the displacement-based fluid elements have zero shear modulus to model the inviscid flow (inter-laminar motion), resulting in numerical problems such as zero-energy modes [121, 122, 123, 124]. By contrast, the fluid elements formulated by the Eulerian approach have a single unknown per node, and then, rotational motion is inherently avoided resulting in no locking behavior [119]. A single DOF saves computational resources as well. Since the variables in fluid and structure domains are different, special treatments should be taken at the fluid-structure interface to enforce compatibility conditions. By assuming water to be a compressible, inviscid and linear medium, the hydrodynamic pressure in the reservoir is governed by a wave equation [125]:

\[ \nabla^2 p = \frac{1}{c_w^2} \frac{\partial^2 p}{\partial^2 t}, \]  

(2.2)
in which \( p(x,y,t) \) is the hydrodynamic pressure, \( c_w \) is the velocity of pressure wave, and \( \nabla^2 \) is the Laplace operator. At the top of the reservoir (free surface), the hydrodynamic pressure is usually assumed to be zero \( (p = 0) \) [126, 127], neglecting the effects of gravity waves. At the fluid-structure interface, compatibility between hydrodynamic pressure (fluid) and normal displacements (structure) should be satisfied:

\[
\frac{\partial p}{\partial n} = -\rho \ddot{u}_n, \tag{2.3}
\]

where \( n \) denotes the outward normal, \( \rho \) is the density of water, and \( \ddot{u}_n \) is the absolute acceleration in the normal direction of the dam–reservoir interface. At the far end of the reservoir (unbounded reservoir), artificial boundary conditions are required to satisfy the radiation conditions [67, 68], as discussed in Section 1.2.1. A Sommerfeld boundary condition [128], which is also the simplest approximation [129], is commonly used to fulfill this requirement in the literature [130, 131, 132]. For a fluid-structure interaction problem under horizontal ground excitation, the boundary conditions to prevent reflected waves from the upstream of the dam is satisfied by:

\[
\frac{\partial p}{\partial n} = -\frac{1}{c_w} \frac{\partial p}{\partial t}. \tag{2.4}
\]

The Eulerian approach is more often utilized in numerical analyses involving the FSI [21, 133, 111, 134, 135, 136], and it is suitable to the substructure method.

### 2.1.3 Soil-structure interaction

The flexibility of the soil that supports the structure affects the earthquake response of the structure by lengthening the period of vibration. Besides, the effective damping of the soil-structure system are increased due to the energy radiation and material damping of the foundation region [7]. Thus the soil-structure interaction should be considered in the seismic analysis of dams.

A simple massless foundation model [62] can be employed to consider the flexibility
of the soil. In this model, a portion of the foundation is discretized as finite elements. The foundation region is assumed to have a rigid horizontal boundary, where the earthquake excitation is applied. However, the outgoing waves, i.e., waves radiating away from the structure, should be avoided from reflecting again from the boundary of the foundation. A massless foundation model cannot realistically simulate this phenomenon.

To overcome the limitations of the massless foundation model, artificial boundaries (non-reflecting boundaries, transmitting boundaries, or absorbing boundaries) [137] are usually constructed to absorb the outgoing waves. Depending on the elements used at the soil-structure interface and their formulations, the artificial boundary conditions (ABC) can be generally categorized into two types: discrete and continuum [138, 139].

Discrete modeling usually utilizes springs and dashpots at the boundary of the foundation. The properties of the spring and dashpot elements are obtained based on the neighboring nodes, i.e., a tributary area [137, 140, 70, 141]. The formulations of discrete modeling contain Winkler model [142], Pasternak foundation [143], Kerr foundation [144], etc [145, 146]. High-order discrete models have been proposed to improve the accuracy by establishing paraxial boundary conditions [147], Bayliss-Turkel boundary condition [148], or introducing auxiliary variables [149]. However, the discrete modeling often requires a sufficient foundation size, resulting in a large number of degrees of freedom.

Continuum approach is widely used in the substructure method. It models the infinite (unbounded) foundation as half-space using either boundary element (BEM) [150] or scaled boundary finite element methods (SBFEM) [151, 114]. Essentially, this technique derives a dynamic stiffness (impedance) matrix for the unbounded foundation, and then, the interacting forces at the soil-structure interface are the product of the displacements and the dynamic stiffness matrix [117, 23, 7]. In this method, the near-field foundation is allowed to have an arbitrary shape [60] and it can be limited to a small size [23]. This feature significantly reduces the computational resources [66]. Compared to the BEM, the SBFEM requires no fundamental solution for the governing equation, and can be
seamlessly combined with the conventional FEM [152, 89].

2.1.4 Seismic input models

In a time history dynamic analysis, the earthquakes are usually converted to equivalent seismic loads applied on the structure. Various seismic input models have been proposed, such as rigid-base input model [153], the massless foundation model [154], the deconvolved base-rock input model [155], and the free-field input model [156, 157]. Depending on the foundation model, different input mechanisms should be adopted.

In the direct method, the massless foundation model is often utilized due to its simplicity [158]. This model is essentially the same as the rigid-base input model. These two models apply the free-field accelerogram time history at the base of the foundation. Compared to the rigid-base input model, the earthquake excitation will propagate instantaneously to the base of the dam in the massless foundation model [158]. This feature eliminates the potential artificial amplification of the accelerogram; however, it neglects the energy-loss mechanisms attributed to the outgoing waves [159]. A massless foundation model, including the material damping, can be used to consume the seismic energy that should be transmitted to the infinity [160].

In the substructure method, the seismic excitation is generally represented by the free-field motion of a half-space (unbounded foundation) [161, 162]. The most common seismic waves, vertically propagating shear waves, are derived as equivalent external forces acting on a semi-infinite elastic foundation by Joyner and Chen [163]. Zhao et al. developed a numerical model for the wave scattering problems in infinite media due to incident compressive and shear waves [164]. Bazyzer [165, 166] proposed the free-field input model of wave scattering problems based on the SBFEM. Domain reduction method (DRM) is developed to uncouple the seismic loading process and the unbounded domain modeling [167, 168]. In this method, the seismic excitation is applied over a single layer of elements at the boundary of the foundation.
2.2 Interfacial problems of post-tensioned gravity dam

The characteristics of post-tensioned concrete gravity dams should be modeled realistically, such as the interfacial problems. Two types of interfacial problems are involved in this thesis: i) the weak interfaces, such as the dam-foundation interface and construction joints, and ii) the bond-slip interaction at the anchor-structure grouting interface. This section reviews the numerical modeling techniques for these two interfacial problems.

2.2.1 Modeling of weak interfaces

The shear resistance of the dam-foundation interface is what a gravity dam mainly relies on to prevent sliding. Besides, the concreting of a large gravity dam usually introduces construction joints. These interfaces are relatively weak compared to the concrete blocks [4]. The sliding of a gravity dam is likely to occur at such weak interfaces [169]. Fronteddu et al. [8] investigated the dynamic responses of concrete gravity dams with construction joints by finite element analysis (FEA), and observed residual displacements caused by the sliding at the weak interfaces, as illustrated in Fig. 2.2. The sliding response mainly depends on the condition of the construction joints. The existence of weak construction joints can reduce the sliding at the base of the dam [169]. The permanent sliding can be estimated by upper bound approaches, i.e., analytical procedures based on either rigid structure [170] or flexible structure [171] assumptions. Non-linear time-history dynamic analyses have been also used to determine the permanent sliding [172, 173, 8, 174, 73].

The shear strength of a weak interface depends on both cohesion and friction [4]. Mohr-Coulomb failure criteria [175] is often adopted as the constitutive law model representing the joint behavior in sliding [172, 173, 8]. This constitutive model formulates the shear resistance $\tau_r$ by

$$\tau_r = c + \sigma \tan \phi,$$

in which $c$ represents the cohesion, $\sigma$ is the normal stress, $\phi$ is the friction angle, $\tan \phi$
is the coefficient of friction. The Mohr-Coulomb parameters \( (c, \phi) \) measured for existing dams have been widely reported in the literature \([4, 176, 177]\). These parameters are constant values in the Mohr-Coulomb model. However, the cohesion varies along the interface because the contact conditions change during sliding \([178]\). In a dynamic problem, the contact conditions, direction and velocity of motion, and the magnitude of the ground excitation affect the coefficient of friction \([179]\). Improved Mohr-Coulomb models have been proposed to consider the nonlocal and nonlinear character of friction \([180]\) and the dynamic friction phenomena \([181]\). The elasto-plastic model has also been used to reflect the degradation of the cohesion \([182, 183]\).

The numerical modeling of the weak interfaces can be classified into two approaches: friction model \([172, 173, 73]\) and joint element \([8, 174]\). In the friction model, kinematic constraints between two contact blocks should be satisfied. Contact pressures and friction forces are applied at each side of the interface. Common utilized joint elements are the spring element and interface element. A spring element represents a certain tributary area of the interface, and it connects two nodes at each side of an interface, as shown in Fig. 2.3a. Gap-friction element \([184, 8]\) is an improvement of the spring element. This element consists of parallel spring and dashpot elements in both normal and tangential directions; thus, the shear resistance’s cohesion and friction can be considered simultaneously. Zero-thickness \([91]\) or thin layer interface element \([92]\) have also been used.
to model the joint behavior [174, 185]. Compared to the spring element, the interface element has a length in the tangential direction. A classical interface element (liner) connects two pairs of nodes at each side of an interface, as shown in Fig. 2.3b. The interested displacement fields of the interface elements are on the upper and lower sides and are obtained by linear interpolation.

2.2.2 Modeling of PTAs

The post-tensioned anchors were first modeled as two equal and opposite concentrated loads at the two ends of the free length, i.e., equivalent force method [186, 187]. This approach is simple as it has no requirement for the meshing of the PTAs. The applied loads representing the prestressing should consider the loss due to the anchor-structure interactions: i) friction, ii) bond-slip, ii) relaxation, and (iv) deformations (short- and long-term) of the structure [186]. However, the variation of the cable forces in the dynamic analysis is neglected. The potential PTA failures, such as the tendon tensile/shear failure and pull-out failure, can not be examined.

Rigorous PTA modeling usually involves truss/beam elements representing the anchors [187, 58, 26, 188, 189]. The truss/beam elements are assigned with initial temperature or stress to simulate the prestressing. The ends of the truss/beam elements, i.e., the anchor head and the end of the free length, are constrained to the structure. In this method,
the stresses in the anchors change due to the oscillation of the structure in a dynamic analysis. Morin [26] and Corn [188] have successfully applied this approach to investigate the PTA failure due to shear. Nonetheless, this method essentially assumes a perfect bond for the bond length. The bond-slip interaction at the anchor-structure grouting interface cannot be simulated, despite the fact that the pull-out failure is deemed to be critical to the capacity of a PTA system in many applications [55, 17, 2].

The bond-slip interaction has been often considered in the numerical simulation of reinforced concrete [190, 191, 192, 193]. Similar to the modeling of the weak interfaces, spring [90] or interface elements [91] are used to simulate the bond-slip interaction. The spring element proposed by Ngo and Scordelis [90] connects one anchor node with one corresponding structure node to represent the bond. Restrictions have been found by using spring elements for the PTA modeling: i) the FE mesh should guarantee that the anchor is located along the edges of the solid elements, and ii) double nodes are necessary to reflect the relative displacement (slippage) between the anchor and structure. The interface element is similar to the spring element, connecting the anchor and structure node pairs. A conventional two-dimensional interface element consists of four nodes (in two adjacent pairs), and its displacement field along the length is obtained by linear interpolation. Rots [194] proposed a new two-dimensional quadratic interface element that contains nine nodes: three anchor nodes and six structure nodes (three at each side of the anchor), as shown in Fig. 2.4b. This interface is special as it incorporates the anchor and structure into one element. Degroot et al. [195] developed a three-dimensional axisymmetric interface element, based on which Rots [194] extended his quadratic element into a three-dimensional version consisting of 15 nodes (three for the anchor and 12 for the structure). Again, the utilization of interface elements involves the difficulty of mesh generation as the meshes of the structure should provide corresponding nodes featuring the anchor layout. To get rid of these restrictions, Kwak and Filippou [87] proposed an embedded model, as shown in Fig. 2.4a. In this model, the truss/beam element is assumed to be embedded inside the structure element, which allows an independent choice of struc-
ture mesh. The DOFs of the structure that are associated with the anchor are condensed out before they are assembled. However, the embedded formulation is complicated, and more computational effort should be involved [187].

The constitutive law of bond-slip models should consider the adhesion, mechanical interlocking, and friction between the rock and grouting [78, 2]. It depends on many variables, such as grout mix, rock type, rock strength, borehole roughness, and diameters [17]. Two analytical models have been widely used in the reinforced concrete: the Bertero-Popov-Eligehausen (BPE) model [6] (see Fig. 2.5a) and the tri-linear model [78] (see Fig. 2.5b). A quadrilinear model [112] (see Fig. 2.5c) was proposed by Woods and Barkhordari for the ground anchor to simulate the bond-slip behavior at the rock-grout interface. This model is more straightforward compared to the BPE model, and it allows a yielding plateau comparable to the trilinear model. The parameters of the BPE and tri-linear models are determined from the experimental results of pull-out tests on short embedded specimens in the laboratory, which guarantees the bond stresses are nearly uniform at the interfaces. However, pull-out tests of grouted rock anchors reported in the literature [17, 112, 196] are usually small-scale or full-scale field investigations and only provide pull force-displacement data. The bond stress distribution is markedly non-uniform along the bond length while tending to be exponentially decayed from the proximal end to zero at the distal end [16, 81]. In this case, the local bond-slip relationship can not be straightforwardly obtained based on the pull force-displacement relationship. Liu et al. [80] developed a theoretical analysis procedure to convert the pull force-displacement to a local bond stress-slip relationship.

### 2.3 Meshing techniques

Mesh generation is vital for obtaining accurate results in numerical analysis. Generally speaking, finer and denser meshes result in more accurate results, but the simulations become more extensive and take longer computational times. Therefore, an ideal mesh
Figure 2.4. Modeling of bond-slip interaction
should be characterized by varying mesh sizes: smaller elements for regions with high stress gradients and larger elements for regions with small stress gradients. The meshes in the transition zones might be distorted [82], especially for quadrilateral elements, as presented in Section 1.2.3. Besides, the mesh generation for post-tensioned concrete gravity dams involves the features of weak interfaces, such as the dam-foundation interface, construction joints, and anchor-structure grouting interfaces. Generating node-to-node matching meshes at those interfaces (interfacial conforming meshes) will facilitate the simulations [86, 88, 12]. The review in this section emphasizes the meshing techniques to generate meshes with smooth transition elements, and conforming interfacial meshes.
2.3.1 Smooth transition meshes

In the framework of conventional FEM, especially using quadrilateral elements, generating meshes with significantly varying sizes can be realized by local mesh refinement [197, 198, 199, 200] or coarsening [201, 202, 203, 204]. No matter which technique is adopted, the transition elements that connect the finer and coarser areas of the mesh should be smoothed to avoid distorted shapes. The strategies of generating smooth transition elements can be generally classified into smoothing [205, 206, 83] and remeshing [197, 198, 82, 199, 200]. In the framework of the smoothing strategy, the nodal connectivity of the elements in an existing mesh is not changed, and the nodes of the meshes are repositioned to obtain elements with good shapes. The Laplacian smoothing technique [205] is the simplest and most widely used local technique. In this method, an internal node is repositioned to the centroid of the polygon formed by its adjacent nodes. However, invalid meshes or elements of worse quality might be generated since the repositioning operation is heuristic [207]. Improvements for Laplacian smoothing have been proposed by replacing the polygon with surrounding elements [208], or combing this algorithm with mesh optimization [207]. The remeshing strategy involves changes of the node connections. There are various remeshing algorithms have been proposed, such as edge swapping [208, 209], prescribed transition elements (patterns) [82], and element elimination [208]. Edge swapping is a well-known technique. In this method, an interior edge adjacent to two quadrilateral elements can be swapped if the swapping results in elements with higher qualities. Figure 2.6 depicts the schematics of Laplacian smoothing and edge swapping.

Quadtree-based mesh generation technique [210] shows excellent advantages in generating meshes with large gradients of mesh size. This technique recursively divides the geometries into squares. The size ratio between two adjacent quadtree meshes can be equal to 2. Square shapes naturally avoid ill-conditioned meshes. These features guarantee that the mesh transition in quadtree meshes is fast and smooth. To obtain boundary-conforming meshes, the quadtree meshes can be trimmed by the boundary [210], or the
nodes can be shifted onto the boundary curve [84]. However, the quadtree meshes have node numbers varying from four to eight, resulting in hanging node issues. They can not be straightforwardly used in the standard FEM. The quadtree meshes should be modified to be triangular or quadrilateral elements to be suitable for the standard FEM. Triangulation is a technique that divides the quadtree meshes into triangular elements [84]. This method is simple and robust, and thus, it has been widely used to combine the quadtree-based mesh generation technique with the standard FEM [211, 84, 212, 213]. Nonetheless, triangular elements are constant strain elements, which might reduce the accuracy compared to quadrilateral elements. All-quadrilateral element mesh can be obtained by introducing quadtree transition patterns [211, 214]. In this approach, quadtree meshes having more four nodes are spitted into several quadrilateral shapes. Smoothing is often involved in improving the gradation and element shapes [211]. Figure 2.7 shows the modified quadtree meshes suitable for the standard FEM by employing triangulation and quadtree transition patterns.

Alternative finite element methods that support polygonal elements can straightforwardly use the quadtree meshes, such as scaled boundary finite element method (SBFEM) [22],
smoothed finite element method (SFEM) [215], virtual element method (VEM) [216], Voronoi cell finite element method (VCFEM) [217]. In the recent years, the quadtree meshes have been successfully applied in the SBFEM to study problems involving complex mesh transitions, to name a few, adaptive analysis [218], fracture mechanics [219], damage analysis [220], wave propagation [166], poroelasticity [221]. The combination of the SBFEM and quadtree mesh technique is a promising approach for the modeling of post-tensioned concrete gravity dams.

2.3.2 Constructing conforming meshes from non-conforming meshes

For the numerical analysis involving multiple subdomains, it is practical to generate the meshes of each subdomain independently. At the interfaces of these subdomains, non-conforming (not node-to-node matching) meshes might be generated. Various methods have been developed for the coupling of non-conforming meshes, such as nearest neigh-
Figure 2.8. An example of mesh matching on hexahedral meshes by pillowing sheet and dicing column [9]

stere interpolation [222, 223], Mortar element method [224], Nitsche’s method [225]. In general, these methods establish interface constraints or special shape functions to fulfill the compatibility at the interfaces, and thus additional unknowns are often introduced. By creating conforming meshes, the requirement for constraint conditions can be eliminated, resulting in straightforward and more precise numerical solutions [9].

Staten et al. [9, 226] developed comprehensive mesh matching algorithms on hexahedral meshes. Local mesh modifications are made at the non-conforming interfaces to create a node-to-node mapping between the nodes and quadrilaterals. His method is based on a theory of hexahedral mesh dual [227], which describes the hexahedral elements by groups of sheets or columns. The interfacial meshes at one side (Surface A) are distinguished into different sheets first. Along the path of each sheet on Surface A, the meshes of the other side (Surface B) are modified by inserting (pillowing [228] or dicing [229]) or extracting quadrilaterals [230] to be conforming with Surface A. Accordingly, the columns formed by those quadrilaterals are expanded (related to inserting) or collapsed (related to extracting). Figure 2.8 depicts an example of mesh modification for one sheet by pillowing the sheet and expanding the column. This method moderately increases the number of elements.

Polytope elements (polygon in two-dimensional and polyhedron in three-dimensional) offer a higher degree of flexibility as they have arbitrary numbers of nodes, edges, and surfaces. They are ideal candidates for converting non-conforming meshes to conforming ones. The interfacial meshes can be modified to be matching by inserting nodes, dividing edges, or splitting surfaces. Sohn et al. [10] utilized the polyhedral elements
Figure 2.9. An example of mesh matching on polyhedral meshes [10] based on the SFEM [215] to deal with non-matching interfaces in domain decomposition problems. In his approach, additional nodes are inserted at the intersection points of two non-conforming meshes to convert them to be node-to-node matching. Figure 2.9 demonstrates an example of two contact blocks with non-conforming interfacial meshes. Zhang et al. [231] introduced an optimization operator to improve the surface mesh quality. By only inserting additional nodes, distorted polyhedral elements (short edges) might be obtained if some nodes and edges on one surface are close to those on the other surface. The optimization shifts the close nodes to eliminate the potential short edges. The mesh matching technique proposed by Zhang et al. has been successfully employed in domain decomposition [231], contact [86, 88], acoustic-structure interaction [232], and cohesive interface [89] problems based on the SBFEM [22].

2.3.3 Anchor inclusions

In FEM the anchors are generally modeled as one-dimensional tension truss or beam elements in post-tensioned concrete gravity dams [187, 233, 234, 188]. To establish proper interaction between the anchors and structure, the mesh of the structure usually needs to be aligned with the anchors. Partitioning of geometry along the anchor layouts is required prior to the mesh generation, which increases the burden on numerical analysts. Besides, multiple inclined anchors requiring irregular partitioning may result in distorted quadrilateral shapes.

To circumvent the meshing compliance requirement mentioned above, several approaches have been developed to automatically embed truss/beam elements into solid ele-
ments. The embedded model [87] discussed in Section 2.2.2 allows an independent choice of concrete mesh. Its formulation is based on the idea of evaluating the anchor's stiffness individually in the concrete element in conjunction with isoparametric shape functions. A so-called projected fiber approach [235] does not require modeling the anchor explicitly; it considers the geometry and mechanical properties of the anchors and projects the DOFs of the fictitious truss elements on the triangular elements. This method was extended by Zouari et al. [236] to use beam elements and quadrilateral elements. However, a perfect bond between the anchor and concrete is assumed in the projected fiber approach. The stress development of anchors can not be monitored. Pike and Oskay [11, 237] proposed enriched elements based on the extended finite element method (XFEM) [238]. The original solid elements that intersect with the anchors are replaced by fully enriched elements (internal anchor segments) or fully enriched fiber tip elements (anchor tips), as shown in Fig. 2.10. A partial enrichment function formulates the elements adjacent to the fully enriched elements. The progressive debonding between the anchor-structure interfaces can be considered. This method is capable of simulating composite materials that include a large number of randomly spread short fiber inclusions; however, it neglects the intersections between different inclusions in two-dimensional problems.

Zhang et al. [12] proposed an anchor inclusion algorithm in polytope elements. At the intersection points between the anchor layouts and the edges of existing meshes, additional nodes are inserted into the original meshes. At the anchor tip, the elements are divided into several elements by the lines connecting the anchor tip and mid-edge points of the meshes. Figure 2.11 demonstrates the situations of anchor inclusion for one anchor, two anchors, and one anchor tip, in which the solid red squares are the inserted nodes. The initial nodes close to the anchor layouts are shifted onto the layouts to avoid distorted elements. Beam/truss elements representing anchors can be formed directed by connecting the inserted nodes along the layouts. This method is easy to implement owing to the polygonal elements allowing arbitrary numbers of nodes and edges. Compared to the methods discussed in the previous paragraph, this approach has the following advan-
tages:

1. No special treatment is required for the compatibility as the beam/truss elements share nodes with the concrete elements;

2. Anchors have been modeled explicitly, and thus, the stress variations in dynamic analysis can be examined;

3. The intersection of anchors is considered.

This algorithm can be modified by duplicating the inserted nodes. In this way, the beam/truss elements are separated from the concrete elements. Bond-slip interaction can be simulated by inserting interface elements that connect the anchors and structure.

2.4 Damage analysis of concrete

Concrete is considered a quasi-brittle material that will experience cracking when it is exposed to excessive tensions. Progressive loss of material integrity at the macro-scale is generated due to the crack growth. Therefore, the seismic cracking simulation of concrete is often involved in numerical analysis of concrete dams [239, 240, 58, 241]. The
cracking simulations are usually carried out by either fracture mechanics (FM) [239, 241] or continuum damage mechanics (CDM) [242, 240]. In this thesis, the CDM is utilized since it is capable of predicting the development of fractures without setting pre-existing defects [243]. The CDM treats a discrete fracture as diffused micro-cracks in a localization zone. Damage variables are used to describe the deterioration of the mechanical properties of the concrete. In this section, the constitutive damage models of concrete are reviewed, followed by the regularization techniques to prevent mesh sensitivity of FE simulations. The plasticity and anisotropy of concrete [244, 245] are not included in this context.

2.4.1 Damage constitutive models

The constitutive relationship of concrete refers to the relation between the internal stress and strain of concrete under external action. It describes the basic mechanical properties of concrete at a specific point, i.e., at the micro-scale. Kachanov [246] first introduced a continuous variable related to the density of the micro-cracks, i.e., a damage variable $\omega$ to represent the average material degradation in a representative volume element (RVE) [247]. This variable can be assumed as a scalar value with an assumption of isotropic damage [248, 249]. It is incorporated in the stress-strain relation of concrete at the macro-scale by

$$\sigma = (1 - \omega)D\varepsilon,$$

Figure 2.11. Anchor inclusion in polygonal elements [12]
2.4.1.1 Evolution laws

There are a variety of softening models defining the evolution laws of quasi-brittle materials, to name a few, linear [13], bi-linear [250], tri-linear [251], exponential [252], and Mazars’ [253, 254] models. Three commonly used damage evolution laws are introduced in detail, i.e., the linear softening model, modified exponential softening model, and Mazars’ model. They are all expressed in terms of equivalent strain. The tensile stress-strain relationships of these three models are compared in Fig. 2.12.

In the linear softening model, the tensile stress linearly decreases during the damage
progress. The evolution law is given as

$$\omega_t(\kappa) = \begin{cases} 
0 & \kappa \leq \varepsilon_0 \\
\frac{\varepsilon_0}{\varepsilon_f - \varepsilon_0} (1 - \frac{\varepsilon_0}{\kappa}) & \varepsilon_0 < \kappa \leq \varepsilon_f \\
1 & \kappa > \varepsilon_f \end{cases}$$

(2.7)

where $\omega_t$ is the tensile damage variable, $\kappa$ denotes the maximum historical equivalent strain (in tension), $\varepsilon_0$ is the threshold of the damage initiation, and $\varepsilon_f$ is the equivalent strain related to the complete failure. The mathematical expression of this model is simple, but its shape has substantial deviation with the experimental (uniaxial tension) result [254].

The tensile stress exponential decreases with the equivalent strain in the exponential softening model. The damage growth of this model is expressed as [252]

$$\omega_t(\kappa) = \begin{cases} 
0 & \kappa \leq \varepsilon_0 \\
1 - \frac{\varepsilon_0}{\kappa} \exp\left( -\frac{\kappa - \varepsilon_0}{\varepsilon_f - \varepsilon_0} \right) & \kappa > \varepsilon_0 \end{cases}$$

(2.8)

The parameter $\varepsilon_f$ affects the ductility of the damage evolution and governs the fracture energy of the concrete. Compared to the linear softening model, the exponential softening model is capable to bear large strains without resulting in non-zero stiffness matrix.

The Mazars’ model is similar to the exponential softening model featuring an exponentially decreasing stress. In addition, it includes a residual stress (crack bridging) of $(1 - \alpha_t)\sigma_0$ ($\sigma_0$ is the maximum tensile stress), which again enhances the non-singularity of the stiffness matrix. The damage evolution law in Mazars’ model is formulated as [253]

$$\omega_t(\kappa) = \begin{cases} 
0 & \kappa \leq \varepsilon_0 \\
1 - \frac{\varepsilon_0}{\kappa} (1 - \alpha_t) - \alpha_t \exp(\beta_t \varepsilon_0 - \beta_t \kappa) & \kappa > \varepsilon_0 \end{cases}$$

(2.9)

in which $\alpha_t$ is the parameter defining the residual stress, $\beta_t$ determines the shape of
the effective damage evolution. A higher value of $\beta_t$ results in a rapider softening, i.e., more brittle response. This model is appropriate for the simulations of both tensile [255] and mixed tensile-shearing failures [256]. Besides, the Mazars’ model defines a compressive damage variable $\omega_c$ following the same expression in Eq. 2.9 but with different parameters $\alpha_c$ and $\beta_c$. Extended developments of Mazars’ model contain an improved Mazars’ model considering the unilateral character of damage [257] and three-dimensional Mazars’ model ($\mu$ model) to simulate the damage of concrete under multi-axial, cyclic and dynamic loadings [113].

2.4.1.2 Equivalent strain

The equivalent strain $\tilde{\varepsilon}$ defines the the evolution laws in Eqs. (2.7–2.9), since they are functions of the maximum historical equivalent strain $\kappa = \max \{ \varepsilon(t) \}$. Under an uniaxial loading, $\varepsilon$ equals the axial strain. To adopt these evolution laws in general loading conditions, such as biaxial and triaxial cases, the corresponding equivalent strain should be defined. This variable should be a scalar value considering all the strain components with weights.

The damage progression results in a release of fracture energy of the material. Therefore, the equivalent strain $\tilde{\varepsilon}$ can be expressed in a norm of energy as

$$\tilde{\varepsilon} = \sqrt{\frac{\varepsilon^T \mathbf{D} \varepsilon}{E}}, \quad (2.10)$$

in which $\varepsilon$ is the strain vector, $E$ is the Young’s modulus of the virgin material. However, the tensile and compressive strains can not be distinguished in Eq. (2.10). This definition is not suitable to describe concrete since it is quasi-brittle in tension but relatively ductile in compression.

A widely used equivalent strain definition is von Mises equivalent strain. It expresses $\tilde{\varepsilon}$ as

$$\tilde{\varepsilon} = \frac{1}{1 + \nu} \sqrt{-3J_2}, \quad (2.11)$$
where $J_2$ is the deviatoric strain vector

$$J_2 = \frac{1}{3} \left[ \varepsilon_{11}^2 + \varepsilon_{22}^2 + \varepsilon_{33}^2 - \varepsilon_{11}\varepsilon_{22} - \varepsilon_{22}\varepsilon_{33} - \varepsilon_{11}\varepsilon_{33} + 3 \left( \varepsilon_{12}^2 + \varepsilon_{23}^2 + \varepsilon_{13}^2 \right) \right]. \quad (2.12)$$

Based on the von Mises equivalent strain, De Vree et al. [258] proposed a definition that considers the tension-compression strain ratio:

$$\tilde{\varepsilon} = \frac{\gamma - 1}{2\gamma(1 - 2\nu)} J_1 + \frac{1}{2\gamma} \sqrt{\left(\frac{(\gamma - 1)^2}{(1 - 2\nu)^2}\right) J_1^2 + \frac{12\gamma}{(1 + \nu)^2} J_2}, \quad (2.13)$$

in which

$$J_1 = \varepsilon_1 + \varepsilon_2 + \varepsilon_3 \quad (2.14)$$

is first invariant of the strain tensor, $\gamma$ is a parameter defining the ratio of compressive to tensile strength under uniaxial loading. The modified von Mises definition of equivalent strain distinguishes the evolution laws in tension and compression for uniaxial loading condition. However, it overestimates the strength under biaxial tensile/compressive stress [259]. Besides, the cracking closure in a dynamic analysis can be reflected using this definition.

Mazars and Pijaudier-Cabot [260] proposed a definition for the concrete material as

$$\tilde{\varepsilon} = \sqrt{\sum_{i=1}^{3} \langle \varepsilon_i \rangle^2}, \quad (2.15)$$

where $\varepsilon_i$ indicates the principal strain, $\langle \rangle$ is the MacAulay bracket resulting in no-negative variable. This definition is suitable for the original Mazars’ model [253]. In the $\mu$ model [113], the equivalent strain is distinguished into $\tilde{\varepsilon}_t$ and $\tilde{\varepsilon}_c$ for cracking and crushing, respectively:

$$\tilde{\varepsilon}_t = \frac{I_\varepsilon}{2(1 - 2\nu)} + \frac{\sqrt{J_\varepsilon}}{2(1 + \nu)}, \quad (2.16a)$$

$$\tilde{\varepsilon}_c = \frac{I_\varepsilon}{5(1 - 2\nu)} + \frac{6\sqrt{J_\varepsilon}}{5(1 + \nu)}, \quad (2.16b)$$
in which \( I_ε \) has the same expression with \( J_1 \) in Eq. 2.14, and

\[
J_ε = 0.5 \left[ (ε_1 - ε_2)^2 + (ε_2 - ε_3)^2 + (ε_1 - ε_3)^2 \right]. \tag{2.17}
\]

Two thermodynamic variables \( Y_t \) and \( Y_c \) are defined based on this definition. They characterize the maximum strains reached in the tension and compression, respectively. In this way, the unilateral behavior of concrete (crack opening and closure) is considered, which is essential for the numerical simulations of concrete structures under seismic loadings.

### 2.4.2 Regularized damage models

As explained in Section 2.4.1, the constitutive model reflects the stress-strain relationship at the material point level, i.e., local damage model. In nonlinear numerical analysis, a local damage model exhibits an extreme mesh sensitivity and numerical instabilities [261, 242, 220]. It is because that the material softening results in the loss of ellipticity in the governing differential equations [262]. To avoid this pathological phenomenon, the local damage model should be enhanced with regularization. Various types of regularization approaches have been proposed, such as crack band model [263], smeared crack model [264], non-local integral formulations [261, 265], higher-order gradient theories [266], phase-field method [267], and thin level set approach [268].

### 2.5 Conclusion

In this chapter, the numerical methods related to seismic analysis of post-tensioned concrete gravity dam-reservoir-foundation systems are reviewed. This thesis will use the substructure method since it can accurately simulate the fluid-structure (FSI) and soil-structure (SSI) interactions. The quadtree mesh generation technique is chosen for the mesh generation as it generates smooth and rapid transition elements between coarser and finer meshes. Interfacial behavior of weak interfaces, such as dam-foundation inter-
face and construction joints, will be simulated by introducing interface elements. Post-tensioned anchors will be explicitly modeled, while the bond-slip interaction at the bond length can be simulated by using interface elements again. Conforming interfacial meshes will be used as they facilitate the interfacial problems by enhancing the contact simulation performance, avoiding interface constraints, etc. The scaled boundary finite element method (SBFEM) will be used for the simulations as this method shows advantages:

1. The near-field foundation can be limited to a small size since the SBFEM rigorously captures the dynamic properties of the unbounded foundation;

2. Polytype elements can be derived by the SBFEM, which significantly alleviates the meshing burden on generating conforming interfacial meshes.
Chapter 3

Scaled boundary finite element method

The scaled boundary finite element method (SBFEM) is in its core nature a semi-analytical method which was first proposed by Wolf and Song [108, 109] for modeling wave propagation in unbounded domains. This method has been shown to be an attractive technique to deal with structure-soil interaction problems [166, 269, 270]. The near-field of bounded domains can be discretized using polytope elements [108], while the far-field of the unbounded foundation can be accurately represented by the unbounded elements [109].

The fundamental theory of the SBFEM is presented in this chapter. Section 3.1 presents the polyhedral elements derived by the SBFEM. This thesis aims at two-dimensional seismic analysis of gravity dams. Therefore, the derivations of two-dimensional unbounded elements constructed by the SBFEM are briefly summarized in Section 3.2. In Section 3.3, the near-field/far-field coupling system subjected to free-field motion is presented. Readers interested in detailed discussions of this method are referred to the monograph by Song [22].

3.1 Polyhedral elements in the SBFEM

Polyhedral elements offer a great deal of flexibility in terms of mesh generation and mesh transitioning, especially for geometrically complex structures and adaptive analyses. One of the salient features of SBFEM is that we can straightforwardly derive such elements. In
general, a polyhedral mesh can be created by discretizing the boundary into surface elements, while the volume of the domain is not discretized itself [109]. A typical example of a polyhedral element employing the SBFEM is illustrated in Fig. 3.1. A so-called scaling center (denoted as “O” in Fig. 3.1a) is selected within the volume, such that every point on the boundary is directly visible, i.e., the domain has to fulfill the requirements of star-convexity. The boundary of the polyhedral domain is discretized using surface elements, which are commonly of triangular and quadrilateral shapes as depicted in Fig. 3.1b. The volume of the polyhedral mesh is represented by scaling the surface elements towards the scaling center. Only the solution on surface elements is interpolated whereas the solution inside the polyhedral element will be obtained analytically, leading to a semi-analytical procedure.

The surface elements on the boundary are isoparametric elements known from the standard FEM [271]. The most commonly used triangular and quadrilateral elements are depicted in Fig. 3.2 in its natural coordinates \((\eta, \zeta)\). The nodes marked in red are only used for quadratic elements. The surface element’s shape functions are in matrix form

\[
\mathbf{N}(\eta, \zeta) = [N_1(\eta, \zeta) \ N_2(\eta, \zeta) \ \cdots],
\]  

(3.1)

where \(N_i(\eta, \zeta)\) is the shape function related to node \(i\). For an isoparametric linear trian-
For an isoparametric linear quadrilateral element, the shape functions are

\[ N_1(\eta, \zeta) = 1 - \eta - \zeta, \]  \hspace{1cm} (3.2a)
\[ N_2(\eta, \zeta) = \eta, \]  \hspace{1cm} (3.2b)
\[ N_3(\eta, \zeta) = \zeta. \]  \hspace{1cm} (3.2c)

For an isoparametric linear quadrilateral element, the shape functions are

\[ N_1(\eta, \zeta) = \frac{1}{4}(1 - \eta)(1 - \zeta), \]  \hspace{1cm} (3.3a)
\[ N_2(\eta, \zeta) = \frac{1}{4}(1 + \eta)(1 - \zeta), \]  \hspace{1cm} (3.3b)
\[ N_3(\eta, \zeta) = \frac{1}{4}(1 - \eta)(1 + \zeta), \]  \hspace{1cm} (3.3c)
\[ N_4(\eta, \zeta) = \frac{1}{4}(1 + \eta)(1 + \zeta). \]  \hspace{1cm} (3.3d)

It is worthwhile to mention that in the framework of the SBFEM, often standard low-order finite elements are used for the surface tessellation, although high-order elements [272] and arbitrary faceted polyhedra [273] are possible.

### 3.1.1 Scaled boundary geometry transformation

The geometry transformation in the SBFEM is formulated by combining the boundary discretization with a scaling operation of the boundary. The scaling procedure is illu-
Figure 3.3. Scaling of the surface element using radial coordinate $\xi$ ($\xi = 0$ at the scaling center and $\xi = 1$ on the boundary)

trated in Fig. 3.3 for a quadrilateral surface element. A volume sector $V_e$ is represented by continuously scaling the surface element $S_e$ to the scaling center. A three-dimensional scaled boundary coordinate system is established in each volume sector, with the circumferential coordinates $\eta, \zeta$ on the surface element and the radial coordinate $\xi$. The radial coordinate $\xi$ is dimensionless, emanating from the scaling center and pointing towards the boundary. At the scaling center $\xi = 0$ and on the boundary $\xi = 1$ hold.

A Cartesian coordinate system ($\hat{x}, \hat{y}, \hat{z}$) is introduced for a polyhedral element to describe the node position. The scaling center is chosen as the origin of the Cartesian coordinate system. The position vector of an arbitrary point ($\hat{x}, \hat{y}, \hat{z}$) within the volume sector $V_e$ is expressed as

$$\mathbf{r} = \hat{x}\mathbf{i} + \hat{y}\mathbf{j} + \hat{z}\mathbf{k}, \quad (3.4)$$

in which $\mathbf{i}, \mathbf{j}$ and $\mathbf{k}$ are unit vectors in the Cartesian coordinates. The spherical coordinates (origin at the scaling center) of the point are

$$r = \sqrt{\hat{x}^2 + \hat{y}^2 + \hat{z}^2}, \quad (3.5a)$$

$$\theta = \arctan \frac{\hat{y}}{\hat{x}}, \quad (3.5b)$$

$$\phi = \arccos \frac{\hat{z}}{r}, \quad (3.5c)$$
where \( r \) is the radial coordinate, \( \theta \) is the azimuthal angle (the angle in \( \hat{x}\hat{y} \)-plane from the \( \hat{x} \)-axis), and \( \phi \) is the polar angle (the angle from positive \( \hat{z} \)-axis). For a point on the boundary, the coordinates are denoted as \((\hat{x}_b, \hat{y}_b, \hat{z}_b)\), and then, it has the point vector

\[
\mathbf{r}_b = \hat{x}_b \mathbf{i} + \hat{y}_b \mathbf{j} + \hat{z}_b \mathbf{k}.
\]  

Since the whole boundary is directly from the scaling center, a point \((\hat{x}, \hat{y}, \hat{z})\) within the volume can be defined through a point on the boundary \((\hat{x}_b, \hat{y}_b, \hat{z}_b)\) by a scaling operation:

\[
\mathbf{r} = \xi \mathbf{r}_b.
\]  

Therefore, the Cartesian coordinates \((\hat{x}, \hat{y}, \hat{z})\) can be rewritten as

\[
\begin{align*}
\hat{x} &= \xi \hat{x}_b, \\
\hat{y} &= \xi \hat{y}_b, \\
\hat{z} &= \xi \hat{z}_b.
\end{align*}
\]  

As explained in Section 3.1, the boundary of a polyhedral element is described as isoparametric surface elements. The point on the boundary \((\hat{x}_b, \hat{y}_b, \hat{z}_b)\) can be expressed in terms of \((\eta, \zeta)\) through the shape functions:

\[
\begin{align*}
\hat{x}_b(\eta, \zeta) &= \mathbf{N}(\eta, \zeta) \hat{x}, \\
\hat{y}_b(\eta, \zeta) &= \mathbf{N}(\eta, \zeta) \hat{y}, \\
\hat{z}_b(\eta, \zeta) &= \mathbf{N}(\eta, \zeta) \hat{z},
\end{align*}
\]  

where \( \hat{x}, \hat{y} \) and \( \hat{z} \) are the nodal coordinate vectors of the surface element \( S_e \) in Cartesian coordinates. Substituting Eq. (3.9) into Eq. (3.8), an arbitrary point \((\hat{x}, \hat{y}, \hat{z})\) can be
expressed by the scaled boundary coordinates \((\eta, \zeta, \xi)\) as:

\[
\hat{x}(\xi, \eta, \zeta) = \xi \hat{x}(\eta, \zeta) = \xi N(\eta, \zeta) \hat{x},
\]

\[
\hat{y}(\xi, \eta, \zeta) = \xi \hat{y}(\eta, \zeta) = \xi N(\eta, \zeta) \hat{y},
\]

\[
\hat{z}(\xi, \eta, \zeta) = \xi \hat{z}(\eta, \zeta) = \xi N(\eta, \zeta) \hat{z}.
\]

Applying the chain rule, the partial differential operators with respect to the scaled boundary coordinates are expressed as

\[
\frac{\partial}{\partial \xi} = \frac{\partial}{\partial \hat{x}} \frac{\partial \hat{x}}{\partial \xi} + \frac{\partial}{\partial \hat{y}} \frac{\partial \hat{y}}{\partial \xi} + \frac{\partial}{\partial \hat{z}} \frac{\partial \hat{z}}{\partial \xi},
\]

\[
\frac{\partial}{\partial \eta} = \frac{\partial}{\partial \hat{x}} \frac{\partial \hat{x}}{\partial \eta} + \frac{\partial}{\partial \hat{y}} \frac{\partial \hat{y}}{\partial \eta} + \frac{\partial}{\partial \hat{z}} \frac{\partial \hat{z}}{\partial \eta},
\]

\[
\frac{\partial}{\partial \zeta} = \frac{\partial}{\partial \hat{x}} \frac{\partial \hat{x}}{\partial \zeta} + \frac{\partial}{\partial \hat{y}} \frac{\partial \hat{y}}{\partial \zeta} + \frac{\partial}{\partial \hat{z}} \frac{\partial \hat{z}}{\partial \zeta}.
\]

It can be written in matrix form as

\[
\begin{bmatrix}
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} \\
\frac{\partial}{\partial \zeta}
\end{bmatrix} = J(\xi, \eta, \zeta) \begin{bmatrix}
\frac{\partial}{\partial \hat{x}} \\
\frac{\partial}{\partial \hat{y}} \\
\frac{\partial}{\partial \hat{z}}
\end{bmatrix},
\]

where \(J(\xi, \eta, \zeta)\) denotes the Jacobian matrix defined as

\[
J(\xi, \eta, \zeta) = \begin{bmatrix}
\hat{x}_\xi & \hat{y}_\xi & \hat{z}_\xi \\
\hat{x}_\eta & \hat{y}_\eta & \hat{z}_\eta \\
\hat{x}_\zeta & \hat{y}_\zeta & \hat{z}_\zeta
\end{bmatrix},
\]

in which the argument \((\xi, \eta, \zeta)\) has been dropped for the sake of clarity. Substituting Eqs. (3.8–3.9) into Eq. (3.13), the explicit expression of the Jacobian matrix is expressed
as

\[
\mathbf{J}(\xi, \eta, \zeta) = \begin{bmatrix}
\dot{x}_b & \dot{y}_b & \dot{z}_b \\
\dot{x}_b,\eta & \dot{y}_b,\eta & \dot{z}_b,\eta \\
\dot{x}_b,\zeta & \dot{y}_b,\zeta & \dot{z}_b,\zeta
\end{bmatrix} = \begin{bmatrix}
1 & 0 & 0 \\
0 & \xi & 0 \\
0 & 0 & \xi
\end{bmatrix} \mathbf{J}(\eta, \zeta),
\]  
(3.14)

where \( \mathbf{J}(\eta, \zeta) \) denotes the Jacobian matrix on the boundary \((\xi = 1)\), which is defined as

\[
\mathbf{J}(\eta, \zeta) = \begin{bmatrix}
\dot{x}_b & \dot{y}_b & \dot{z}_b \\
\dot{x}_b,\eta & \dot{y}_b,\eta & \dot{z}_b,\eta \\
\dot{x}_b,\zeta & \dot{y}_b,\zeta & \dot{z}_b,\zeta
\end{bmatrix}.
\]  
(3.15)

The entries of \( \mathbf{J}(\eta, \zeta) \) are obtained from Eqs. (3.9–3.10) as

\[
\dot{x}_b = N\dot{x}, \quad \dot{y}_b = N\dot{y}, \quad \dot{z}_b = N\dot{z},
\]  
(3.16a)

\[
\dot{x}_b,\eta = N,\eta \dot{x}, \quad \dot{y}_b,\eta = N,\eta \dot{y}, \quad \dot{z}_b,\eta = N,\eta \dot{z},
\]  
(3.16b)

\[
\dot{x}_b,\zeta = N,\eta \dot{x}, \quad \dot{y}_b,\zeta = N,\eta \dot{y}, \quad \dot{z}_b,\zeta = N,\eta \dot{z}.
\]  
(3.16c)

The determinant of \( \mathbf{J}(\eta, \zeta) \) is given as

\[
|\mathbf{J}| = \dot{x}_b (\dot{y}_b,\eta \dot{z}_b,\zeta - \dot{z}_b,\eta \dot{y}_b,\zeta) + \dot{y}_b (\dot{z}_b,\eta \dot{x}_b,\zeta - \dot{x}_b,\eta \dot{z}_b,\zeta) + \dot{z}_b (\dot{x}_b,\eta \dot{y}_b,\zeta - \dot{y}_b,\eta \dot{x}_b,\zeta).
\]  
(3.17)

The inverse of the Jacobian matrix on the boundary \( \mathbf{J}^{-1} \) is

\[
\mathbf{J}^{-1} = \begin{bmatrix}
j_{11} & j_{12} & j_{13} \\
j_{21} & j_{22} & j_{23} \\
j_{31} & j_{32} & j_{33}
\end{bmatrix},
\]  
(3.18)

with entries

\[
j_{11} = \frac{1}{|\mathbf{J}|} (\dot{y}_b,\eta \dot{z}_b,\zeta - \dot{z}_b,\eta \dot{y}_b,\zeta),
\]  
(3.19a)

\[
j_{21} = \frac{1}{|\mathbf{J}|} (\dot{z}_b,\eta \dot{x}_b,\zeta - \dot{x}_b,\eta \dot{z}_b,\zeta).
\]  
(3.19b)
\[
j_{31} = \frac{1}{|J_b|} (\hat{x}_b \eta \hat{y}_b \zeta - \hat{y}_b \eta \hat{x}_b \zeta),
\]
\[
j_{12} = \frac{1}{|J_b|} (\hat{z}_b \hat{y}_b \zeta - \hat{y}_b \hat{z}_b \zeta),
\]
\[
j_{22} = \frac{1}{|J_b|} (\hat{x}_b \hat{z}_b \zeta - \hat{z}_b \hat{x}_b \zeta),
\]
\[
j_{32} = \frac{1}{|J_b|} (\hat{y}_b \hat{x}_b \zeta - \hat{x}_b \hat{y}_b \zeta),
\]
\[
j_{31} = \frac{1}{|J_b|} (\hat{y}_b \hat{z}_b \eta - \hat{z}_b \hat{y}_b \eta),
\]
\[
j_{32} = \frac{1}{|J_b|} (\hat{z}_b \hat{y}_b \eta - \hat{y}_b \hat{z}_b \eta),
\]
\[
j_{33} = \frac{1}{|J_b|} (\hat{x}_b \hat{y}_b \eta - \hat{y}_b \hat{x}_b \eta).
\]

Inverting Eq. (3.12) and replacing $J(\xi, \eta, \zeta)$ by $J_b(\eta, \zeta)$, the partial derivatives with respect to Cartesian coordinates can be written as

\[
\begin{pmatrix}
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} \\
\frac{\partial}{\partial \zeta}
\end{pmatrix} =
\begin{pmatrix}
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1
\end{pmatrix} J_b^{-1}
\begin{pmatrix}
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} \\
\frac{\partial}{\partial \zeta}
\end{pmatrix}
\]

\[
= \begin{pmatrix}
j_{11} \\
j_{21} \\
j_{31}
\end{pmatrix} \frac{\partial}{\partial \xi} + \begin{pmatrix}
j_{12} \\
j_{22} \\
j_{32}
\end{pmatrix} \frac{1}{\zeta} \frac{\partial}{\partial \eta} + \begin{pmatrix}
j_{13} \\
j_{23} \\
j_{33}
\end{pmatrix} \frac{1}{\zeta^2} \frac{\partial}{\partial \zeta}.
\]

(3.20)

3.1.2 Geometrical properties in scaled boundary coordinates

The partial derivatives of the position vector $\mathbf{r}$ with respect to the scaled boundary coordinates are

\[
\mathbf{r}_\xi = \mathbf{r}_b = \hat{x}_b \mathbf{i} + \hat{y}_b \mathbf{j} + \hat{z}_b \mathbf{k},
\]
\[
\mathbf{r}_\eta = \hat{\xi} \mathbf{r}_b \eta = \xi (\hat{x}_b \eta \mathbf{i} + \hat{y}_b \eta \mathbf{j} + \hat{z}_b \eta \mathbf{k}),
\]
\[
\mathbf{r}_\zeta = \hat{\xi} \mathbf{r}_b \zeta = \xi (\hat{x}_b \zeta \mathbf{i} + \hat{y}_b \zeta \mathbf{j} + \hat{z}_b \zeta \mathbf{k}).
\]
where \( \mathbf{r}_{b,\eta} \) and \( \mathbf{r}_{b,\zeta} \) represent the tangential vectors to the surface element, as shown in Fig. 3.3. The outward normal vector to the surface \((\eta, \zeta)\) can be obtained as

\[
\mathbf{g}^\xi = \mathbf{r}_{b,\eta} \times \mathbf{r}_{b,\zeta} \\
= \begin{vmatrix}
\hat{x}_{b,\eta} & \hat{y}_{b,\eta} & \hat{z}_{b,\eta} \\
\hat{x}_{b,\zeta} & \hat{y}_{b,\zeta} & \hat{z}_{b,\zeta}
\end{vmatrix}
\]

\[
= (\hat{y}_{b,\eta} \hat{x}_{b,\zeta} - \hat{x}_{b,\eta} \hat{y}_{b,\zeta}) \mathbf{i} + (\hat{z}_{b,\eta} \hat{x}_{b,\zeta} - \hat{x}_{b,\eta} \hat{z}_{b,\zeta}) \mathbf{j} + (\hat{z}_{b,\eta} \hat{y}_{b,\zeta} - \hat{y}_{b,\eta} \hat{z}_{b,\zeta}) \mathbf{k}.
\] (3.22)

Likewise, the outward normal vectors to the side surfaces \((\zeta, \xi)\) and \((\xi, \eta)\) are

\[
\mathbf{g}^\eta = \mathbf{r}_{b,\zeta} \times \mathbf{r}_{b,\xi} = (\hat{z}_{b,\xi} \hat{y}_{b,\zeta} - \hat{y}_{b,\xi} \hat{z}_{b,\zeta}) \mathbf{i} + (\hat{x}_{b,\xi} \hat{z}_{b,\zeta} - \hat{z}_{b,\xi} \hat{x}_{b,\zeta}) \mathbf{j} + (\hat{x}_{b,\xi} \hat{y}_{b,\zeta} - \hat{y}_{b,\xi} \hat{x}_{b,\zeta}) \mathbf{k},
\] (3.23a)

\[
\mathbf{g}^\zeta = \mathbf{r}_{b,\xi} \times \mathbf{r}_{b,\eta} = (\hat{y}_{b,\xi} \hat{x}_{b,\eta} - \hat{x}_{b,\xi} \hat{y}_{b,\eta}) \mathbf{i} + (\hat{z}_{b,\xi} \hat{y}_{b,\eta} - \hat{y}_{b,\xi} \hat{z}_{b,\eta}) \mathbf{j} + (\hat{z}_{b,\xi} \hat{x}_{b,\eta} - \hat{x}_{b,\xi} \hat{z}_{b,\eta}) \mathbf{k}.
\] (3.23b)

An infinitesimal volume \( dV \) is calculated as

\[
dV = \mathbf{r}_{b,\xi} \cdot (\mathbf{r}_{b,\eta} \times \mathbf{r}_{b,\zeta}) d\xi d\eta d\zeta = \xi^2 \mathbf{r}_b \cdot (\mathbf{r}_{b,\eta} \times \mathbf{r}_{b,\zeta}) d\xi d\eta d\zeta.
\] (3.24)

Considering Eq. (3.21), the vector product equals to the determinant of the Jacobian matrix on the boundary \(|\mathbf{J}_b|\):

\[
\mathbf{r}_b \cdot (\mathbf{r}_{b,\eta} \times \mathbf{r}_{b,\zeta}) = \begin{vmatrix}
\hat{x}_b & \hat{y}_b & \hat{z}_b \\
\hat{x}_{b,\eta} & \hat{y}_{b,\eta} & \hat{z}_{b,\eta} \\
\hat{x}_{b,\zeta} & \hat{y}_{b,\zeta} & \hat{z}_{b,\zeta}
\end{vmatrix} = |\mathbf{J}_b|.
\] (3.25)

Therefore, the infinitesimal volume can be rewritten as

\[
dV = \xi^2 |\mathbf{J}_b| d\xi d\eta d\zeta.
\] (3.26)
The infinitesimal surface area \( dS_\xi \) for an arbitrary \( \xi \) is equal to

\[
dS_\xi = |r_\eta \times r_\zeta| \, d\eta d\zeta = \xi^2 \left| g^\xi \right| \, d\eta d\zeta, \tag{3.27}
\]

where \( \left| g^\xi \right| \) denotes the magnitude of the vector \( g^\xi \). Similarly, the infinitesimal surface area \( dS_\eta \) and \( dS_\zeta \) (on the side faces) are given as

\[
dS_\eta = |r_{b, \xi} \times r_\xi| \, d\zeta d\xi = \xi \left| g^\eta \right| \, d\zeta d\xi, \tag{3.28a}
\]
\[
dS_\zeta = |r_\xi \times r_{b, \zeta}| \, d\xi d\eta = \xi \left| g^\zeta \right| \, d\xi d\eta, \tag{3.28b}
\]

in which \( \left| g^\eta \right| \) and \( \left| g^\zeta \right| \) are the magnitude of the vectors \( g^\eta \) and \( g^\zeta \), respectively.

The normalized outward normal vectors can be expressed as

\[
n^\xi = \frac{g^\xi}{|g^\xi|} = n^\xi_i + n^\xi_j + n^\xi_k, \tag{3.29a}
\]
\[
n^\eta = \frac{g^\eta}{|g^\eta|} = n^\eta_i + n^\eta_j + n^\eta_k, \tag{3.29b}
\]
\[
n^\zeta = \frac{g^\zeta}{|g^\zeta|} = n^\zeta_i + n^\zeta_j + n^\zeta_k. \tag{3.29c}
\]

### 3.1.3 Governing equations of elastodynamics

In three-dimensional elasticity theory, the strain vector \( \varepsilon \) at an arbitrary point is calculated from the displacement field \( u \) as

\[
\varepsilon = Lu, \tag{3.30}
\]
where $L$ is the differential operator

$$
L = \begin{bmatrix}
\frac{\partial}{\partial \xi} & 0 & 0 \\
0 & \frac{\partial}{\partial \eta} & 0 \\
0 & 0 & \frac{\partial}{\partial \zeta} \\
0 & \frac{\partial}{\partial \eta} & \frac{\partial}{\partial \zeta} \\
\frac{\partial}{\partial \xi} & 0 & \frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} & \frac{\partial}{\partial \zeta} & 0
\end{bmatrix}.
$$

Substituting Eq. (3.20) into Eq. (3.31), the linear differential operator $L(\xi, \eta, \zeta)$ in scaled boundary coordinates is defined as

$$
L = b_1(\eta, \zeta) \frac{\partial}{\partial \xi} + \frac{1}{\xi} \left( b_2(\eta, \zeta) \frac{\partial}{\partial \eta} + b_3(\eta, \zeta) \frac{\partial}{\partial \zeta} \right),
$$

where the matrices $b_1, b_2, b_3$ are

$$
b_1 = \begin{bmatrix}
j_{11} & 0 & 0 \\
0 & j_{21} & 0 \\
0 & 0 & j_{31} \\
0 & j_{31} & j_{21} \\
j_{31} & 0 & j_{11} \\
j_{21} & j_{11} & 0
\end{bmatrix},
$$

$$
b_2 = \begin{bmatrix}
j_{12} & 0 & 0 \\
0 & j_{22} & 0 \\
0 & 0 & j_{32} \\
0 & j_{32} & j_{22} \\
j_{32} & 0 & j_{12} \\
j_{22} & j_{12} & 0
\end{bmatrix},
$$

where $j_{ij}$ are constants.
Moreover, by comparing Eq. (3.33) and Eq. (3.18–3.19), the following relationship can be derived

\[(|J_b|b_2)_\eta + (|J_b|b_3)_{\zeta} = -2|J_b|b_1.\]  

(3.34)

The governing equations in elastodynamics (neglecting body forces) are known as

\[L^T\sigma = \rho \ddot{u},\]  

(3.35)

where \(\rho\) is the mass density, the double dots (\(\ddot{\cdot}\)) denotes the accelerations. With introducing \(L(\xi, \eta, \zeta)\), the governing equations in scaled boundary coordinates are expressed as

\[b_1^T(\eta, \zeta) \frac{\partial \sigma}{\partial \xi} + \frac{1}{\xi} (b_2^T(\eta, \zeta) \frac{\partial \sigma}{\partial \eta} + b_3^T(\eta, \zeta) \frac{\partial \sigma}{\partial \zeta}) = \rho \ddot{u}.\]  

(3.36)

The tractions on the surfaces \((\eta, \zeta), (\zeta, \xi)\) and \((\xi, \eta)\) can be expressed as

\[t^\xi = \begin{bmatrix} n_\xi^\xi & 0 & 0 & n_\zeta^\xi & n_\eta^\xi \\ 0 & n_\eta^\xi & 0 & n_\zeta^\xi & n_\eta^\xi \\ 0 & 0 & n_\zeta^\xi & n_\eta^\xi & n_\xi^\xi \end{bmatrix} \sigma,\]  

(3.37a)

\[t^\eta = \begin{bmatrix} n_\xi^\eta & 0 & 0 & n_\zeta^\eta & n_\eta^\eta \\ 0 & n_\eta^\eta & 0 & n_\zeta^\eta & n_\eta^\eta \\ 0 & 0 & n_\zeta^\eta & n_\eta^\eta & n_\xi^\eta \end{bmatrix} \sigma,\]  

(3.37b)

\[b_3 = \begin{bmatrix} j_{13} & 0 & 0 \\ 0 & j_{23} & 0 \\ 0 & 0 & j_{33} \\ j_{33} & 0 & j_{23} \\ j_{23} & j_{13} & 0 \end{bmatrix} .\]  

(3.33c)
Using Eqs. (3.21–3.23), Eq. (3.37) can now be written as

\[
\begin{bmatrix}
    n_\xi \xi & 0 & 0 & n_\xi \eta & n_\xi \zeta \\
    0 & n_\xi \xi & 0 & n_\xi \eta & n_\xi \zeta \\
    0 & 0 & n_\xi \xi & 0 & n_\xi \eta \\
    0 & 0 & n_\xi \eta & 0 & n_\xi \xi \\
\end{bmatrix} \begin{bmatrix}
    u_\xi \\
    u_\eta \\
    u_\zeta \\
\end{bmatrix} = \sigma.
\]

3.1.4 Displacement, strain and stress fields

As a semi-analytical procedure, the SBFEM introduces unknown nodal displacement functions \( u^e(\xi) \) for the representation of the displacement field of a volume sector \( V_e \) in radial direction, i.e., those functions express the displacements along the radial lines connecting the scaling center and boundary nodes on \( e \)-th surface element \( S_e \), and they are usually determined analytically. For the sake of completeness, we remark that in Refs. [274, 275] a numerical approximation in radial direction has been successfully established. The nodal displacement functions \( u(\xi) \) of the entire element are assembled by

\[
\begin{align*}
    u(\xi) &= \sum_{e} A^e u^e(\xi), \\
    u(\xi, \eta, \zeta) &= \sum_{e} N_\alpha(\eta, \zeta, \xi) u(\xi),
\end{align*}
\]

where the symbol \( A^e \) indicates the standard finite element assembly procedure.

The displacements at a point \((\xi, \eta, \zeta)\) within the volume sector \( V_e \) are interpolated based on \( u^e(\xi) \), thus the displacement field \( u(\xi, \eta, \zeta) \) is expressed as

\[
u(\xi, \eta, \zeta) = \sum_{e} N_\alpha(\eta, \zeta) u(\xi),
\]
where the shape function matrix $N_u(\eta, \zeta)$ is written as

$$N_u(\eta, \zeta) = [N_1(\eta, \zeta)I, N_2(\eta, \zeta)I, \cdots, N_n(\eta, \zeta)I], \quad (3.41)$$

with a $3 \times 3$ identity matrix $I$.

Substituting Eq. (3.40) into Eq. (3.30), the relationship between the strain field and the nodal displacement functions can be obtained as

$$\varepsilon(\xi, \eta, \zeta) = B_1(\eta, \zeta)u(\xi, \eta, \zeta) + \frac{1}{\xi} B_2(\eta, \zeta)u(\xi), \quad (3.42)$$

where

$$B_1(\eta, \zeta) = b_1(\eta, \zeta)N_u(\eta, \zeta), \quad (3.43a)$$
$$B_2(\eta, \zeta) = b_2(\eta, \zeta)N_u(\eta, \zeta) + b_3(\eta, \zeta)N_u(\eta, \zeta, \zeta). \quad (3.43b)$$

For linear elastic problems, the stress field $\sigma(\xi, \eta, \zeta)$ is related to the strain field by

$$\sigma(\xi, \eta, \zeta) = D\varepsilon(\xi, \eta, \zeta), \quad (3.44)$$

where $D$ is the elasticity matrix, which is expressed as follows for linear isotropic material:

$$D = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix} 1 - \nu & \nu & \nu & 0 & 0 & 0 \\ 1 - \nu & \nu & 0 & 0 & 0 & 0 \\ 1 - \nu & 0 & 0 & 0 & 0 & 0 \\ 1 - 2\nu & 0 & 0 & 0 & 0 & 0 \\ \frac{1 - 2\nu}{2} & 0 & 0 & 0 & 0 & 0 \\ \frac{1 - 2\nu}{2} & 0 & 0 & 0 & 0 & 0 \\ \text{Symmetric} & \frac{1 - 2\nu}{2} & 0 & 0 & 0 & 0 \end{bmatrix}, \quad (3.45)$$

where $E$ is Young’s modulus and $\nu$ is Poisson’s ratio. Substituting Eq. (3.42) into Eq. (3.44),
the relationship between the stress field and the nodal displacement functions can be expressed as
\[
\sigma(\xi, \eta, \zeta) = D \left( B_1(\eta, \zeta) u(\xi) + \frac{1}{\xi} B_2(\eta, \zeta) u(\xi) \right).
\] (3.46)

### 3.1.5 Scaled boundary finite element equation

The scaled boundary finite element equation can be derived using the Galerkin’s weighted residual method [276] or the virtual work principle [277]. In this section, the derivation based on the Galerkin’s weighted residual method is presented.

The field of weighting function \( \omega(\xi, \eta, \zeta) \) in the Galerkin’s weighted residual technique is constructed similarly with the displacement field (see Eq. (3.40))

\[
\omega(\xi, \eta, \zeta) = N_u(\eta, \zeta) \omega(\xi),
\] (3.47)

where \( \omega(\xi) \) is the vector of the weighting functions along the radial coordinate \( \xi \). Corresponding to the internal displacement functions \( u(\xi) \), internal nodal force functions \( q(\xi) \) are also introduced along the radial coordinate. As to the surface with a constant \( \xi \), \( q(\xi) \) is statically equivalent to the surface traction \( t^\xi \)

\[
\omega^T(\xi) q(\xi) = \int_{S_\xi} \omega^T t^\xi dS_\xi.
\] (3.48)

Substituting Eqs. (3.27), (3.38a) and (3.47) into Eq. (3.48) leads to

\[
q(\xi) = \xi^2 \int_{S_\xi} N_u^T(\eta, \zeta) b_1^T(\eta, \zeta) \sigma |J_b| d\eta d\zeta.
\] (3.49)

Using Eq. (3.43a), Eq. (3.49) can be further simplified as

\[
q(\xi) = \xi^2 \int_{S_\xi} B_1^T(\eta, \zeta) \sigma |J_b| d\eta d\zeta.
\] (3.50)
Substituting Eq. (3.46) into Eq. (3.50) yields

$$q(\xi) = \xi^2 \int_{S_\xi} B_1^T D \left( B_1 u(\xi) + \frac{1}{\xi} B_2 u(\xi) \right) |J_b| d\eta d\zeta,$$  \hspace{1cm} (3.51)

where the argument ($\eta$, $\zeta$) is omitted. Introducing the coefficient matrices

$$E_0 = \int_{S_\xi} B_1^T D B \frac{|J_b|}{d\eta d\zeta},$$ \hspace{1cm} (3.52a)
$$E_1 = \int_{S_\xi} B_2^T D B \frac{|J_b|}{d\eta d\zeta},$$ \hspace{1cm} (3.52b)

into Eq. (3.51) results in the internal nodal forces

$$q(\xi) = E_0 \xi^2 u(\xi) + E_1 \xi u(\xi).$$ \hspace{1cm} (3.53)

The Galerkin’s weighted-residual technique is applied to the governing partial differential equations. A domain $V$ covered by scaling a surface element on the boundary in the range $\xi_1 \leq \xi \leq \xi_2$ is considered. Premultiplying Eq. (3.36) by the weighting functions $\omega^T$ and integrating over the domain yields

$$\int_V \omega^T \left( b_1^T \sigma_{,\xi} + \frac{1}{\xi} \left( b_2^T \sigma_{,\eta} + b_3^T \sigma_{,\zeta} \right) - \rho \ddot{u} \right) dV = 0.$$ \hspace{1cm} (3.54)

Substituting Eqs. (3.47) and (3.40), Eq. (3.54) is separated into three terms $W_I$, $W_{II}$ and $W_{III}$

$$\left\{ \int_{W_I} \omega^T \left( b_1^T \sigma_{,\xi} \right) dV \right\} + \left\{ \int_{W_{II}} \omega^T \left( \frac{1}{\xi} \left[ N_{ii} (b_2^T \sigma_{,\eta} + b_3^T \sigma_{,\zeta}) \right) \right) dV \right\} - \left\{ \int_{W_{III}} \omega^T (\xi) \frac{1}{N_{ii} \rho N_{ii}^T \ddot{u}(\xi)} dV \right\} = 0.$$ \hspace{1cm} (3.55)

By using Eq. (3.26), the first term $W_I$ is expressed as

$$W_I = \int_{\xi_1}^{\xi_2} \omega^T (\xi)^2 \int_{S_\xi} B_1^T (\eta, \zeta) \sigma_{,\xi} |J_b| d\eta d\zeta d\zeta,$$ \hspace{1cm} (3.56)
and the second term $W_{II}$ is equal to

$$W_{II} = \int_{\xi_1}^{\xi_2} \omega^T(\xi) \xi \int_{S_\xi} N_u^T(\eta, \zeta) (|J_b| b_2^T(\eta, \zeta) \sigma_{\eta} + |J_b| b_3^T(\eta, \zeta) \sigma_{\zeta}) d\xi d\eta d\zeta. \quad (3.57)$$

Applying the Green’s theorem to the surface integral over $S_\xi$ leads to

$$W_{II} = \int_{\xi_1}^{\xi_2} \omega^T(\xi) \xi \left( \int_{\Gamma_\xi} N_u^T(|J_b| b_2 \sigma d\xi + |J_b| b_3 \sigma d\eta) P_s(\xi) \right) - \xi \int_{S_\xi} \left( (N_u^T|J_b| b_2^T)_{\eta} + (N_u^T|J_b| b_3^T)_{\zeta} \right) \sigma d\eta d\zeta \right) d\xi, \quad (3.58)$$

where $\Gamma_\xi$ denotes the edges of the surface $S_\xi$. $\Gamma_\xi$ is formed by scaling the edges of the surface element on the boundary. Substituting Eqs. (3.38b–3.38c) into $P_s(\xi)$ leads to

$$P_s(\xi) = \xi \int_{\Gamma_\xi} N_u^T(t^\eta |g^\eta| d\xi + t^\zeta |g^\zeta| d\eta), \quad (3.59)$$

which simply shows that the nodal force functions are equivalent to the surface tractions at $\xi$. Using Eqs. (3.34) and (3.43), the term $W_{IIIB}$ can be derived as

$$W_{IIIB} = (-2B_1^T + B_2^T) |J_b|. \quad (3.60)$$

As a result, Eq. (3.58) is reformulated as

$$W_{II} = \int_{\xi_1}^{\xi_2} \omega^T(\xi) \left( P_s(\xi) + \xi \int_{S_\xi} (2B_1^T - B_2^T) \sigma |J_b| d\eta d\zeta \right) d\xi. \quad (3.61)$$

The third term of Eq. (3.55) $W_{III}$ is the contribution of the inertial forces. It can be rewritten as

$$W_{III} = \int_{\xi_1}^{\xi_2} \omega^T(\xi) \xi^2 M_0 \ddot{u}(\xi) d\xi \quad (3.62)$$
by introducing a mass coefficient matrix

\[ M_0 = \int_{\tilde{S}_\xi} N_u^T \rho N_u^T |J_b| d\eta d\zeta. \] (3.63)

Substituting Eqs. (3.56), (3.61) and (3.62) back into Eq. (3.55) yields

\[ \int_{\tilde{S}_\xi} \omega^T(\xi) \left( \int_{\tilde{S}_\xi} \left( B_1^T(\xi^2 \sigma, 2 \xi \sigma) |J_b| \right) d\eta d\zeta \\right) - \xi \int_{\tilde{S}_\xi} B_2^T \sigma |J_b| d\eta d\zeta + P_s(\xi) - \xi^2 M_0 \ddot{u}(\xi) \right) d\xi = 0. \] (3.64)

Considering \( \xi^2 \sigma, \xi + 2 \xi \sigma = (\xi^2 \sigma), \xi \), this equation is rewritten as

\[ \int_{\tilde{S}_\xi} \omega^T(\xi) \left( \int_{\tilde{S}_\xi} \left( B_1^T(\xi^2 \sigma), \xi |J_b| \right) d\eta d\zeta \right) - \xi \int_{\tilde{S}_\xi} B_2^T \sigma |J_b| d\eta d\zeta + P_s(\xi) - \xi^2 M_0 \ddot{u}(\xi) \right) d\xi = 0. \] (3.65)

Eq. (3.65) is satisfied by setting the integrand of the integral over \( \xi \) equal to zero

\[ \int_{\tilde{S}_\xi} B_1^T(\xi^2 \sigma), \xi |J_b| d\eta d\zeta - \xi \int_{\tilde{S}_\xi} B_2^T \sigma |J_b| d\eta d\zeta + P_s(\xi) - \xi^2 M_0 \ddot{u}(\xi) = 0. \] (3.66)

Using Eq. (3.50), the first term is rewritten as

\[ \int_{\tilde{S}_\xi} B_1^T(\xi^2 \sigma), \xi |J_b| d\eta d\zeta = q(\xi), \xi. \] (3.67)

By substituting Eq. (3.46) and assuming

\[ E_2 = \int_{\tilde{S}_\xi} B_2^T DB_2 |J_b| d\eta d\zeta, \] (3.68)
the second term is expressed as

\[
\xi \int_{S_2} B^T \sigma |J_b| d\eta d\zeta = E_1 \xi u(\xi) + E_2 u(\xi). \tag{3.69}
\]

Substituting Eqs. (3.67) and (3.69), Eq. (3.66) is reformulated as

\[
q(\xi) - E_1 \xi u(\xi) - E_2 u(\xi) + P_s(\xi) - \xi^2 M_0 \ddot{u}(\xi) = 0. \tag{3.70}
\]

Eq. (3.53) and Eq. (3.70) are system of equations for the nodal displacement functions \(u(\xi)\) and nodal force functions \(q(\xi)\). Differentiating Eq. (3.53) leads to

\[
q(\xi) + E_0 \xi^2 u(\xi) + 2E_0 \xi u(\xi) + E^T_1 u(\xi) + E^T_1 \xi u(\xi) = 0. \tag{3.71}
\]

Substituting Eq. (3.71) into Eq. (3.70) produces

\[
E_0 \xi^2 u(\xi) + (2E_0 - E_1 + E^T_1) \xi u(\xi) + (E^T_1 - E_2) u(\xi) + P_s(\xi) - \xi^2 M_0 \ddot{u}(\xi) = 0. \tag{3.72}
\]

Note that the coefficient matrices \(E_0, E_1, E_2\) and \(M_0\) of the entire element are assembled from the coefficient matrices \(E^e_0, E^e_1, E^e_2\) and \(M^e_0\) belonging to each surface element. To simplify the nomenclature, the same symbols of these matrices are used in this chapter for the assembled ones. During the assemblage process, the contribution of the surface tractions \(P_s(\xi)\) on the side faces of each sector will be canceled, leading to the following

**scaled boundary finite element equation in displacements**

\[
E_0 \xi^2 u(\xi) + (2E_0 + E^T_1 - E_1) \xi u(\xi) + (E^T_1 - E_2) u(\xi) - \xi^2 M_0 \ddot{u}(\xi) = 0. \tag{3.73}
\]

The system of non-homogeneous second-order ordinary differential equations (ODEs) in Eq. (3.73) can be transferred into a system of homogeneous first-order ODEs by dou-
bling the number of unknowns and neglecting the inertial loads. The variable

\[ X(\xi) = \begin{bmatrix} \xi^{0.5}u(\xi) \\ \xi^{-0.5}q(\xi) \end{bmatrix} \]  \hspace{1cm} (3.74)

is introduced. Substituting Eqs. (3.74) and (3.53) into Eq. (3.73), the scaled boundary finite element equation is reformulated as

\[ \xi X(\xi),_\xi = Z_p X(\xi), \]  \hspace{1cm} (3.75)

where the coefficient matrix \( Z_p \) is a Hamiltonian matrix and defined as

\[ Z_p = \begin{bmatrix} -E_0^{-1}E_1^T + 0.5I & E_0^{-1} \\ E_2 - E_1E_0^{-1}E_1^T & E_1E_0^{-1} - 0.5I \end{bmatrix}. \]  \hspace{1cm} (3.76)

### 3.1.6 Solution of the scaled boundary finite element equation

The scaled boundary finite element equation constitutes a system of Euler-Cauchy equations, see Eq. (3.75), which can be solved by applying eigenvalue or block-diagonal Schur decompositions [278]. The simpler eigenvalue method is utilized for the numerical implementation of our work, and therefore, a brief summary is presented in this subsection.

The eigenvalue problem of \( Z_p \) can be expressed in matrix form as

\[ Z_p \phi = \phi \lambda, \]  \hspace{1cm} (3.77)

where \( \phi \) denotes the eigenvector matrix, and \( \lambda \) is the diagonal matrix of eigenvalues. The general solution of \( X(\xi) \) is expressed as:

\[ X(\xi) = \phi \xi^\lambda \mathbf{c}, \]  \hspace{1cm} (3.78)

where \( \mathbf{c} \) is the vector of integration constants which depends on the boundary conditions.
The general solution of $X(\xi)$, given in Eq. (3.78), can be sorted by the real parts of the eigenvalues in descending order. It is well-known that if $\lambda$ is an eigenvalue of a Hamiltonian matrix, $-\lambda$ is also an eigenvalue of this particular matrix [279]. Thus, Eq. (3.78) can be rewritten as

$$X(\xi) = \begin{bmatrix} \phi_{u1} & \phi_{u2} \\ \phi_{q1} & \phi_{q1} \end{bmatrix} \begin{bmatrix} \xi^{\lambda^+} & 0 \\ 0 & \xi^{\lambda^-} \end{bmatrix} \begin{bmatrix} c_1 \\ c_2 \end{bmatrix},$$

(3.79)

where $\phi_{u1}$, $\phi_{u2}$, $\phi_{q1}$, and $\phi_{q2}$ are the sub-matrices of $\phi$, and $\lambda^+$ and $\lambda^-$ are the diagonal matrices of eigenvalues with positive and negative real parts, respectively. $c_1$ and $c_2$ denote the vectors of integration constants related to $\lambda^+$ and $\lambda^-$, respectively. For bounded domains, the displacements have to remain finite inside the element and therefore, the integration constants contained in $c_2$ have to be equal to zero. Otherwise the power function $\xi^{\lambda^-}$ would go to infinity near the scaling center ($\xi = 0$). Consequently, the general solution given by Eq. (3.79) can be simplified as

$$X(\xi) = \begin{bmatrix} \phi_{u1} & \phi_{u2} \\ \phi_{q1} & \phi_{q1} \end{bmatrix} \xi^{\lambda^+} c_1.$$

(3.80)

Using the definition of $X(\xi)$ in Eq. (3.74) together with the general solution provided in Eq. (3.80), the unknown nodal displacement functions $u(\xi)$ and nodal force functions $q(\xi)$ can be expressed as

$$u(\xi) = \phi_{u1} \xi^{\lambda^+} - 0.5I c_1,$$  

(3.81a)

$$q(\xi) = \phi_{q1} \xi^{\lambda^+} + 0.5I c_1.$$  

(3.81b)

The relationship between the nodal displacement functions and the nodal force functions can be obtained by eliminating the integration constants as

$$q(\xi) = \phi_{q1} \phi_{u1}^{-1} \xi u(\xi).$$

(3.82)
On the boundary ($\xi = 1$), the nodal force vector $\mathbf{F} = \mathbf{q}(\xi = 1)$ and the nodal displacement vector $\mathbf{d} = \mathbf{u}(\xi = 1)$ apply. The relationship between nodal force and displacement vectors is $\mathbf{F} = \mathbf{K}\mathbf{d}$, therefore, the stiffness matrix of the subdomain is expressed as

$$
\mathbf{K} = \phi_{q1}\phi^{-1}_{u1}. \quad (3.83)
$$

The mass matrix of a volume element is formulated as [22]

$$
\mathbf{M} = \phi_{u1}^{-T} \int_{0}^{1} \xi^{\lambda^{+}} m_{0}\xi^{\lambda^{+}} \xi \phi_{u1}^{-1}, \quad (3.84)
$$

where the coefficient matrix $m_{0}$ is given by

$$
m_{0} = \phi_{u1}^{T} \mathbf{M}_{0} \phi_{u1}. \quad (3.85)
$$

Transferring the integration part in Eq. (3.84) into a matrix form, the expression of the mass matrix can be rewritten as

$$
\mathbf{M} = \phi_{u1}^{-T} \mathbf{m} \phi_{u1}^{-1}, \quad (3.86)
$$

where

$$
\mathbf{m} = \int_{0}^{1} \xi^{\lambda^{+}} m_{0}\xi^{\lambda^{+}} \xi d\xi. \quad (3.87)
$$

Each entry of $\mathbf{m}$ can be evaluated in an analytical fashion, resulting in

$$
m_{ij} = \frac{m_{0ij}}{\lambda_{ii}^{+} + \lambda_{jj}^{+} + 2}, \quad (3.88)
$$

where $m_{0ij}$ is the entry of $\mathbf{m}_{0}$ and $\lambda_{ii}^{+}$ and $\lambda_{jj}^{+}$ are particular entries of $\lambda^{+}$. 

82
3.2 Two-dimensional unbounded element

A two-dimensional unbounded element is depicted in Fig. 3.4. Similar to polytope elements, a scaling center “O” from which the total discretized boundary is visible is selected, but this time outside the unbounded element. One unbounded sector $V_e$ is covered by scaling a line element $S_e$ to infinity. The assembly of all unbounded sectors represents the semi-infinite half space. The radial coordinate $\xi$ ranges from 1 at the boundary to infinity. The unbounded domain is the assembly of all unbounded sectors. The boundary can be discretized into arbitrary numbers of line elements. The side faces represent free surfaces of the unbounded domain and are not discretized. The scaled boundary coordinate system is established in each sector with the radial coordinate $\xi$ and a circumferential coordinate $\eta$. The circumferential coordinate $\eta$ runs from -1 to 1 on the line element, following a counter-clockwise direction around the scaling center.

The two-dimensional scaled boundary finite element equation in displacement is

$$E_0 \xi^2 \ddot{u}(\xi)_{,\xi\xi} + (E_0 + E_1^\top - E_1)\xi u(\xi)_{,\xi} - E_2 u(\xi) - \xi^2 M_0 \ddot{u}(\xi) = 0.$$  (3.89)

It can be derived as the scaled boundary finite element equation in frequency domain at the boundary ($\xi = 1$) by introducing a dynamic stiffness matrix $S_{\omega}(\omega)$ for unbounded
\[(S_\infty(\omega) + E_1) E_0^{-1} (S_\infty(\omega) + E_1^T) - E_2 - \omega S_\infty(\omega) + \omega^2 M_0 = 0. \quad (3.90)\]

Solving Eq. (3.90) and applying an inverse Fourier transform, the dynamic stiffness can be formulated in terms of a displacement unit-impulse response matrix in the time domain

\[S_\infty(t) = C_\infty \dot{\delta}(t) + K_\infty \delta(t) + S_r^\infty(t) \quad (3.91)\]

with the Dirac delta function \(\delta(t)\). \(C_\infty\) and \(K_\infty\) in Eq. (3.91) are the constant damping and stiffness matrices, respectively, and are obtained from the singular part of the dynamic stiffness matrix for high frequencies. \(S_r^\infty(t)\) is the Fourier transform pair of the regular part \(S_r^\infty(\omega)\) (subscript r for regular). The ground interaction force \(R_b(t)\) at a specific time \(t\) is defined by a convolution integral of the product of displacement unit-impulse response matrix and the historical displacement \(u_b\)

\[R_b(t) = \int_0^t S_\infty(t - \tau) u_b(\tau) d\tau. \quad (3.92)\]

Substituting Eq. (3.91) in Eq. (3.92) yields the interaction force-displacement relationship

\[R_b(t) = C_\infty \dot{u}_b(t) + K_\infty u_b(t) + \int_0^t S_r^\infty(t - \tau) u_b(\tau) d\tau, \quad (3.93)\]

in which the first two terms are instantaneous responses and the integral term describes a lingering response.

The specific expressions of \(C_\infty\), \(K_\infty\) and \(S_r^\infty\) are summarized following Refs. [23, 269]. The matrices \(E_0\) and \(M_0\) are positive definite. The solution of a generalized eigenvalue problem \((M_0, E_0)\) results in eigenvectors \(\phi_\infty\) with positive eigenvalues \(\lambda_\infty^2\). The damping matrix \(C_\infty\) is formulated as

\[C_\infty = \phi_\infty^{-T} \lambda_\infty \phi_\infty^{-1}, \quad (3.94)\]
where $\lambda_\infty$ is the diagonal matrix comprised of the positive roots of each eigenvalue. The stiffness matrix $K_\infty$ is obtained by

$$K_\infty = \phi^{-T}_\infty K_\infty \phi^{-1}_\infty. \quad (3.95)$$

The coefficient matrix $k_\infty$ satisfies a Lyapunov equation with $\lambda_\infty$, for which each entry equals

$$k_{ij}^\infty = \frac{1}{\lambda_{i\infty} + \lambda_{j\infty}} \left( -\lambda_{i\infty} e_{ij}^1 - \lambda_{j\infty} e_{ij}^1 + \lambda_{i\infty} \delta_{ij} \right), \quad (3.96)$$

where $e_{ij}^1$ is the element of the matrix $e_1 = \phi^T_\infty E_1 \phi_\infty$, and $\delta_{ij}$ is the Kronecker delta.

To perform the convolution integral on the right-hand side of Eq. (3.93), an efficient estimation approach based on the implicit trapezoidal rule is presented in Ref. [270]. By employing such trapezoidal algorithm on a truncation time $n\Delta\bar{t}$, the integral term in Eq. (3.93) can be estimated as

$$\int_0^t S_\infty^r(t-\tau)u(\tau)d\tau \simeq \frac{\Delta\bar{t}}{2}(S_0^r u(t) + S_n^r u(t-n\Delta\bar{t})) + \Delta\bar{t} \sum_{j=1}^n S_j^r u(t-j\Delta\bar{t}), \quad (3.97)$$

in which $S_j^r = S_{\infty}^r(j\Delta\bar{t})$, and $\Delta\bar{t}$ is the time range of a subinterval. Owing to the mentioned approximation of the convolution integral involving $S_{\infty}^r(t)$, only the specific expression of $S_j^r$ at each time station needs to be determined. The lingering response matrix $S_j^r$ is expressed as

$$S_j^r = \phi^{-T}_\infty s_j^r \phi^{-1}_\infty, \quad (3.98)$$

in which the matrix $s_j^r$ can be obtained step by step following $s_j^r = s_{j-1}^r + \frac{\Delta\bar{t}}{2}(s_{j-1}^r + s_j^r)$ as demonstrated in Ref. [270]. It is worthwhile to mention that $S_{\infty}^r(t)$ decays with time, thus after a certain time $(n\Delta\bar{t}) S_n^r$ is small and therefore, $S_{\infty}^r(t)$ becomes negligible. With a sufficient short $\Delta\bar{t}$, Eq. (3.97) should be a precise estimation of the the convolution integral. The selection of $\Delta\bar{t}$ and $n$ has been discussed in Ref. [269]. It is suggested that, for an unbounded foundation with a width of $2b$, $\Delta\bar{t}$ should be smaller than $0.01 \frac{b}{c_s}$ ($c_s$ denotes the velocity of shear waves) and $S_{\infty}^r(0)$ should be smaller than $0.01 S_{\infty}^r(0)$. 

85
3.3 Near-field/far-field coupling system subjected to free-field motion

The coupling of near-field and far-field is reflected by the interaction force vector $R_b(t)$ acting on the nodes at the interface as shown in Fig. 3.5. The general equation of motion of the near-field system is written, by introducing an additional external force vector $-R_b(t)$ on the degrees of freedom (DOFs) of the boundary nodes, as

$$
\begin{bmatrix}
M_{ss} & M_{sb} \\
M_{bs} & M_{bb}
\end{bmatrix}
\begin{bmatrix}
\ddot{u}_s(t) \\
\ddot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
C_{ss} & C_{sb} \\
C_{bs} & C_{bb}
\end{bmatrix}
\begin{bmatrix}
\dot{u}_s(t) \\
\dot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
K_{ss} & K_{sb} \\
K_{bs} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
{u}_s(t) \\
{u}_b(t)
\end{bmatrix}
= \begin{bmatrix}
F_s(t) \\
F_b(t)
\end{bmatrix}
- \begin{bmatrix}
0 \\
R_b(t)
\end{bmatrix},
$$

(3.99)

where the subscripts s and b represent the DOFs of internal and boundary nodes of the near-field, respectively. Neglecting other external forces ($F_s(t)$ and $F_b(t)$ in Eq. (3.99)) and introducing seismic equivalent nodal forces, the *equation of motion of a near-field system experiencing a free-field motion* is expressed as [166]

$$
\begin{bmatrix}
M_{ss} & M_{sb} \\
M_{bs} & M_{bb}
\end{bmatrix}
\begin{bmatrix}
\ddot{u}_s(t) \\
\ddot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
C_{ss} & C_{sb} \\
C_{bs} & C_{bb}
\end{bmatrix}
\begin{bmatrix}
\dot{u}_s(t) \\
\dot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
K_{ss} & K_{sb} \\
K_{bs} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
{u}_s(t) \\
{u}_b(t)
\end{bmatrix}
= \begin{bmatrix}
0 \\
0
\end{bmatrix}
- \begin{bmatrix}
0 \\
R_b(t)
\end{bmatrix},
$$

(3.100)

in which the first term at the right hand side is the force vector due to free-field motion (denoted by superscript f) acting on the boundary nodes. The equivalent nodal forces consist of two contributions which represent the effects of the wave propagation on the near-field $F_b^s(t)$ and free-field motion of the unbounded domain $F_b^\infty(t)$, respectively.

In this work, the seismic analysis is performed for the near-field/far-field coupled system, with a special focus on the common vertically propagating S-waves. The displace-
Figure 3.5. Coupling of near-field/far-field

ment $u_f^b(t)$ and velocity $\dot{u}_f^b$ due to a free-field motion at the near-field/far-field interface can be derived if the motion is prescribed. Figure 3.6 demonstrates a near-field with height $H$, in which a upwards vertically propagated S-wave ($u_s(t)$ and $\dot{u}_s(t)$) is imposed at the bottom boundary. The horizontal motion $u^f$ and $\dot{u}^f$ generated by the S-wave are identical at a specific level, i.e. they are functions of $y$ and $t$, and the vertical motion $v^f$ and $\dot{v}^f$ are zero in the whole system. At Node B from the left boundary with coordinates $(0,y_b)$, the horizontal free-field motion ($u^f(y_b,t)$ and $\dot{u}^f(y_b,t)$) are expressed as [280]

\begin{align}
    u^f(y_b,t) &= u^I(y_b,t) + u^R(y_b,t) = u_s(t - \frac{y_b}{c_s}) + u_s(t - \frac{2H - y_b}{c_s}), \\
    \dot{u}^f(y_b,t) &= \dot{u}^I(y_b,t) + \dot{u}^R(y_b,t) = \dot{u}_s(t - \frac{y_b}{c_s}) + \dot{u}_s(t - \frac{2H - y_b}{c_s}),
\end{align}

in which $c_s$ is the velocity of S-wave in the medium. It consists of the incident ($u^I$ and $\dot{u}^I$) and reflected waves ($u^R$ and $\dot{u}^R$), which are the prescribed waves delayed by $\frac{y_b}{c_s}$ and $\frac{2H - y_b}{c_s}$, respectively.

The equivalent forces induced by wave propagation on the near-field boundary $F_f^b(t)$ can be represented by surface tractions as [166]

\begin{equation}
    F_f^b(t) = \int_S t^f dS,
\end{equation}
Figure 3.6. A free-field motion system subjected to a vertically propagated S-wave

in which $t^f = \{t_x, t_y\}^T$ is the surface traction due to free-field motion and $S$ is the boundary surface. For vertically propagating S-waves in an elastic half plane, the components of $t^f$ are given as

$$t_x = n_y \tau_{xy} = n_y G \frac{\partial u^f}{\partial y}, \quad t_y = n_x \tau_{xy} = n_x G \frac{\partial u^f}{\partial y},$$

(3.103)
in which $\tau_{xy}$ is the shear stress caused by the wave propagation, and $n_x$ and $n_y$ are the components of the outward unit normal vector $\{n_x, n_y\}^T$ of the boundary. The shear stress is a function of $y$ and $t$, formulated as

$$\tau_{xy}(y, t) = \frac{G}{c_s} \left( -\ddot{u}_s(t) - \frac{y}{c_s} + \dot{u}_s(t) \frac{2H - y}{c_s} \right).$$

(3.104)

The force vector $F^f_{\infty}(t)$ due to the free-field motion of the unbounded domain is expressed as [166]

$$F^f_{\infty}(t) = \int_0^t S_{\infty}(t - \tau) u^b_{\infty}(\tau) d\tau = C_{\infty} \ddot{u}^b_{\infty}(t) + K_{\infty} u^b_{\infty}(t) + \int_0^t S^f_{\infty}(t - \tau) u^b_{\infty}(\tau) d\tau, \quad (3.105)$$

which is similar with the expression of the interaction force $R_{b}(t)$ in Eq. (3.93) by replacing $u_b$ with $u^b_{\infty}$.

The solution dependent terms at the right hand side of Eq. (3.100), i.e., the instantaneous responses $C_{\infty} \ddot{u}^b_{\infty}(t)$ and $K_{\infty} u^b_{\infty}(t)$ in $R_{\infty}(t)$, can be re-arranged to the left-hand side and assembled with the near-field system. By such an operation, the equation of motion of a near-field/far-field coupling system experiencing a free-field motion can be obtained.
as

\[
\begin{bmatrix}
    M_{ss} & M_{sb} \\
    M_{bs} & M_{bb}
\end{bmatrix}
\begin{bmatrix}
    \ddot{u}_s(t) \\
    \ddot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
    C_{ss} & C_{sb} \\
    C_{bs} & C_{bb} + C_{\infty}
\end{bmatrix}
\begin{bmatrix}
    \dot{u}_s(t) \\
    \dot{u}_b(t)
\end{bmatrix}
+ \begin{bmatrix}
    K_{ss} & K_{sb} \\
    K_{bs} & K_{bb} + K_{\infty}
\end{bmatrix}
\begin{bmatrix}
    u_s(t) \\
    u_b(t)
\end{bmatrix}
= \begin{bmatrix}
    0 \\
    F^f_b(t) + F^f_{\infty}(t) - \int_0^t S^e_\infty(t - \tau) u_b(\tau) d\tau
\end{bmatrix}.
\]

(3.106)

3.4 Conclusion

This chapter briefly introduces the basic concepts and key equations of the SBFEM in modeling the near-field, far-field and their coupling. The derivations of the polyhedral and two-dimensional unbounded elements are presented. The equation of motion of a two-dimensional near-field/far-field coupling system subjected to vertically propagating S-waves is summarized.

Polyhedral elements allow an arbitrary number of surfaces/nodes, which offers great flexibility in terms of spatial discretization. A scaling center is selected within a polyhedral element from where every node on the boundary is directly visible. The boundary of a polyhedral domain is discretized as surface elements that are used in standard FEM, i.e., triangular and quadrilateral shapes. A three-dimensional scaled boundary coordinate system \((\eta, \zeta, \xi)\) is established in each volume sector scaled from the scaling center \((\xi = 0)\) to the surface element \((\xi = 1)\). Using SBFE approach, the solution on surface elements is interpolated whereas the solution inside a polyhedral element is obtained analytically. The scaled boundary finite element equation can be formulated using nodal displacement functions \(u(\xi)\) and solved by applying an eigenvalue decomposition.

Two-dimensional unbounded elements only require a small number of line elements discretized at the boundary. A scaling center (outside the domain) is selected from which the total boundary is visible. The scaled boundary coordinate system \((\eta, \xi)\) is established
in each sector scaled from the line element ($\xi = 1$) to infinity ($\xi = \infty$). The dynamic stiffness $S_\infty(t)$ of an unbounded element can be formulated in terms of a displacement unit-impulse response matrix in the time domain. Interaction forces $R_b(t)$ are induced at the near-field/far-field interface, they are calculated from a convolution integral of the product of displacement unit-impulse response matrix and the historical displacement.

The equation of motion of a near-field/far-field coupling system experiencing a free-field motion is established by i) assembling the matrices of polygonal and unbounded elements and ii) introducing the interaction and seismic equivalent nodal forces. The seismic equivalent nodal forces consist of two contributions due to the wave propagation on the near-field boundary and free-field motion of the unbounded domain.
Chapter 4

AB AQUS implementation of SBFEM

In this chapter\(^1\), three-dimensional polytope (polyhedral) elements and two-dimensional unbounded elements derived by the SBFEM are implemented into the commercial finite element software package ABAQUS as user elements (UEL). Section 4.1 presents an introduction explaining the advantages of the SBFEM and the purposes of the implementation. In Section 4.2, the implementation of polyhedral user elements are explained in details. The workflow and ABAQUS input file formats involving UELs are demonstrated. Although the seismic analysis is two-dimensional in this thesis, the implementation is still provided in three-dimensional cases since the topology of polyhedral elements is more complex. The technique of recovering the topology of polyhedral elements in the implementation is presented. Besides, a detailed explanation of the definition of element-based surfaces needed for contact modeling is given. The implementation of two-dimensional unbounded elements, specially the definition of the output arrays, is presented in Section 4.3. Section 4.4 presents numerical examples include academic benchmark tests and more complex structures clearly demonstrating the main advantages of the SBFEM. Finally, important conclusions are provided in Section 4.5.

\(^1\)This chapter expands upon the article published by the candidate in Ref. [89]. Co-authors of this publication are acknowledged.
4.1 Introduction

The finite element method (FEM) has been widely used in engineering and science for solving partial differential equations arising in many different areas of application. To this end, the FEM discretizes a problem domain into a number of subdomains of simple shape, so-called finite elements, and assembles them to derive numerical solutions. Geometrically simple elements are commonly utilized in the standard FEM, e.g., tetrahedral and hexahedral elements in three-dimensional applications. Many commercial finite element software packages are available, such as ANSYS, ABAQUS, MARC, etc. which makes the FEM easily accessible for engineers and researchers alike. Nonetheless, there are some aspects that the FEM is unable to handle in a satisfactory manner: a) stress singularities occurring in fracture mechanics problems [281] and b) the representation of the unbounded domains [282]. Additionally, mesh generation is still laborious and time-consuming and therefore, often constitutes a severe bottleneck in the analysis pipeline. Considering special data formats, such as digital images and stereolithography (STL), which are nowadays widely adopted by industry, this problem is prominent. To alleviate the aforementioned issues, various alternatives have been proposed by researchers, to name a few, the boundary element method (BEM) [151], the scaled boundary finite element method (SBFEM) [109], and the extended finite element method (X-FEM) [238].

The SBFEM is in its core nature a semi-analytical method which was first proposed by Wolf and Song [108, 109] for modeling wave propagation in unbounded domains. However, over the last two decades, the SBFEM has been extended to various types of analyses including wave propagation in bounded domains [165], fracture mechanics [283, 219, 284], acoustics [285, 286], contact mechanics [86, 88], seepage [287], elastoplasticity [288], damage analysis [242], adaptive analysis [289, 290], among many others [291, 292, 293, 294]. Due to its versatility and applicability to a wide class of problems, the SBFEM can be seen as a general numerical method to solve PDEs.

Due to the following advantages of the SBFEM over standard FEM, this method is used for the seismic analysis of post-tensioned gravity dams. Firstly, the unbounded
foundation can be accurately modeled by the SBFE approach by employing displacement unit-impulse response matrices [23]. An unbounded element formulated by the SBFEM (unbounded S-element) only requires the discretization of the near-field/far-field interface. It can rigorously capture the radiation condition at infinity and transfer the free-field motions as equivalent seismic inputs [166, 66]. Moreover, polytope elements derived by the SBFEM (polygonal/polyhedral S-elements) offer great flexibility regarding spatial discretization for the dam and near-field foundation. They are highly complementary with quadtree/octree meshes [84, 295] allowing smooth and rapid transitions in element size. Local mesh refinement can be easily performed at the regions where high-stress gradients are expected, such as the regions near the anchor head and the bond length, to obtain accurate solutions. Last but not least, polytope S-elements provide the convenience of generating conforming interfacial meshes for the simulations of the structural interfaces [231] and bond-slip interaction. The non-linear behavior of the weak interfaces can be simulated by a cohesive-frictional scheme, i.e., inserting cohesive elements in conjunction with establishing contact pairs at the interfaces. This scheme has been proved to be a reliable tool to model weak interfaces and can be conveniently operated on conforming meshes [86, 89]. The bond-slip interaction can be represented by linkage [90] or interface elements [91, 92, 93] that connect the anchor and structure. These elements have to be implemented on conforming meshes [87].

Despite the aforementioned advantages of the SBFEM, the popularity of this method is still restricted to a small part of the scientific community. One important reason is seen in the more involved theoretical derivation of the SBFEM compared to the FEM. The FEM is practically taught in any engineering program at universities all over the world, concise mathematical theories on its properties exist, and powerful commercial software packages are readily available, while the SBFEM has been so far only implemented for research purposes. With the implementation of the SBFEM into a commercial software package, experienced analysts will have easy access to SBFEM and can exploit its advantages. It is our intent to facilitate the acceptance of the method itself and spark its further
development to other classes of problems. Considering scholars interested in starting research on the SBFEM, the availability of the implementation will provide a user-friendly and robust platform and comprehensive access to non-linear solvers.

The possibility of implementing other numerical methods which are related to the FEM into commercial software packages provides a whole new set of possibilities and can be realized through programming user element subroutines. Many commercial FEM codes support this functionality, e.g., USELEM in MARC [297], and UEL in ABAQUS [18]. In recent years, many non-standard numerical methods have been implemented in ABAQUS. Before the X-FEM was available as an additional module of commercial software packages, it was implemented in the form of a user element [298, 299, 300]. Elguedj et al. [301] introduced the user element implementation of NURBS based on isogeometric analysis. The cell-based smoothed finite element method (CSFEM) was implemented as user elements in ABAQUS by Cui et al. [302] and Kumbhar et al. [303] for alleviating mesh distortion issues seen in standard finite elements. Yang et al. [304] implemented the SBFEM in ABAQUS and showed the benefits of using polytope (polygon/polyhedron) elements in linear elastic stress analyses.

In the chapter, linear elastic polyhedral element and two-dimensional unbounded S-element are implemented into ABAQUS through UEL. To be consistent with ABAQUS’ element library, the surfaces of polyhedral element are discretized only as triangular and quadrilateral shapes, and only low-order (linear and quadratic) finite elements are used for the surface tessellation in this section. In this way, the SBFEM and FEM can be easily coupled since the discretization of element surfaces is identical. In addition, another ABAQUS’ user subroutine, UEXTERNALDB to manage external databases, is involved in the current implementation. Because of the arbitrariness of polyhedron in terms of topology, the nodal connectivity of the polyhedral user element should be recovered by a supplementary input file. The UEXTERNALDB is used to open the supplementary input file and store the polyhedral mesh information in a memory-efficient way.

The main goal of this chapter is to verify the implementation, and illustrate the advan-
tages of polyhedral elements on interfacial problems, and of two-dimensional unbounded S-elements on satisfying the radiation conditions of unbounded media. For this type of applications, some salient features of ABAQUS can be utilized, such as i) the comprehensive approaches for the contact modeling, ii) enriched library of cohesive elements for the modeling of interfacial behaviors, and iii) powerful non-linear solution techniques. It is also our belief that ABAQUS can benefit from the salient features of the SBFEM in different aspects mentioned in the previous paragraphs. In addition to a more flexible meshing paradigm provided by the SBFEM, the performance of ABAQUS can be enhanced for further specific applications. ABAQUS models containing non-matching interface meshes, e.g., fail to pass the patch test both in domain decomposition [231] and contact [86, 88] problems, especially when the interfaces are curved. Using polytope elements, the theoretical results are recovered again, as we can easily generate matching surfaces for a wide variety of cases.

4.2 Implementation of polyhedral elements

The implementation of SBFEM user elements involves two user subroutines. One is required to manage external databases (UEXTERNALDB) provided by the user, while the other defines the specific user element (UEL). They are written in FORTRAN [18] and incorporated into one single source code (*.for). The subroutine UEXTERNALDB imports the polyhedral mesh data from a supplementary input file and stores the mesh information in several user-defined COMMON blocks. The subroutine UEL performs the computations of the matrices and load vectors of SBFEM user elements.

In this work, both the pre- and post-processing are executed outside of ABAQUS. The pre-processing, performed by an in-house code, prepares two files for the simulation, an ABAQUS input file (*.inp) and a text file (*.txt) for describing the topology of the defined user elements. There are also commercial software packages available for generating polyhedral meshes and can be used together with our source code, to name a few,
STAR-CCM+ [305], ANSYS Fluent [306], and VoroCrust [307]. As the visualization of user elements is not directly supported by ABAQUS CAE, we output the nodal displacements through ABAQUS and write the result to external files which can be visualized by ParaView [308]. ParaView is a popular visualization tool implemented on the Visualization ToolKit (VTK). The VTK is a software system that supports a variety of visualization algorithms including scalar, vector, tensor, which are compatible with the analysis results (nodal displacements, stresses, etc.).

At the end, four files, i.e., the source code containing UEXTERNALDB and UEL, an ABAQUS input file, a text file storing the mesh data, and a user-defined include file (*.inc) specifying the dimensions of arrays in the COMMON blocks, should be stored in the working directory in order to run an analysis in ABAQUS featuring the proposed user element. The actual analysis is carried out by executing the following command:

```
abaqus job=<input file name> user=<fortran file name>.
```

In this section, detailed explanations concerning the implementation are given. For a better understanding of the analysis process in ABAQUS involving user subroutines, a workflow considering the general static analysis step is provided. The input file formats, including the ABAQUS input file and the supplementary text file, are demonstrated explicitly by means of an example, followed by the data structures for storing polyhedral elements. The structure and detailed descriptions of the UEL are presented as well. In addition, a method for defining element-based surfaces for SBFEM user elements is proposed. These surfaces are needed to establish interactions for interfacial problems.

### 4.2.1 General workflow of ABAQUS analysis involving user subroutines

The general workflow of an ABAQUS analysis involving SBFEM user subroutines is depicted in Fig. 4.1. Only the simulation pipeline is included in the flowchart, neglecting both pre- and post-processing procedures. The simulation procedure is similar to a stan-
standard ABAQUS analysis except for the fact that two user subroutines UEXTERNALDB and UEL are called during run-time. Before the actual analysis starts, UEXTERNALDB is called for importing the polyhedral mesh, and UEL is called for the pre-calculation and storage of the stiffness and mass matrices. In each increment of the actual analysis, the pre-calculated stiffness and mass matrices are read by UEL, and directly used for the definition of required output arrays AMATRX and RHS. During the iteration procedure, the UEL is called again to check the convergence of the solution.

In the present implementation, the SBFEM user element is linear elastic. To avoid repetitive calculation of the stiffness matrices of the user element in a non-linear analysis, a pre-calculation analysis step (general static) is created before the actual load steps. In such a step, there is only one increment with no external loading. It converges after one iteration. The stiffness and mass matrices are calculated and stored to provide the required output arrays of UEL in the iterative analysis of actual load steps. The implementation is presented in Section 4.2.4.

4.2.2 Input file formats

The input formats are illustrated by means of a simple example consisting of two polyhedral elements, as depicted in Fig. 4.2. The two polyhedral elements are comprised of 11 nodes and 12 surface elements, respectively. Element 1 consists of six square-shaped surface elements with eight unique nodes, while Element 2 features seven surface elements (four triangles, three quadrilaterals) with seven nodes. The ABAQUS input file involving these SBFEM user elements is depicted in Fig. 4.3. The text file being read by the user subroutine UEXTERNALDB is shown in Fig. 4.4. For the sake of simplicity, only the first three nodes and surfaces are included in the listing.

4.2.2.1 ABAQUS input file format

The ABAQUS input file consists of the following information: nodal coordinates, SBFEM user elements definition, material properties, boundary conditions, and the analysis type.
Figure 4.1. General workflow for ABAQUS/Standard analysis involving SBFEM user subroutines
Compared to a standard ABAQUS analysis using default element types, only the formats for defining user elements and their material properties are different.

![Figure 4.2. A polyhedron mesh with two elements](image)

Figure 4.3 lists the definition used for the SBFEM user elements depicted in Fig. 4.2. With the keyword `*USER ELEMENT`, a new type of user element is declared. Then additional information regarding this element type, such as the number of nodes (NODES), the type name (TYPE), the number of property values (PROPERTIES), the number of degrees of freedom (DOFs) per node (COORDINATES), and the number of solution-dependent variables (VARIABLES) are given. In the example, two user element types U8 and U7 are defined. In the next line, the DOFs per node are specified. In our examples, we use three-dimensional elements and therefore, each node has three translational DOFs. In ABAQUS, 1 denotes a displacement DOF in x-direction, while 2 and 3 repre-
sent the corresponding displacement DOFs in y- and z-directions, respectively. The definition of individual elements is similar to the standard format starting with the keyword *ELEMENT. Here, only the element type that usually belongs to the standard ABAQUS element library is replaced by our user element type. The following line creates a single element by listing the element number and its nodes. The element property definition starts with the keyword *UEL PROPERTY, then specifies the element set (ELSET) it is assigned to. In the example, the element set is SBFES which includes Element 1 and Element 2 defined in the user element types U8 and U7, respectively. The property values are provided in the following line representing Young’s modulus, Poisson’s ratio and mass density. In our implementation, the solution-dependent variables contains static residual of the previous increment, static residual of the current increment, stiffness matrix, and mass matrix. Therefore for a user element with \( n \) DOFs, the value of VARIABLES should be \( n + n + n^2 + n^2 \).

Among the various definitions, two points deserve special mentions:

1. The SBFEM user element type is essentially determined by the number of nodes. For a two- or three-dimensional problem, the number of element nodes determines the number of DOFs of each element and further the dimensions of the output arrays of the UEL. The SBFEM user elements having the same number of nodes can be classified as the same type, regardless of the element topology.

2. There is no requirement for the node ordering for one SBFEM user element, i.e., it can be arbitrary. For an ABAQUS built-in element, the node ordering should follow specific rules because it defines the topology of this element type. However, it is impossible to set a universal principle of describing the topology of the polyhedral element based only on the node ordering. The element topology will be described in a supplementary input text file introduced in Section 4.2.2.2. The node ordering of the SBFEM user element only affects the location vector for the assembly of the global stiffness matrix and force vector.
4.2.2.2 Text file format

As mentioned above, the ABAQUS input file only lists the node identifiers of each element, which is insufficient to describe the topology of polyhedral elements. Therefore, it is compulsory to provide additional information for the SBFEM user elements through a supplementary input file. There are four types of information in the supplementary file, shown in Fig. 4.4, i.e., (i) nodal coordinates, (ii) surface information, (iii) element information, and (iv) scaling center coordinates. At the beginning of each type of information, one integer is declared to identify the number of the specific entity such as nodes, surfaces, and elements. For each surface, one integer indicating the number of nodes is declared followed by the nodal connectivity of this surface, either clockwise or counter-clockwise. For the definition of a polyhedron, the number of surfaces is stated first, and then the identifiers of the surfaces enclosing the polyhedron are listed. The sign of surface ID is positive if the normal vector of the surface is pointing outward of the cell, otherwise it is negative.

| 11 | %Number of nodes |
| 0 0 0 | %Nodal coordinates |
| 1 0 0 |
| 1 1 0 |
| ... |
| 12 | %Number of surfaces |
| 4 1 2 3 4 |
| 4 1 4 8 5 |
| 4 1 2 6 5 |
| ... |
| 2 | %Number of elements |
| 6 -1 -2 3 4 5 6 |
| 7 -6 -7 8 9 10 11 12 |
| 2 | %Number of elements |
| 0.5 0.5 0.5 |
| 0.4 0.6 1.5 |

Figure 4.4. An example of text input format describing SBFEM polyhedral mesh
4.2.3 Data structure of the mesh information

The user subroutine UEXTERNALDB is the interface subroutine which allows communication between other programs and user subroutines within ABAQUS/Standard. In our implementation, it only plays a role at the beginning of the analysis. UEXTERNALDB reads the text file describing the SBFEM discretization and stores the mesh data into user-defined COMMON blocks. The mesh information allows the UEL to construct the element topology for defining the stiffness matrix $K$, the mass matrix $M$, etc.

Three COMMON blocks are created when the subroutine UEXTERNALDB is called. They are named as NDINFO, SFINFO and ELINFO, for storing the node, surface and element information, respectively. The explicit data structure of each block for the previous example is shown in Table 4.1.

Table 4.1. Data structure stored in the COMMON blocks

<table>
<thead>
<tr>
<th>NDCRD</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0 0.0 0.0</td>
</tr>
<tr>
<td>1.0 0.0 0.0</td>
</tr>
<tr>
<td>1.0 1.0 0.0</td>
</tr>
<tr>
<td>...</td>
</tr>
</tbody>
</table>

(a) Node information block NDINFO

<table>
<thead>
<tr>
<th>SFCONN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 3 4 1 4 8 5 1 2 6</td>
</tr>
</tbody>
</table>

(b) Surface information block SFINFO

<table>
<thead>
<tr>
<th>SFINDEX</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 8 12 16 20 24 28 31 34 38 41 44</td>
</tr>
</tbody>
</table>

(c) Element information block ELINFO

| ELINDEX | CTCRD |
|-------------------------|
| 6 13 |
| 0.5 0.5 0.5 |
| 0.4 0.6 1.5 |

The COMMON block NDINFO contains a matrix named NDCRD with dimension
$n_{nd} \times 3$ to store nodal coordinates, in which $n_{nd}$ denotes the number of nodes.

The COMMON block SFINFO contains two vectors named SFCONN and SFINDEX, respectively. The vector SFCONN stores the nodal connectivity of each surface in a continuous fashion. An indexing vector SFINDEX is stored to recover the individual surface elements. It has $n_{sf}$ entries, where $n_{sf}$ represents the number of surfaces. Each entry of SFINDEX indicates the final storage position in SFCONN for each surface. The number of nodes for one surface can be calculated by the difference between the positioning entries of itself and the former surface. The first entry of SFINDEX is the number of nodes for the first surface as well.

The COMMON block ELINFO contains a vector named ELCONN, a vector named ELINDEX, and a matrix named CTCRD with dimension $n_{el} \times 3$, where $n_{el}$ is the number of elements. Similar to SFCONN, ELCONN stores the surfaces enclosing each element continuously, by the order of element ID. The indexing vector ELINDEX has $n_{el}$ entries, which identify the final storage position of each element in ELCONN. The matrix CTCRD stores the scaling center coordinates of each element. For a subdomain without cracking, the scaling center may be calculated as the average coordinates of all nodes in each element. However, for cracked subdomain used in fracture mechanics, the scaling center should be specified as the crack tip [283]. To make the implementation more versatile, we purposely decided to provide the scaling center as additional input parameter.

The proposed scheme for storing polyhedral meshes is comparably memory-efficient. Only the matrices for storing coordinates, i.e., NDCRD and CTCRD, are provided as float data type while the components of other vectors are all integers. To minimize the memory requirements, we opted to store the surface and element information in vectors instead of matrices. Note that the number of nodes of each surface is not uniform and the polyhedral elements have arbitrary surfaces, which may induce zero entries in matrices. This issue can be avoided by using vectors to store the surface and mesh information.
4.2.4 User element

The user subroutine UEL is used to define a special element type not available in ABAQUS. It must provide all element level calculations, such as the definition of the stiffness and mass matrices as well as the residual force vector. To this end, ABAQUS provides some input variables to the UEL, such as the current element number JELEM, nodal coordinates COORDS, material properties PROPS, current nodal displacement vector U, and current solution procedure type LFLAGS. Those variables are defined in the ABAQUS input file or generated automatically according to the simulation procedure. The most important arrays to output by the UEL are AMATRX and RHS. For static analysis, AMATRX and RHS are the current stiffness matrix and residual force vector, respectively. In other analysis types, these generic arrays have different meanings. Detailed definitions of AMATRX and RHS for different solution procedures are out of the scope of the current section. The reader is referred to the ABAQUS user subroutine reference manual [18] for a comprehensive discussion.

The flowchart of the UEL subroutine implementing SBFEM is shown in Fig. 4.5. In the following, the functions and specific implementations of each process are explained in detail:

1. Localization of nodal coordinates and construction of element topology: In this step, the global nodal coordinates are transformed into relative nodal coordinates with respect to the scaling center, and the element topology is constructed using local nodal identifiers. Two matrices named as RCRD and SELE are generated for storing the local nodal coordinates and the element topology, respectively. The global nodal coordinates \((x, y, z)\) of the nodes belonging to the current element are passed into UEL as input matrix COORDS. They are transformed into relative nodal coordinates \((\hat{x}, \hat{y}, \hat{z})\) with respect to the scaling center, which are defined as in Eq. (3.4). The sequence of the nodal coordinates conforms to the node ordering of each element given in the ABAQUS input file. The element topology can be extracted from the COMMON blocks when the element number JELEM is
1. Localization of nodal coordinates and construction of element topology
2. Calculation of elasticity matrix $D$
3. Preparation of shape function matrix $N_u$, Jacobian matrix $J_b$ and strain-displacement matrices $B_1$ and $B_2$
4. Integration of surface coefficient matrices $E_0^e$, $E_1^e$, $E_2^e$ and $M_0^e$, assembly of $E_0$, $E_1$, $E_2$ and $M_0$
5. Calculation and eigenvalue decomposition of coefficient matrix $Z_p$
6. Formulation and storage of stiffness matrix $K$ and mass matrix $M$
7. Definition of output arrays AMATRX and RHS

Figure 4.5. Flowchart of user subroutine UEL for SBFEM
known. Note that the connectivity is given in terms of the global nodal identifiers. To meet the global assembly requirement, the calculation of AMATRX and RHS should match the node ordering given in the ABAQUS input file. Therefore, the element topology should be described using local nodal identifiers. Matching the nodal coordinates stored in COORDS and NDINFO, the correspondence between local and global nodal identifiers is obtained. Then, the element topology can be re-constructed by local nodal identifiers. During the construction of the element topology, the nodal connectivity of negative surfaces should be reordered for the mapping consideration. The RCRD and SELE for Element 2 (depicted in Fig. 4.2) are shown explicitly in Table 4.2, in which $n_{nd}^s$ represents the node number of each surface.

Table 4.2. Data structures of relative nodal coordinates and element topology

<table>
<thead>
<tr>
<th>Global</th>
<th>Local</th>
<th>$\hat{x}$</th>
<th>$\hat{y}$</th>
<th>$\hat{z}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>-0.4</td>
<td>-0.6</td>
<td>-0.5</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>0.6</td>
<td>-0.6</td>
<td>-0.5</td>
</tr>
<tr>
<td>7</td>
<td>3</td>
<td>0.6</td>
<td>0.4</td>
<td>-0.5</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>-0.4</td>
<td>0.4</td>
<td>-0.5</td>
</tr>
<tr>
<td>9</td>
<td>5</td>
<td>-0.4</td>
<td>-0.6</td>
<td>0.5</td>
</tr>
<tr>
<td>10</td>
<td>6</td>
<td>0.6</td>
<td>0.4</td>
<td>0.5</td>
</tr>
<tr>
<td>11</td>
<td>7</td>
<td>-0.4</td>
<td>0.4</td>
<td>0.5</td>
</tr>
</tbody>
</table>

(a) Node ID correspondence  
(b) RCRD

<table>
<thead>
<tr>
<th>$n_{nd}^s$</th>
<th>Local nodal connectivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>4 3 2 1</td>
</tr>
<tr>
<td>4</td>
<td>5 7 4 1</td>
</tr>
<tr>
<td>3</td>
<td>1 2 5 0</td>
</tr>
<tr>
<td>3</td>
<td>2 3 6 0</td>
</tr>
<tr>
<td>4</td>
<td>3 4 7 6</td>
</tr>
<tr>
<td>3</td>
<td>2 6 5 0</td>
</tr>
<tr>
<td>3</td>
<td>5 6 7 0</td>
</tr>
</tbody>
</table>

(c) SELE

2. Calculation of the elasticity matrix $D$: The material properties such as Young’s modulus, Poisson’s ratio and mass density are stored in the input array named PROPS. Using Young’s modulus and Poisson’s ratio, the elasticity matrix $D$ can
be formulated as in Eq. (3.45).

3. Preparation of matrices $N_u$, $J_b$, $B_1$ and $B_2$: The shape function matrix $N_u$ depends on the surface element type, which can be either a triangular or quadrilateral element, with linear or quadratic shape functions. The surface element type has been defined in the matrix SELE generated in the previous step. Extracting the nodal coordinate vectors $\hat{x}$, $\hat{y}$, and $\hat{z}$ of the surface from the matrix RCRD, together with the shape function matrix $N_u$, the Jacobian matrix $J_b$ is formulated following Eqs. (3.10), (3.13) and (3.15). After computing the matrices $b_1$, $b_2$ and $b_3$ (see Eq. (3.33)) based on $J_b^{-1}$, the matrices $B_1$ and $B_2$ are calculated by Eq. (3.43).

4. Integration of $E_0$, $E_1$, $E_2$ and $M_0$: The integrals to determine the coefficient matrices of the individual surface elements, see Eqs. (3.52), (3.68) and (3.63), are solved using numerical quadrature rules, in our case, a standard Gaussian quadrature rule. Then the coefficient matrices $E_0$, $E_1$, $E_2$ and $M_0$ of the entire element are assembled from the coefficient matrices $E^c_0$, $E^c_1$, $E^c_2$ and $M^c_0$ of all surface elements.

5. Calculation and eigenvalue decomposition of $Z_p$: The expression of the coefficient matrix $Z_p$ is provided in Eq. (3.76), which can be computed based on $E_0$, $E_1$, $E_2$ and $E_0^{-1}$. For computing the inverse of $E_0$, two subroutines DGETRF and DGETRS from the Linear Algebra PACKage (LAPACK) [309] are called. DGETRF prepares the LU factorization of $E_0$ for DGETRS to compute $E_0^{-1}$. The eigenvalue decomposition of $Z_p$ is implemented by calling the subroutine DGEEV of LAPACK. DGEEV provides a matrix and two vectors as output parameters. The matrix alternately stores the real and imaginary parts of the right eigenvectors in its columns. The two vectors store the real and imaginary parts of eigenvalues, respectively. They are essentially the eigenvector matrix and eigenvalues in Eq. (3.77), but in the DGEEV output format. The real parts of eigenvalues are sorted by the descending order through bubble sort algorithm [310]. The imaginary parts of eigenvalues and eigenvectors are sorted also following the sorted real parts of the eigenvalues.
Combining real parts and imaginary parts, the eigenvector matrix $\phi$ and the diagonal matrix of eigenvalues $\lambda$ given in Eq. (3.79) are obtained. They are complex matrices and stored with double complex data type. The sub-matrices $\phi_{u1}$, $\phi_{q1}$ and $\lambda^+$ can be extracted easily from $\phi$ and $\lambda$.

6. Formulation and storage of $K$ and $M$: The stiffness matrix $K$ can be calculated by Eq. (3.83), in which the calculation of the inverse of sub-matrices $\phi_{u1}$ can be done analogously to the inversion of $E_0$ discussed in Step 5. The mass matrix $M$ is computed step by step through implementing the Eqs. (3.84)–(3.88). At the end of the first increment of an analysis, the stiffness and mass matrices are stored in the array SVARS of the UEL for later use.

7. Definition of AMATRX and RHS: Generally speaking, for all types of solution procedure related to solid mechanics, we can defined the output arrays AMATRX and RHS based on the existing calculated arrays and input arrays. AMATRX is always defined as stiffness matrix $K$, mass matrix $M$, damping matrix $C$ or a combination thereof obeying specific rules. The damping matrix $C$ of a linear user element can be defined by Rayleigh damping [18]

$$C = \alpha_R M + \beta_R K,$$

(4.1)

where $\alpha_R$ and $\beta_R$ are the user-specified Rayleigh damping coefficients. RHS is always defined through the internal force vector or its combination with the inertia force vector obeying specific rules. The internal force vector can be obtained by

$$F_S = KU,$$

(4.2)

where $U$ is the current nodal displacement vector, one input array of the UEL. For linear static problem, RHS is formulated as $-F_S$. The inertia force vector can be
calculated as

\[ \mathbf{F}_I = \mathbf{M} \mathbf{A}. \]  \hspace{1cm} (4.3)

\( \mathbf{A} \) is the current nodal acceleration vector, and it is also one input array of the UEL. The inertia force vector \( \mathbf{F}_I \) contributes to RHS in dynamic problems.

8. Reading of matrices (\( \mathbf{K}, \mathbf{M} \)) stored in Step 6: For the increments except for the first one, the stiffness and mass matrices are retrieved directly from the array SVARS.

### 4.2.5 Element-based surface definition

The element-based surface definition for the SBFEM user element is important to take advantage of different features inherent to ABAQUS. By defining an element-based surface, distributed loads such as pressure and surface traction can be assigned to that surface. More importantly, for interfacial problems involving interactions such as tie constraints or contact definitions, it is a prerequisite to define surfaces to establish these interactions.

Currently, element-based surfaces can be created on solid, continuum shell, and cohesive elements in ABAQUS. On user elements, only node-based surfaces can be created, and they act only as slave surfaces and always use node-to-surface schemes in contact analysis, which is a severe shortcoming as the node-to-surface contact may fail the contact patch test even for a flat interface [88].

By overlaying standard elements with nearly zero stiffness on the SBFEM user elements, it is possible to define element-based surfaces for the SBFEM user elements. In the numerical examples, presented in the following section, the Young’s modulus of the overlaid elements is chosen in accordance with the value of the adjacent user element and scaled accordingly, i.e., \( E_{ov} = 10^{-16} E_{UEL} \). This methodology has originally been proposed in the ABAQUS manual [18] to visualize the user elements and is modified in this contribution to define element-based surfaces. The modification is related to the fact that instead of overlaying the entire polyhedral element with the standard finite elements only the volume sector needed for the surface definition are included. The volume sectors of
the three-dimensional SBFEM user elements are either of tetrahedral or pyramidal shape depending on the corresponding surface element type, i.e., either triangular or quadrilateral. Those shapes can be easily overlaid by standard elements from ABAQUS’ element library. As the scaling center fulfills the star-convexity requirement, i.e., the entire boundary is directly visible from this point, the process of creating this overlay mesh is robust for all types of polyhedrons. Hence, the element-based surfaces can be directly created on the overlaid standard elements as they share the DOFs with the corresponding SBFEM user elements. Note that this methodology does not affect the solution quality as the overlay elements contribute a negligible stiffness to the global system. Considering the shape functions on the surface of the SBFEM user elements it must be stressed again that they coincide with those of the elements from ABAQUS’ element library.

The operation of overlaying standard elements on the volume sectors of the SBFEM user element is illustrated in Fig. 4.6. This particular example is based on the SBFEM user element depicted in Fig. 4.6a which consists of six surfaces and eight nodes. Note that the scaling center is marked in red (Node 9). Here, a quadrilateral element-based surface is created by introducing a pyramidal overlay element that is superimposed on the volume sector created be surface element S1 and the scaling center. This procedure is straightforward as ABAQUS offers pyramidal finite elements such as the C3D5 element. The ABAQUS input file format for defining this element-based surface is given in Fig. 4.6b. It is important to keep in mind that in our implementation, the scaling center is not an actual node of the element and consequently, must be defined in the ABAQUS input file. The surface identifier S1 indicates the surface formed by the first four nodes of the pyramidal element [18]. If a triangular element-based surface is to be created, the corresponding volume sector must be superimposed by a standard tetrahedral element, e.g., the C3D4 element.
4.3 Implementation of two-dimensional unbounded elements

As discussed in Section 4.2, the UEL subroutine requires inputs such as nodal coordinates, nodal connectivity, material and element properties, and outputs to ABAQUS two arrays AMATRX (stiffness or other matrix) and RHS (right hand side force vector), which are determined by the analysis type. Following the mathematical procedures discussed in Section 3.2, the dashpot ($C_\infty$), spring ($K_\infty$) and lingering response ($S_{ij}^r$) matrices of unbounded S-element can be obtained for implementation in the UEL subroutine. Detailed explanations regarding the implementation procedures are given in the Section 4.2. It is worth mentioning that, a supplementary input file and re-construction procedure to recover the topology of polyhedral element are needed, which are not necessary for two-dimensional case since the node ordering in a counter-clockwise fashion is sufficient to describe the nodal connectivity. This section emphasizes the definitions of the output ar-
rays AMATRX and RHS of the unbounded S-element for the quasi-static analysis and implicit dynamic analysis using HHT-α time integration [96].

The ABAQUS implicit dynamic analysis utilizes HHT-α time integration [96]. This method introduces an integrator parameter \( \alpha (-\frac{1}{2} \leq \alpha \leq 0) \) [18] to adjust the numerical damping of the time integration, and employs the weighted sum of the displacement and velocity vectors from the previous and current increments. A modified equation of motion is obtained as:

\[
\mathbf{M}\ddot{\mathbf{u}}(t) + (1 + \alpha)(\mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t)) - \alpha(\mathbf{C}\dot{\mathbf{u}}(t - \Delta t) + \mathbf{K}\mathbf{u}(t - \Delta t)) = \mathbf{F}(t), \tag{4.4}
\]

in which \( t \) and \( \Delta t \) are the ending and incremental time of the current increment, respectively. Accordingly, the output arrays AMATRX and RHS related to implicit dynamic analysis are defined as [18]

\[
\text{AMATRX} = \frac{1}{\Delta t^2 \beta} \mathbf{M} + (1 + \alpha) \left( \frac{\gamma}{\Delta t \beta} \mathbf{C} + \mathbf{K} \right), \tag{4.5a}
\]

\[
\text{RHS} = -\mathbf{M}\ddot{\mathbf{u}}(t) - (1 + \alpha)(\mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t)) + \alpha(\mathbf{C}\dot{\mathbf{u}}(t - \Delta t) + \mathbf{K}\mathbf{u}(t - \Delta t)), \tag{4.5b}
\]

where \( \beta = \frac{1}{4}(1 - \alpha^2) \) and \( \gamma = \frac{1}{2} - \alpha \) are parameters of the time integration algorithm. All variables in Eq. (4.5) are known except for the damping matrix \( \mathbf{C} \). For linear user elements [18], the damping matrix can be calculated by \( \mathbf{C} = \alpha_R \mathbf{M} + \beta_R \mathbf{K} \), where \( \alpha_R \) and \( \beta_R \) are the user-specified Rayleigh damping factors.

### 4.3.1 Quasi-static analysis

A quasi-static analysis procedure should be established before the seismic analysis, for imposing static forces, such as the dam gravity, hydrostatic, prestressing of anchor, uplifts at weak interfaces. In such an analysis, ABAQUS manual states the definitions of AMATRX and RHS are identical to those of static analysis, i.e., AMATRX is defined as static stiffness matrix, and RHS is the reaction due to the displacement at the current state.
If the size of the near field is large enough, the interaction force-displacement relationship of the unbounded S-element in Eq. (3.93) for quasi-static analysis can be approximated as

\[
K_\infty + \int S'_r(\tau) d\tau \mathbf{u}_b = \mathbf{R}_b, \quad (4.6)
\]
in which we assume \(\dot{u}_\infty = 0\) and a constant \(u_b\) used for the integral. Applying the trapezoidal rule, the output arrays AMATRX and RHS of unbounded S-element for quasi-static analysis are defined as

\[
\text{AMATRX} = K_\infty + \frac{\Delta t}{2} (S'_0 + S'_n) + \Delta t \sum_{j=1}^{n-1} S'_j, \quad (4.7a)
\]
\[
\text{RHS} = - \left( K_\infty + \frac{\Delta t}{2} (S'_0 + S'_n) + \Delta t \sum_{j=1}^{n-1} S'_j \right) \mathbf{u}_b. \quad (4.7b)
\]

### 4.3.2 Dynamic analysis

The definition of the AMATRX of one element considers its contributions to the global mass, damping and stiffness matrices. According to Eq. (3.106), \(C_\infty\) and \(K_\infty\) are assembled to the damping (\(C\)) and stiffness (\(M\)) matrices of the coupling system. Regarding the right hand side force vector (RHS), its definition involves the responses due to acceleration, velocity and displacement. To employ the HHT-\(\alpha\) time integration, the force vectors due to the displacement and velocity should be weighted by \(1 + \alpha\) for the current increment and \(-\alpha\) for the previous increment. Eq. (3.100) can be re-arranged by merging all the responses related to the unbounded S-element, resulting in

\[
\begin{bmatrix}
M_{ss} & M_{sb} \\
M_{bs} & M_{bb}
\end{bmatrix}
\begin{bmatrix}
\ddot{u}_s(t) \\
\ddot{u}_b(t)
\end{bmatrix}
+
\begin{bmatrix}
C_{ss} & C_{sb} \\
C_{bs} & C_{bb}
\end{bmatrix}
\begin{bmatrix}
\dot{u}_s(t) \\
\dot{u}_b(t)
\end{bmatrix}
\begin{bmatrix}
K_{ss} & K_{sb} \\
K_{bs} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
u_s(t) \\
u_b(t)
\end{bmatrix}
= \begin{bmatrix}
0 \\
0
\end{bmatrix}
- \begin{bmatrix}
\mathbf{F}_b(t) \\
\mathbf{R}_b(t)
\end{bmatrix}, \quad (4.8)
\]
in which \(-R_s^b(t) = -(R_b(t) - F^f_{\infty}(t))\) is the overall responses of the unbounded S-element. \(R_s^b(t)\) is expressed by the scattered motion \(u_s^b(t) = u_b(t) - u_f^b(t)\) as

\[
R_s^b(t) = \int_0^t S_{\infty}(t - \tau)u_b^s(\tau) d\tau = C_{\infty}u_b^s(t) + K_{\infty}u_b^s(t) + \int_0^t S_{\infty}(t - \tau)u_b^s(\tau) d\tau. \tag{4.9}
\]

The responses \(R_s^b(t)\) depend on the displacement and velocity, but are independent of the acceleration. Employing the HHT-\(\alpha\) time integration, the output arrays AMATRX and RHS of an unbounded S-element for implicit dynamic analysis are defined as

\[
AMATRX = (1 + \alpha)\left(\frac{\gamma}{\Delta\bar{\nu}\beta}C_{\infty} + K_{\infty}\right), \tag{4.10a}
\]

\[
RHS = -(1 + \alpha)R_b^s(t) + \alpha R_b^s(t - \Delta t), \tag{4.10b}
\]

in which the integral term in \(R_b^s(t)\) can estimated by a trapezoidal rule in Eq. (4.11).

The definition of AMATRX is straightforward since all variables are given, while the computation of \(R_b^s(t)\) used in the RHS relies on the scatter motion which are not directly available. \(u_b^s(t)\) and \(\dot{u}_b^s(t)\) should be calculated by deducting \(u_f^b(t)\) and \(\dot{u}_f^b(t)\) from the current solutions of \(u_b(t)\) and \(\dot{u}_b(t)\), respectively. Each entry of \(u_b^s(t)\) and \(\dot{u}_b^s(t)\) for vertically propagated shear waves can be calculated using Eq. 3.101. The third term in \(R_b^s(t)\) can be computed by applying a trapezoidal rule like Eq. (3.97), as

\[
\int_0^t S_{\infty}(t - \tau)u_b^s(\tau) d\tau \simeq \frac{\Delta\bar{\nu}}{2} (S_0u_b^s(t) + S_nu_b^s(t - n\Delta\bar{\nu})) + \Delta\bar{\nu}\sum_{j=1}^n S_j^nu_b^s(t - j\Delta\bar{\nu}). \tag{4.11}
\]

in which \(u_b^s(t - j\Delta\bar{\nu})\) is the scatter motion at \(t - j\Delta\bar{\nu}\), \(\Delta\bar{\nu}\) is the time range of a subinterval, and \(n\) is the number of subintervals for the trapezoidal rule. Again, the entries of \(u_b^s(t - j\Delta\bar{\nu})\) are derived through Eq. 3.101. The total displacement vectors \(u_b(t - j\Delta\bar{\nu})\) should be obtained from the historical solutions of \(u_b\), i.e., the solutions of the previous increments.

Figure 4.7 depicts the procedure to calculate \(u_b(t - j\Delta\bar{\nu})\) using the historical solutions of \(u_b\). The red solid circles represent the numerical solutions of the previous increments. The dark solid squares are the displacements which should be used in the trapezoidal rule.
Figure 4.7. Extraction of $u_b$ for the trapezoidal rule based on historical solutions

In a dynamic analysis with automatic increment setting, $\Delta t$ might vary from different increments, and it differs from $\Delta \bar{t}$ (constant). Thus, the displacement vector $u_b(t - j\Delta \bar{t})$ should be obtained by interpolating the solutions of $u_b$ from the two adjacent increments. The numerical solutions of previous increments and their related time can be stored in a COMMON block. The stored solutions should have accumulative time not smaller than $n\Delta \bar{t}$ (the total truncation time of the trapezoidal rule). Besides, for an accurate interpolation, a maximum increment size of the analysis $\Delta t_{\text{max}} \leq \Delta \bar{t}$ should be set.

### 4.3.3 Equivalent seismic loads

As discussed in Section 4.3.3, the equivalent seismic forces consist of $F_{b}^f(t)$ and $F_{\infty}^f(t)$. $F_{\infty}^f(t)$ has been incorporated in $R_{b}^s(t)$, see Eq. (4.8), and defined in the RHS of the unbounded S-element. To complement the equivalent seismic forces, the wave propagation induced forces $F_{b}^i(t)$ acting on the near field boundary should be implemented.

The user subroutine UTRACLOAD to define surface tractions by specifying the magnitude ALPHA and directions ($T - \text{USER}$) on integration points are used for the implementation. According to Eqs. (3.103) and (3.104), the output arrays ALPHA and $T - \text{USER}$ for tractions induced by vertically propagated S-wave should be defined as

$$\text{ALPHA} = \sqrt{t_x^2 + t_y^2} = \frac{G}{c_s} \left( \frac{-\ddot{u}_s(t - \frac{y}{c_s}) + \ddot{u}_s(t - \frac{2H - y}{c_s})}{c_s} \right),$$

$$T - \text{USER} = \begin{bmatrix} \frac{t_x}{\sqrt{t_x^2 + t_y^2}} & \frac{t_y}{\sqrt{t_x^2 + t_y^2}} \end{bmatrix} \begin{bmatrix} n_y & n_x & 0 \end{bmatrix},$$
in which $y_i$ is the $y$-coordinate of an integration point. For a near-field foundation with rectangular shape, the normal vector is easily determined. For example, at the left boundary ($x_i = 0$), the normal vector is $\{-1, 0\}^T$, which results in $T_{\text{USER}} = \begin{bmatrix} 0 & -1 & 0 \end{bmatrix}$.

### 4.4 Numerical examples

The objective of this section is to verify the implementation and demonstrate the advantages of including SBFEM user elements. This is achieved by means of various numerical examples. In the first subsection, benchmark tests are performed to check the correct implementation. The second subsection presents examples considering the treatment of non-matching meshes. The third subsection includes examples that are related to automatically generated meshes based on image and STL data. In these cases, robust and efficient octree mesh generation algorithms are employed. The computational times reported are measured on a DELL workstation with an Intel Xeon E5-2637 CPU running Windows 10 Pro, 64 bit.

#### 4.4.1 Benchmark tests

This subsection is designated to verify the implementation of polyhedral user elements by performing static, modal, and transient analyses on simple geometries, and two-dimensional unbounded S-element by a uniform semi-infinite half space subjected to distributed pressure. A patch test is performed on a quadrangular prism under uniaxial tension in order to check the implementation of the stiffness matrix. Modal and transient analyses are performed on a cantilever beam to verify the implementation of the mass matrix. A semi-infinite half space subjected to distributed pressure is used to examine the performance of the unbounded S-element on satisfying radiation conditions of unbounded media.
4.4.1.1 Patch test

A quadrangular prism is discretized by five polyhedral elements as shown in Fig. 4.8. The dimensions are indicated in Fig. 4.8 with \( b = h = 1 \) m. The material constants are Young’s modulus \( E = 10 \) GPa and Poisson’s ratio \( \nu = 0.25 \). The left surface (nodes with \( x = 0 \)) of the quadrangular prism is constrained in \( x \)-direction, the front surface (nodes with \( y = 0 \)) is constrained in \( y \)-direction, and the bottom surface (nodes with \( z = 0 \)) is constrained in \( z \)-direction. A vertical force \( F = 1000 \) kN is applied at Nodes 19, 20, 21 and 22 on the top surface.

The theoretical solutions of displacement \( u_z \) and stress \( \sigma_z \) of the uniaxial tension problem are expressed as:

\[
\sigma_z = \frac{F}{b^2} = 1 \text{ MPa},
\]

\[
u_z = \left( \frac{\sigma_z}{E} - \frac{\nu \sigma_x}{E} \right) z = \frac{F}{E b^2} z = 0.0001 z.
\]

The corresponding contours are depicted in Fig. 4.9. The maximum nodal \( L^2 \) error norms of displacement \( u_z \) and stress \( \sigma_z \), which are calculated based on the analytical solutions in
Figure 4.9. Contours of a quadrangular prism under uniaxial tension

Table 4.3. Maximum nodal $L^2$ error norm of a quadrangular prism under uniaxial tension

<table>
<thead>
<tr>
<th></th>
<th>$u_z$</th>
<th>$\sigma_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L^2$ error norm</td>
<td>1.199e-14</td>
<td>1.695e-14</td>
</tr>
</tbody>
</table>

Eq. (4.13), are listed in Table 4.3. The displacement and stress results are considered to be accurate up to the machine precision.

4.4.1.2 Modal analysis of a cantilever beam

The second example is a three-dimensional cantilever beam with a depth of $d = 0.4\,\text{m}$, a thickness of $t = 0.2\,\text{m}$, and a length of $l = 1\,\text{m}$, where one end of the beam is clamped, as shown in Fig. 4.10a. The material properties are Young’s modulus $E = 1\,\text{MPa}$, Poisson’s ratio $\nu = 0.25$, and the mass density $\rho = 2000\,\text{kg/m}^3$.

Three-dimensional SBFEM user elements and 8-node linear brick elements (ABAQUS: C3D8) are assigned to the same mesh to compare their performances. In the following, we refer to the mesh assigned with ABAQUS library’s elements as ABAQUS model and the mesh assigned with SBFEM user elements as SBFEM model. Four meshes of the cantilever beam with different element sizes, i.e., 0.1 m, 0.05 m, 0.025 m and 0.0125 m, are generated and one is shown in Fig. 4.10b (mesh size: 0.05 m). The reference model
For all models, an eigenvalue analysis is performed using the default settings of ABAQUS’ solver [18]. The first ten eigenvalues for the different models are plotted in Fig. 4.11a. As can be observed, the first ten eigenvalues converge algebraically for both element types. The convergence in the relative error norm of eigenvalue $\lambda_{10}$ is also studied. The relative error of eigenvalue $\lambda_{10}$ is determined by $E_{\text{rel}} = (\lambda_{10} - \lambda_{10}^{\text{ref}})/\lambda_{10}^{\text{ref}}$, where $\lambda_{10}^{\text{ref}}$ is the 10th eigenvalue obtained from the reference model. The relative errors versus the number of nodes for the two types of models are plotted in Fig. 4.11b. The results are in good agreement, while the SBFEM models achieve slightly more accurate results.

4.4.1.3 Transient analysis of a cantilever beam

A transient analysis is performed on the three-dimensional cantilever beam structure introduced in the previous section. The boundary conditions and load history are shown in Fig. 4.12. A distributed line load $q(t) = 2\sin(2\pi t)$ (Unit: kN/m) acting on the top edge of the free end varies harmonically in time for 2.5 s, as depicted in Fig. 4.12b. As a monitor point, we choose the middle point on the lower edge of the free end of the cantilever beam as indicated in Fig. 4.12a. The vertical responses of the monitor point are recorded.

The responses of the SBFEM and ABAQUS reference models are compared. The mesh size of the SBFEM model is chosen as 0.025 m which is related to the modal analy-
Figure 4.11. Modal analysis comparison of a cantilever beam
Figure 4.12. A cantilever beam subjected to dynamic loading
sis results. It has been shown that the first ten eigenvalues are accurately captured by this model (error less than 1%). The reference model is identical to the one used in the previous section. The time stepping method is the Hilber-Hughes-Taylor (HHT) method [96] implemented in ABAQUS/Standard, which is an unconditionally stable method for linear problems. Again, the default settings for the transient (implicit dynamic) analysis [18] have been employed. A maximum time step size of 0.01 s has been chosen and the automatic time stepping of ABAQUS is activated. The vertical responses at the monitor point during the 2.5 s simulation time are obtained from the two models and plotted in Fig. 4.13. An excellent agreement is found as the difference between the results obtained from the SBFEM and ABAQUS reference models is negligible. Only in the acceleration response during the first 0.5 s small deviations are visible.

4.4.1.4 Wave source problem in a semi-infinite half space

A two-dimensional uniform half space has Young’s modulus of \( E = 2.5 \text{ GPa} \), Poisson’s ratio of \( \nu = 0.25 \) and mass density of \( \rho = 1000 \text{ kg/m}^3 \). A near-field foundation with dimension of \( 2b \times h \) is selected, where \( b = h = 2 \text{ m} \), as shown in Fig. 4.14. A time-varying uniformly distributed pressure \( q(t) \) is applied at the middle part (with a length of \( b \)) of the top surface. The distributed pressure \( q(t) \) is expressed as

\[
q(t) = \begin{cases} 
  t & 0 \leq t < 1 \text{ s}, \\
  2 - t & 1 \leq t < 2 \text{ s}, \quad (10^9 \text{N/m}) \\
  0 & t > 2 \text{ s}.
\end{cases}
\] (4.14)

Three models are established for the modeling of the semi-infinite half space: i) a model with a classical spring-dashpot boundary [140, 70], ii) a model with the implemented unbounded S-element, and iii) a reference model. The reference model has extended meshes of 20 m towards three directions (downwards, left and right). The near-field foundation is discretized as squares with a finite element size of 0.05 m for all
Figure 4.13. Response history of a 3D cantilever beam: vertical response at monitor point
Figure 4.14. A semi-infinite half space subjected to uniformly distributed pressure

models. Plane stress conditions are assumed. The schematic of the models with spring-dashpot boundary and unbounded S-element is shown in Fig. 4.15. The spring-dashpot boundary consists of parallel-connected spring and dashpot elements in both normal and tangential directions. The subscripts n and t of spring stiffness coefficient $K$ and dashpot damping coefficient $C$ denote that they are acting in normal and tangential directions, respectively. Those coefficients are formulated as [70]

$$
K_n = \frac{A_s G}{r_b}, \quad K_t = \frac{A_s G}{2r_b}, \quad C_n = A_s \rho c_p, \quad C_t = A_s \rho c_s,
$$

(4.15)

where $G$ is the shear modulus of the medium, respectively. $c_s = \sqrt{\frac{G}{\rho}}$ and $c_p = \sqrt{\frac{\lambda + 2G}{\rho}}$ are the wave velocities of the S- and P-waves, respectively. $A_s$ is the tributary area of a point at the boundary from its immediate neighboring elements, $r_b$ is the distance from the scattering wave source to the artificial boundary.

Dynamic implicit analyses are performed with the ABAQUS default setting (HHT-$\alpha$ time integration, $\alpha = -0.05$). A fixed time increment $\Delta t = 0.01$ s is applied for all three models. The displacement histories in 12 s of four monitor points A, B, C, D are examined and compared in Fig. 4.16. Before 10 s, there is negligible difference
Figure 4.15. Two models of a semi-infinite half space

(a) Spring-dashpot boundary

(b) Unbounded S-element boundary

Unbounded S-element
Figure 4.16. Displacement histories of monitor points of a semi-infinite half space

between results obtained from the model with unbounded S-element and the reference model. Thus, it can be conclude that the two-dimensional unbounded S-element has been implemented correctly. Besides, compared to the the model with spring-dashpot boundary, it is obvious that the model with unbounded S-element has more close results to the reference solutions. This example clearly shows that the unbounded S-element can rigorously capture the radian conditions of unbounded domains with a small truncation area.
4.4.2 Examples for non-matching meshes

Non-matching meshes often occur in finite element analyses involving domain decomposition [231] and contact mechanics [88] problems. Typically, special techniques are required to enforce the compatibility and equilibrium at the interfaces of non-matching meshes, which often introduce additional unknowns. Non-matching meshes in the finite element sense can be easily converted to matching meshes owing to flexibility provided by polyhedral SBFEM domains, which allows for elements with an arbitrary number of faces.

This section presents three examples to demonstrate the advantages of the matching meshes. The first example investigates the performance of the proposed elements in contact patch tests having curved interfaces. In the second example, a patch test including cohesive elements in conjunction with contact is presented. This example highlights the convenience of inserting cohesive elements at matching interfaces and verifies the behavior of cohesive elements combined with SBFEM user elements. The third example presents a cube with a spherical inclusion. Here, interaction states containing both contact and traction are examined and compared with the results obtained from non-matching meshes.

4.4.2.1 Patch test for contact analysis

ABAQUS/Standard provides comprehensive approaches for defining contact interactions: general contact, contact pairs and contact elements. For these approaches, there are different formulations available such as node-to-surface (NTS), or surface-to-surface (STS) techniques which are available for contact pairs [18]. Although many approaches can be utilized, the non-uniformity of the contact interface discretization may reduce the accuracy of results in ABAQUS [86, 88]. The performance of contact models in ABAQUS is expected to be enhanced when matching contact interfaces can be easily generated.

Three-dimensional contact patch tests are studied for the most commonly used NTS and STS schemes. The model is illustrated in Fig. 4.17 with dimensions defined as $W =$
6m and \( h = 3 \) m. It features a spherical contact interface between two blocks with a maximum deviation distance \( d \). The shape of the interface is described as

\[
z_i(x_i, y_i) = d \left[ 1 - \frac{2(x_i^2 + y_i^2)}{W^2} \right],
\]

where \( x_i, y_i, z_i \) are the coordinates of a point on the interface. The parameter \( d \) varies from 0 to 1.5 m to examine the contact performance depending on the actual shape of the curved contact interface. The model has a Young’s modulus of \( E = 10 \) MPa, and a Poisson’s ratio of \( \nu = 0.3 \). The contact has a friction coefficient of \( \mu = 0.5 \). The lower interface is selected as the master surface of the contact. Six degrees of freedom of the lower block (as indicated in Fig. 4.17) are fixed to prevent rigid body motions. A uniform pressure \( P = 1 \) MPa is applied at the bottom and top surfaces and consequently, a constant stress state \( \sigma_z = 1 \) MPa is expected in the contact patch test.

The basic set-up of the discretization for the model is shown in Fig. 4.17b, where it is observed that the top and bottom contact bodies are divided into nine and four elements, respectively. C3D8 elements are assigned to the non-matching meshes directly, as shown in Fig. 4.18a. The special treatment ‘slave tolerance’ in ABAQUS is used to detect the effective contact pairs for the ABAQUS models due to initial gaps or penetrations. Through re-meshing the interface, the hexahedral non-matching meshes are converted...
to polyhedral matching meshes as illustrated in Fig. 4.18b. The SBFEM user elements can be directly assigned to the matching meshes (see Fig. 4.18b) as they allow polyhedral shapes. Element-based surfaces can be created for the SBFEM models through the methodology proposed in Section 4.2.5, which are used to establish the contact pairs.

The stress contours of $\sigma_z$ (S33), obtained from the ABAQUS and SBFEM models (STS) are plotted in Figs. 4.19 and 4.20, respectively. The relative $L^2$ error norms for $\sigma_z$ are listed in Table 4.4, in which the maximum error norms of nodal stress are given. As shown in Fig. 4.19 and Table 4.4, for the ABAQUS models applying the STS scheme, only the model with flat contact interface obtains accurate result and passes the patch test. By increasing the curvature of the contact surfaces, the accuracy of the ABAQUS models deteriorates. This effect is attributed to initial gaps or penetrations in non-matching interface meshes which reduce the accuracy of the contact simulation. All ABAQUS models applying the NTS scheme fail the patch tests with substantial errors. In contrast, the SBFEM models provide accurate results (up to machine precision) and pass the patch tests for various curvatures using both STS and NTS schemes, as depicted in Fig. 4.20 and Table 4.4.
Table 4.4. $L^2$ Error norm of $\sigma_z$ of different models with different curvatures

<table>
<thead>
<tr>
<th></th>
<th>$d = 0$</th>
<th>$d = 0.5\text{ m}$</th>
<th>$d = 1\text{ m}$</th>
<th>$d = 1.5\text{ m}$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>STS</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ABAQUS model</td>
<td>&lt;1e-9</td>
<td>3.72e-3</td>
<td>1.31e-2</td>
<td>2.72e-1</td>
</tr>
<tr>
<td>SBFEM model</td>
<td>4.77e-14</td>
<td>4.10e-14</td>
<td>6.48e-14</td>
<td>5.02e-14</td>
</tr>
<tr>
<td><strong>NTS</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ABAQUS model</td>
<td>1.24e-1</td>
<td>1.16e-1</td>
<td>1.12e-1</td>
<td>1.11e-1</td>
</tr>
<tr>
<td>SBFEM model</td>
<td>5.13e-14</td>
<td>5.24e-14</td>
<td>6.76e-14</td>
<td>6.17e-14</td>
</tr>
</tbody>
</table>

Figure 4.19. Stress contours ($\sigma_z$) of ABAQUS models (Unit: Pa)
4.4.2.2 Patch test of cohesive element in conjunction with contact

In composite materials, such as concrete [311] and fiber-reinforced plastics [312], interfacial stresses between the bulk materials develop when they separate. To investigate the behavior of the interfacial transition zone (ITZ), cohesive elements are widely used in literature [313, 314, 315]. The geometry of the microstructure and the interface conditions have a significant influence on the macroscopic behavior of composite materials [316]. Therefore, surface meshes located at the material interface should provide a highly accurate representation of the actual geometry being suitable for simulating the interactions. Matching meshes, which are easily obtained from polyhedral elements, eliminate the gaps and penetrations at the interface and two matching surfaces can directly form a cohesive element. The performance of the combination of existing cohesive elements in ABAQUS and the proposed user elements is investigated in the following.

The scheme of the numerical modeling for the ITZ based on the matching meshes is
illustrated in Fig. 4.21. There are two hexahedral domains representing the bulk materials, and they exhibit a matching discretization at the interface. For the sake of clarity, the matching meshes in Fig. 4.21 have been separated, while essentially they are exactly matching with no gap, i.e., Nodes 5 and 9 have the same coordinates, which is also true for the other matching nodes. These two surfaces can straightforwardly form a zero thickness cohesive element. An 8-node three-dimensional cohesive element COH3D8 can be defined with node ordering ‘5, 6, 7, 8, 9, 10, 11, 12’. In this way, no constraints are required to connect the cohesive element to the other components. To prevent the interpenetration, a contact pair is established based on the two surfaces of the two blocks. The cohesive element generates traction applied to the surrounding bulk materials when they separate. The contact pair will be activated when interpenetration between bulk materials might occur. For triangular matching interfaces, a 6-node three-dimensional cohesive element COH3D6 must be used.

For a better understanding, the basic description of the constitutive response of cohesive elements using a traction-separation law is provided. Readers interested in a comprehensive discussion should refer to the ABAQUS manual [18]. Cohesive elements typically employ a traction-separation response in the normal direction based on an exponential damage evolution law which is depicted in Fig. 4.22. The traction-separation response in tension contains a damage evolution described by a scalar damage value $D$,
Figure 4.22. Typical traction-separation response with exponential damage evolution

and can be expressed as

\[
\begin{align*}
  t_n &= \begin{cases} 
    E_n \delta_n & \delta_n^{\text{max}} \leq \delta_n^o, \\
    (1 - D) E_n \delta_n & \delta_n^o < \delta_n^{\text{max}} \leq \delta_n^f, \\
    0 & \delta_n^{\text{max}} > \delta_n^f,
  \end{cases}
\end{align*}
\]

(4.17)

where \( t_n \) and \( \delta_n \) represent the traction and separation, and \( E_n \) is the initial elastic modulus of the cohesive element in the normal direction. The parameters \( \delta_n^o, \delta_n^f, \delta_n^{\text{max}} \) represent the separation at different states, they indicate damage initiation, complete failure, and the maximum value attained during loading, respectively. The damage variable, \( D \), represents the overall damage of the cohesive element. It has an initial value of 0 before damage initiation and then monotonically evolves from 0 to 1 until total damage occurs. For an exponential softening law, when \( \delta_n^o < \delta_n^{\text{max}} < \delta_n^f \), the damage variable \( D \) is expressed as

\[
D = 1 - \left( \frac{\delta_n^o}{\delta_n^{\text{max}}} \right) \left\{ 1 - \exp \left[ -\alpha \left( \frac{\delta_n^{\text{max}}}{\delta_n^f - \delta_n^o} \right) \right] \int \frac{1 - \exp(-\alpha)}{1 - \exp(-\alpha)} \right\},
\]

(4.18)

where \( \alpha \) is the exponential coefficient. The cohesive element will not be damaged if it is in compression, thus the response in Fig. 4.22 is straight when \( \delta_n < 0 \).

To assess the compatibility of SBFEM user elements and cohesive elements, we use the same mesh that was already used for the contact investigations, i.e., a simple hexahedral domains with a flat interface, and insert cohesive elements at the interface. As shown
in Fig. 4.23, zero thickness cohesive elements are generated at the matching interface. In addition, the contact pairs are established on the matching surfaces for preventing the interpenetration of the two parts. The bottom of the lower block is constrained in the vertical direction, and three DOFs are fixed in the horizontal directions to prevent rigid body motions.

The material properties used in this example are listed in Table 4.5. The material properties of the cohesive elements are chosen for the purpose of the verification, they do not represent any specific mechanical behavior. The tangential properties of the cohesive elements are not listed because the patch test only generates normal stresses. The SBFEM elements are relatively rigid, such that the displacement load $d$ will be mainly transferred to $\delta_n$ when it is positive. When the cohesive elements have no damage ($\delta_{n}^{\text{max}} \leq \delta_{n}^{o}$), the ratio between $\delta_n$ and $d$ is 0.99997 for $d > 0$ based on elasticity theory, and it will increase if the damage initiation occurs in the cohesive elements.

A static analysis is performed to observe the behavior of the cohesive elements. Three harmonically varying cyclic displacement-controlled loads $d$ are applied on the top surface of the upper block, as plotted in Fig. 4.24a. The expression of $d$ (Unit: mm) in terms

Table 4.5. Material properties of the patch test inserting cohesive element

<table>
<thead>
<tr>
<th></th>
<th>Young’s modulus $E$ [MPa]</th>
<th>Poisson’s ratio $\nu$</th>
<th>$\delta_n^{o}$ [m]</th>
<th>$\delta_n^{f}$ [m]</th>
<th>$\alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cohesive</td>
<td>1</td>
<td>-</td>
<td>0.001</td>
<td>0.005</td>
<td>2</td>
</tr>
<tr>
<td>Bulk</td>
<td>200000</td>
<td>0.3</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
of time $t(s)$ is
\[
d = \begin{cases} 
0.8\sin 2\pi t & 0 \leq t \leq 1 \\
2\sin 2\pi(t-1) & 1 < t \leq 2 \\
6\sin 2\pi(t-2) & 2 < t \leq 3 
\end{cases}
\] (4.19)

The magnitudes of $d$ are chosen to generate different $\delta_n^{\text{max}}$ representing different damage states in the three cycles. In the first cycle ($0 \leq t \leq 1$ s), the harmonic loading generates a $\delta_n^{\text{max}} = 0.08$ mm $< \delta_n^o$, which indicates there is no damage in the cohesive elements. The second cycle ($1$ s $< t \leq 2$ s) has a $\delta_n^{\text{max}} = 2$ mm which satisfies $\delta_n^o < \delta_n^{\text{max}} < \delta_n^f$, and the cohesive elements are partially damaged in this stage. The third cycle ($2$ s $< t \leq 3$ s) will cause complete failure of the cohesive elements, since it results in $\delta_n^{\text{max}} = 6$ mm $> \delta_n^f$.

The numerical result for the history of $\delta_n$ in the three cycles is plotted in Fig. 4.24a. During the loading history, the normal separation $\delta_n$ has the following results:
\[
\delta_n \begin{cases} 
\approx d & d \geq 0 \\
= 0 & d < 0 
\end{cases}
\] (4.20)

When $d \geq 0$, $\delta_n$ is quite similar to $d$ with negligible distinction as explained before. If $d < 0$, $\delta_n$ equals to 0 because the surfaces of cohesive elements share the same DOFs with the contact surfaces of the two blocks, while the interpenetration between them is prevented by the contact pairs.

The evolution of the normal traction $t_n$ versus $\delta_n$ is depicted in Fig. 4.24b. Some feature points are plotted in Fig. 4.24 to clarify the behavior of cohesive elements during different loading cycles. During unloading, the normal traction $t_n$ always varies linearly and goes back to the origin. The response of the normal traction $t_n$ during loading in the three cycles is described as follows:

1. In Cycle 1, $t_n$ varies linearly according to $\delta_n$ following $t_n = E_n \delta_n$. It reaches its peak value 0.8 kPa at Point A and then decreases to 0, after which it remains 0.
Figure 4.24. Cohesive element behavior under cyclic loading (with contact)
2. In Cycle 2, the initial relationship between $t_n$ and $\delta_n$ is $t_n = E_n \delta_n$ until $\delta_n$ increases to $\delta_n^0$ (Point B), where the damage initiation occurs and $t_n$ reaches its peak value 1kPa. After Point B, $t_n$ decreases with the increase of $\delta_n$ to 0.54kPa (Point C), following the analytical damage evolution law. At Point C, $\delta_n$ reaches its current $\delta_n^{\text{max}}$ which is between $\delta_n^0$ and $\delta_n^f$, thus the cohesive element is partially damaged with damage variable $D = 0.27$. The numerical result of $D$ has a good agreement with the analytical calculation based on Eq. (4.18).

3. In Cycle 3, before $\delta_n$ reaches 2mm (Point D), $t_n$ varies linearly following $t_n = (1 - D) E_n \delta_n$. After Point D with the increase of $\delta_n$, $t_n$ is reduced to 0 at Point E, where $\delta_n^{\text{max}} = \delta_n^f$ and the complete failure of the cohesive elements occurs ($D = 1$). Once the complete failure occurs, $t_n$ always equals 0, although $\delta_n$ has increased to 6mm at Point F.

To conclude, the numerical solutions coincide exactly with the theoretical solutions. Inserting cohesive elements in matching meshes is convenient and performs excellently. By defining contact pairs at the interface, the penetration of bulk materials can be prevented as $\delta_n$ has never been negative during the entire loading history. The simulation of the interaction that involves both traction and contact is easily achieved by the proposed modeling approach on the matching meshes.

4.4.2.3 A cube with a spherical inclusion

The inclusion of particles in composite materials has been proved to increase the stiffness greatly [317], and the behavior of the ITZs has a significant effect on the performance of those composite materials [316]. The interfaces in particle reinforced materials are always complex and most often curved shapes. The polytree based method proposed by Zhang et al. [231] is capable of generating complex matching interface meshes, which offers a convenient way of inserting cohesive elements at the interfaces to study the behavior of ITZs with complex shapes.
A cube with a spherical inclusion, as shown in Fig. 4.25, is considered as an example for analyzing ITZs with curved faces. To this end, we inserted a cohesive layer and a contact pair between the cube and the embedded sphere to model the separation and prevent interpenetration, respectively. The length of the cube is $L = 4\, \text{m}$ and the radius of the sphere is $R = 1\, \text{m}$. Table 4.6 lists the material properties of each part in this example.

The cohesive element has the same initial elastic Young’s modulus in the normal and tangential directions. The damage evolution law of the cohesive element is of linear type, which actually has the same expression as the exponential type with $\alpha = 0$. The STS approach is used for the contact modeling and a Coulomb friction coefficient of $\mu = 0.2$ is chosen for the sliding. At the bottom of the cube, the vertical displacement and the rigid movement in horizontal plane are constrained. A traction of $T = 5\, \text{kPa}$ is applied at the top surface of the cube. The interaction state at the interface will be examined along a path in the $y-z$ plane ($x = 0$), which is shown in Fig. 4.25.

### Table 4.6. Material properties of a cube with a spherical inclusion

<table>
<thead>
<tr>
<th></th>
<th>Young’s modulus $E$ [MPa]</th>
<th>Poisson’s ratio $\nu$</th>
<th>$\delta^c_\nu$ / $\delta^0_\nu$ / $\delta^0_\gamma$ [m]</th>
<th>$\delta^f_\nu$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cube</td>
<td>5</td>
<td>0.25</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Sphere</td>
<td>1</td>
<td>0.25</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cohesive</td>
<td>2</td>
<td>-</td>
<td>0.00005</td>
<td>0.01</td>
</tr>
</tbody>
</table>

138
Two analyses are performed. In one analysis, the model is created with built-in elements of ABAQUS only (referred to as ABAQUS model below). In the other analysis, the model includes SBFEM user elements (referred to as SBFEM model). The ABAQUS model is shown in Fig. 4.26a. It features a coarser mesh for the sphere, which makes the interface meshes non-conforming. In the SBFEM model, matching meshes are generated by re-meshing the interfacial surfaces, as shown in Fig. 4.26b. The re-meshing operation yields polyhedral elements, which can be easily handled employing the SBFEM-based user elements. The interface meshes before and after the re-meshing of an octant (highlighted in Fig. 4.26a) are depicted in Fig. 4.27.

In the ABAQUS model, there are 3,168 C3D8 elements and 4,471 nodes. The cohesive layer is represented by 384 COH3D8 elements, which are conforming to the inner
surface of the cube. The cohesive layer in this model has a thickness of 1 µm and it is connected to the two solid blocks by tie constraints. In the SBFEM model, only the surface meshes are adjusted in the re-meshing, thus the number of solid elements does not change. The interfaces are discretized into 864 matching quadrilaterals, which form 864 COH3D8 elements and 1728 overlaid elements. Besides, 64 elements are overlaid to create the top surface where the traction is imposed. The total number of nodes of the matching mesh is 4,827. As discussed before, the cohesive elements formed by matching meshes have zero thickness and no constraint is required to connect them with adjacent elements. A reference solution is also obtained using ABAQUS built-in elements and a matching mesh. The reference model consists of 201,800 C3D8 elements and 6,144 COH3D8 elements.

The values for the normal traction $t_n$ of the cohesive elements obtained from the three models are compared in Fig. 4.28. Note that the parameter S33 in ABAQUS denotes the normal traction $t_n$ of three-dimensional cohesive element. The contour of the SBFEM model is smoother compared to the ABAQUS model, and it is closer to the reference solution.

The interaction occurring at the interface along the designated path is plotted in Fig. 4.29. A positive value indicates the interaction is traction, whereas a negative value denotes the interaction is contact. The horizontal axis is the angle $\theta$ denoted in Fig. 4.25. As we can see in the figure, the result of the SBFEM model is more accurate than that of the ABAQUS model for both traction and contact zones. At the region $\pi/3 < \theta < 2\pi/3$, the interaction value is 0, which indicates the cohesive elements around this region have been damaged completely.

The computational times for the ABAQUS and SBFEM models are compared in Table 4.7. Because the increment size setting affects the computational time, the default automatic setting of ABAQUS [18] is applied for both models. For the SBFEM model, a pre-calculation step is created before the actual analysis step to obtain stiffness and mass matrices, as discussed in Section 4.2.1. The time used for the element analysis of UEL comprises of both the time consumed in the pre-calculation and the time in the actual step.
Figure 4.28. Comparison of the traction contours of cohesive elements (Unit: Pa)
Figure 4.29. Interaction stress comparison in Y-Z plane

Table 4.7. Comparison of computational times (s) for non-linear analysis

<table>
<thead>
<tr>
<th>Model</th>
<th>No. of elements</th>
<th>UEL time</th>
<th>ABAQUS solution time</th>
<th>Total CPU time</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABAQUS</td>
<td>3552</td>
<td>-</td>
<td>-</td>
<td>373.70</td>
</tr>
<tr>
<td>SBFEM</td>
<td>5824</td>
<td>19.90</td>
<td>89.4</td>
<td>109.30</td>
</tr>
</tbody>
</table>

Apart from the convenience of inserting cohesive elements, the matching meshes have obvious higher accuracy and efficiency compared to the non-matching ones, especially for curved surfaces. The matching meshes are efficient for the numerical simulation of curved ITZs, which commonly exist in composite materials.
4.4.3 Examples for automatic mesh generation

In previous studies, the octree algorithm has been proven to be a robust and fully automatic mesh generation approach [318, 319]. This technique generates octree cells by recursively subdividing one cell into eight smaller ones of equal size until only one material exists in one cell or a certain threshold is reached. Despite its robustness, the application of octree algorithms has been hindered in conventional FEM due to the issue of hanging nodes.

Polytope user elements, which allow highly flexible shapes, are complementary to the meshes generated by the octree algorithm. Their combination provides a promising technique for integrating geometric models and numerical analysis in a fully automatic manner. In recent years, octree algorithms have been successfully employed in the SBFEM to generate polyhedral meshes based on digital images [295, 320] and STL models [321]. They are extremely simple and efficient in meshing complex geometries such as concrete and sculptures. Especially the STL-based automatic mesh generation approach is capable of handling curved boundaries by trimming the octree cells [321].

As commercial software, ABAQUS has powerful non-linear solver and it allows parallel equation solution, which benefits the solutions of interfacial problems, especially for geometrically complex problems. Besides, ABAQUS provides comprehensive contact approaches and offers enriched element library that we can combine SBFEM user element with. Especially the contact cohesive behavior and cohesive element, those robust algorithms can be utilized for simulating the traction-separation response occurring in interfacial problems.

In this section, two numerical examples are presented to show the possibilities of using octree cells in ABAQUS in conjunction with our user elements. The first example performs a quasi-static analysis of a porous asphalt concrete specimen meshed by the image-based octree algorithm, in which the interface stripping is simulated by cohesive elements. In the second example, contact analysis is performed on a knee joint meshed by the STL-based octree algorithm with trimming.
4.4.3.1 Analysis of a porous asphalt concrete specimen

Interface stripping is a common distress appeared in asphalt mixture due to the adhesive failure, which is commonly simulated through cohesive elements in ABAQUS [322]. The internal states of the asphalt mixtures can be obtained from high-resolution X-CT scans and processed by digital image processing techniques [323, 322]. The microstructure can be reconstructed by octree meshes based on the digital images in a fully automatic manner [295, 320]. A porous asphalt concrete specimen is illustrated in Fig. 4.30a, which is represented by digital images consisting of $200 \times 200 \times 200$ voxels. The image is a part of the X-CT scans of an asphalt concrete specimen from Ref. [323]. In the figure, the black, gray, and white phases represent the aggregates, asphalt mastic, and air-voids, respectively.

The images can be decomposed automatically using the octree structure, as explained in Ref. [295]. The discretization of the asphalt concrete specimen is shown in Fig. 4.30b. The elements representing air-voids are removed. At the interface, cohesive elements are inserted for the simulation of the stripping damage, and STS contact with a friction coefficient 0.5 [322] is assigned to prevent interpenetration. The minimum and maximum element sizes are $2^3$ and $16^3$ voxels, respectively, and the size of one pixel is 0.1 mm. There are 539,170 polyhedral user elements, 208,880 cohesive elements, and 909,402 nodes in the model. The number of nodes in one polyhedral element varies from 8 to 26. The main feature of such octree meshes is the meshes at the interfaces between material phases are fine. Besides, the interfacial meshes are conforming naturally. The meshes having such features can be constructed by the SBFEM user elements, and they are exactly the meshes required for the simulation of the ITZs as discussed in the previous section.

The mechanical properties used in this examples are listed in Table 4.8, in which the viscoelasticity of the asphalt mastic is not taken into consideration. The interface has the same damage behaviors in the normal and two tangential directions, and allows a maximum traction 4 MPa in each direction. Because the dynamic modulus is often
adapted for the asphalt mastic [323], a quasi-static analysis is performed on the model. The bottom of the concrete specimen is fixed, and the vertical planes are constrained for the simulation of confinement. A displacement controlled load up to $-0.5 \text{mm}$ is applied on the top surface with a loading rate $0.02 \text{mm/s}$.

The stress and damage states of the cohesive elements at the end are depicted in Fig. 4.31. From Fig. 4.31a we can see, the Mises stress has reached up to $9.459 \text{MPa}$ in some area, which may cause damage initiation. The damage contour in Fig. 4.31b (the parameter SDEG in ABAQUS represents the scalar damage value $D$) shows clearly that the damage has occurs at the interface. In some areas, the damage value reaches to 1, where the interface has complete failure. Those areas are concentrated at the edges of interface or smooth interfaces. This phenomenon is reasonable, because the damage in
the normal and tangential directions tends to initiate at the edges of interface and smooth interfaces, respectively.

The total CPU time of this example is 233.51 hours, in which 11.48 hours are spent on the element analysis of UEL. The UEL time comprises of 1.96 hours on the precalculation and 9.52 hours on the step of nonlinear analysis. It accounts for only 4.9% of the total CPU time.
4.4.3.2 Analysis of a knee joint

A knee joint given in STL format [325] is considered for performing a contact analysis. The knee joint is composed of femur and tibia, a meniscus and two articular cartilages as illustrated in Fig. 4.32. The femur and the tibia are covered with articular cartilages, thus the interactions between bones and articular cartilages can be treated as perfect bond (tie constraint in ABAQUS). The meniscus is inserted into the articular cartilage covering the tibia, thus the interaction between them can also be regarded as perfect bond. The interaction between the meniscus and the articular cartilage covering the femur is treated as contact.

The mesh generated by the octree algorithm with trimming [321] is shown in Fig. 4.33. The base sizes are 0.75 mm and 1 mm for the contact parts and other parts, respectively. There are 253,079 elements and 361,296 nodes, and the number of nodes in one element varies from 4 to 27. A slice in the vertical plane passing the geometrical center is also shown in Fig. 4.33. The contact interface meshes are non-matching for this example,
because the contact domains are geometrically non-conforming and large sliding might occurs at the interface during the loading process. The meshes generated through the STL-based octree algorithm are suitable for contact analysis. In the standard FEM, local mesh refinement may be required to generate finer meshes on the contact surfaces. However, in an automatic manner, octree cells with fast mesh size transition can be generated through the octree mesh generation technique. Compared to the interior elements, the elements near the boundaries are smaller, which will generate more accurate surfaces for tie constraints and contact pairs. Besides, compared to the image-based technique, the STL-based octree algorithm allows trimming on the octree cells, which leads to more accurate meshes representing complex boundaries especially for curved shapes.

The tissue material properties in the computational modeling of human knee joints have been studied in Ref. [326], however, there are considerable variations of the material properties within the wide body of literature. The mechanical properties used in this example are mainly taken from Ref. [15] and are listed in Table 4.9. The contact is STS contact with a friction coefficient of $\mu = 0.05$ [327].
Table 4.9. Material properties of the knee joint [15]

<table>
<thead>
<tr>
<th></th>
<th>Femur</th>
<th>Tibia</th>
<th>Articular cartilage</th>
<th>Meniscus</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus $E$ [MPa]</td>
<td>20,000</td>
<td>20,000</td>
<td>30</td>
<td>2</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.2</td>
<td>0.2</td>
<td>0.25</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Figure 4.34. Displacement contours of the knee joint (Unit: mm)

The bottom surface of the tibia head is fixed, and two translational DOFs in the horizontal plane of the top surface of the femur head are also constrained. On the top surface of the femur head, a vertical load $F$ of up to 1000N [15] is applied to induce contact in the knee joint. The displacement contours ($F = 1000$N) from different views are shown in Fig. 4.34. It is obvious that the deformation occurs mainly in the meniscus. The femur, tibia and articular cartilages are similar to rigid bodies. This phenomenon meets the expectation because the Young’s modulus of the meniscus is significantly smaller than those of the bones and articular cartilages. The meniscus is squeezed into the joint space and there is lateral extension of the meniscus. The large deformation of the meniscus increases the contact area which is helpful to reduce the contact pressure.
The contact pressure distributions \( (F = 1000\, \text{N}) \) are depicted in Fig. 4.35. To illustrate the distribution clearly, contact pressure values smaller than \( 10^{-4}\, \text{MPa} \) are not included in the contour. It is obvious that the contact pressure on the articular cartilage has a similar distribution and magnitude compared to that on the meniscus. Besides, the contact pressure on the articular cartilage has a similar distribution from Ref. [15].

The development of the contact on the articular cartilage is recorded, as depicted in Fig. 4.36. Note that only the area where the contact pressure is greater than \( 10^{-4}\, \text{MPa} \) has been taken into account. Generally speaking, with increasing the vertical load \( F \), the contact area is increasing while its increasing rate is reducing. The average contact pressure increases during the loading history. Before the vertical load increases up to 650N, the increasing rate of the average contact pressure is basically reducing. However, when \( F > 650\, \text{N} \) the average contact pressure increases almost linearly because the contact area increases only slightly.

The total CPU time of this example is 16.68 hours, in which 0.90 hours are spent on the element analysis of UEL. The UEL time comprises of 0.52 hours on the pre-calculation and 0.38 hours on the step of nonlinear analysis. It accounts for only 5.4\% of the total CPU time.
Figure 4.36. Development of the contact on the articular cartilage
4.5 Conclusion

In this section, polyhedral element and two-dimensional unbounded element based on the SBFEM are implemented as user elements within the commercial finite element software ABAQUS. The implementation involves the user subroutines UEXTERNALDB and UEL to store the topology and perform the calculations of the polyhedral element, respectively. The required data structures for storing the polyhedral mesh and the elemental topology have been presented and described in detail. A comprehensive explanation of the framework of the UEL has been provided in conjunction with the theoretical derivations needed for the implementation. Furthermore, a modified approach of overlaying standard elements on the SBFEM user elements is proposed to create element-based surfaces, which allows establishing interactions for interfacial problems. The definitions of output arrays for two-dimensional unbounded user element are presented explicitly.

The availability of polyhedral user elements enhances the performances of ABAQUS for interfacial problems, and it significantly reduces the meshing burden encountered in the standard FEM. To verify the implementation, three benchmark tests (static, modal, and transient) are performed. It is observed that very accurate results are obtained when compared to theoretical and numerical reference solutions. The advantages of polyhedral elements, i.e., the flexibility of dealing with non-matching meshes and its compatibility with automatic mesh generation techniques, are illustrated by five numerical examples. By introducing matching meshes for complex geometries consisting of several parts featuring different element sizes, the performance of existing contact modeling approaches in ABAQUS can be significantly enhanced. Matching meshes provide interface representations without initial gaps or penetrations, and thus, zero thickness cohesive elements for the simulation of ITZs in the composite materials can be easily inserted. The SBFEM user elements are highly complementary to octree-based mesh generation techniques, which are efficient and robust to mesh complex geometries and promising for integrating geometric models and numerical analysis in a fully automatic manner. In a nutshell, the SBFEM user element will enable ABAQUS users to benefit from the advantages of this
method. The commercial FEM software ABAQUS offers a user-friendly interface and outstanding nonlinear solvers, which will facilitate the use of the SBFEM to scholars.

The two-dimensional unbounded user elements derived by the SBFEM rigorously capture the dynamic properties of unbounded media. They only require small truncation areas to satisfy the radiation condition.
Chapter 5

Interfacial problems in post-tensioned gravity dams

As discussed in Section 1.2.2, the behavior of interfaces existing in post-tensioned gravity damS, such as the contact and cohesive fracture at the weak interfaces (dam-foundation interface and construction joints) and the bond-slip interaction at the anchor-structure grouting interfaces, should be considered in a rigorous numerical simulation. Section 2.2 reviews the modeling techniques for the weak interfaces and anchor-structure grouting interfaces. Inserting interface elements [91] has been widely used to model the cohesive behavior of the weak interfaces and the bond-slip phenomena occurring at the anchor-structure grouting interfaces [194, 87].

In this chapter, the numerical simulation of interfacial problems utilizing four-node interface elements is demonstrated. Since the framework has been implemented in ABAQUS in this thesis, the cohesive elements (COH2D4) from ABAQUS’ element library are used as the interface elements. The fundamental derivations of interface elements and the constitutive laws of cohesive elements are introduced in Section 5.1. In Section 5.2, the modeling schemes of the weak interfaces and PTAs considering bond-slip interactions are explained. To facilitate the proposed modeling schemes, interfacial conforming meshes are utilized. The advantages of conforming meshes and the related mesh-conforming
techniques to convert non-conforming meshes to conforming ones are presented in Section 5.3. Two simple numerical examples to examine the proposed modeling schemes on conforming meshes are presented in Section 5.4. Some of the material in this chapter has been included in a manuscript titled as ‘Seismic analysis of post-tensioned concrete gravity dams using scaled boundary finite elements implemented as ABAQUS UEL’ submitted to Soil Dynamics and Earthquake Engineering.

5.1 Interface element

5.1.1 Stiffness matrix

A four-node interface element (isoparametric) with a length of $2a$ is shown in Fig. 5.1. The thickness of this element should be thin or zero. A dimensionless coordinate $\xi = \frac{x}{a}$ in the longitudinal direction is introduced, where $x$ is a local coordinate. The position of each node can be described through $\xi$ as: $\xi_1 = \xi_4 = -1$, $\xi_2 = \xi_3 = 1$.

The nodal displacement vector $\mathbf{u}$ of an interface element is

$$\mathbf{u} = \begin{bmatrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & u_4 & v_4 \end{bmatrix}^T,$$

where $u_i$ and $v_i$ denote the displacements of node $i$ in the tangential and normal directions of the interface element, respectively, as indicated in Fig. 5.1. The displacement field $\mathbf{u}(\xi)$ of an interface element consists the displacements of two surfaces, i.e., the displacements...
Figure 5.2. Internal fields of an interface element

\(u_I(\xi), v_I(\xi)\) of Surface 1-2 and the displacements \(u_{II}(\xi), v_{II}(\xi)\) of Surface 3-4, as shown in Fig. 5.2. Through linear interpolation, the displacement field can be obtained as:

\[
\mathbf{u}(\xi) = \begin{bmatrix} u_I(\xi) \\ v_I(\xi) \\ u_{II}(\xi) \\ v_{II}(\xi) \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & 0 & 0 & 0 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & N_2 & 0 & N_1 & 0 \\ 0 & 0 & 0 & 0 & 0 & N_2 & 0 & N_1 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \mathbf{N} \mathbf{u}, \quad (5.2)
\]

where \(N_1\) and \(N_2\) are shape functions and expressed as:

\[
N_1(\xi) = \frac{1}{2}(1 - \xi), \quad (5.3a)
\]
\[
N_2(\xi) = \frac{1}{2}(1 + \xi). \quad (5.3b)
\]

The equivalent strain field \(\mathbf{\varepsilon}(\xi)\) of an interface element describes the slippage \(\delta_t\) and separation \(\delta_n\) between Surface 1-2 and Surface 3-4, i.e., relative displacements between
the two surfaces in the tangential and normal directions, respectively:

$$\epsilon(\xi) = \begin{bmatrix} \delta_t(\xi) \\ \delta_n(\xi) \end{bmatrix} = \begin{bmatrix} -1 & 0 & 1 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_I(\xi) \\ v_I(\xi) \\ u_{II}(\xi) \\ v_{II}(\xi) \end{bmatrix} = \mathbf{L}u(\xi). \quad (5.4)$$

Substituting Eq. (5.2) into Eq. (5.4), the relation between the equivalent strain field $\epsilon(\xi)$ and the nodal displacement vector $u$ is obtained as

$$\epsilon(\xi) = \begin{bmatrix} -N_1 & 0 & -N_2 & 0 & N_2 & 0 & N_1 & 0 \\ 0 & -N_1 & 0 & -N_2 & 0 & N_2 & 0 & N_1 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \mathbf{B}u. \quad (5.5)$$

The stress field $\sigma(\xi)$ of an interface element consists of shear stress $\tau(\xi)$ normal stress $\sigma(\xi)$ applying at the surfaces, as shown in Fig. 5.2:

$$\sigma(\xi) = \begin{bmatrix} \tau(\xi) \\ \sigma(\xi) \end{bmatrix}.$$ 

The stress-equivalent strain relationship is non-linear for an interface element, and it can be expressed in an incremental form as

$$\Delta \sigma = \begin{bmatrix} \Delta \tau \\ \Delta \sigma \end{bmatrix} = \begin{bmatrix} E_t(\delta_t) & 0 \\ 0 & E_n(\delta_n) \end{bmatrix} \begin{bmatrix} \Delta \delta_t \\ \Delta \delta_n \end{bmatrix} = \mathbf{D}(\epsilon)\Delta \epsilon, \quad (5.6)$$
in which $E_t$ and $E_n$ are the equivalent stiffness of the interface element in the tangential and normal directions, and they are developing with the slippage and separation, respectively. The off-diagonal terms in $D$ are set to be zero if uncoupled behavior between the normal and shear components is assumed. For an interface element used to model the bond-slip interaction, the development of $E_t$ should reflect the local bond stress-slippage relationship, while the normal stiffness $E_n$ should be large enough to prevent the penetration between the anchor and structure. For an interface element used for the simulation of weak interfaces, the tangential stiffness $E_t$ represents the cohesion occurring with sliding, and the normal stiffness $E_n$ is used to simulation the cohesive fracture when the interfaces are separated.

The element stiffness matrix $K$ in the local coordinate system for an interface element can be written in a standard form (similar to the solid element in the FEM):

$$K = \int_S B^T DBdS,$$  \hspace{1cm} (5.7)

where $S$ is the area of the interface surface. The area $S$ is the product of the length $2a$ and the out of plane thickness $T$ of the interface element. Thus the stiffness matrix can be rewritten as

$$K = aT \int_{-1}^{1} B^T DBd\xi.$$  \hspace{1cm} (5.8)

For an interface element used for simulating the bond-slip interaction between the anchor and structure, the thickness $T$ is the perimeter of the drilled hole $d_h$, i.e., $T = \pi d_h$.

The stiffness matrix $K$ should be transformed into the stiffness matrix $\hat{K}$ in the global coordinate system. Figure 5.3 shows the coordinate transformation of an interface element, where $(\bar{x}, \bar{y})$ is the global coordinate system. The nodal displacement vector $\bar{u}$ in global coordinate system is
\[ \mathbf{\tilde{u}} = \begin{bmatrix} \tilde{u}_1 & \tilde{v}_1 & \tilde{u}_2 & \tilde{v}_2 & \tilde{u}_3 & \tilde{v}_3 & \tilde{u}_4 & \tilde{v}_4 \end{bmatrix}^T. \tag{5.9} \]

The displacements of a node \( i \) in the local coordinate system can be transformed from the displacements in the global coordinate system by

\[
\begin{align*}
    u_i &= \bar{u}_i \cos \alpha + \bar{v}_i \sin \alpha, \tag{5.10a} \\
    v_i &= \bar{v}_i \cos \alpha - \bar{u}_i \sin \alpha, \tag{5.10b}
\end{align*}
\]

in which \( \alpha \) is the angle between the local and global coordinate systems. It can be written in a matrix form as

\[
\begin{bmatrix} u_i \\ v_i \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 & 0 & 0 & 0 & 0 \\ -\sin \alpha & \cos \alpha & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \cos \alpha & \sin \alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & -\sin \alpha & \cos \alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \cos \alpha & \sin \alpha & 0 & 0 \\ 0 & 0 & 0 & 0 & -\sin \alpha & \cos \alpha & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \cos \alpha & \sin \alpha \\ 0 & 0 & 0 & 0 & 0 & 0 & -\sin \alpha & \cos \alpha \end{bmatrix} \begin{bmatrix} \tilde{u}_i \\ \tilde{v}_i \end{bmatrix} = \mathbf{T} \mathbf{\tilde{u}}, \tag{5.11}
\]

where \( \mathbf{T} \) is the transformation matrix. Similar to the beam/truss element in the FEM, the stiffness matrix \( \mathbf{\tilde{K}} \) in the global coordinate system of an interface element can be transformed from the stiffness matrix \( \mathbf{K} \) in the local coordinate system, by

\[ \mathbf{\tilde{K}} = \mathbf{T}^T \mathbf{K} \mathbf{T}. \tag{5.12} \]
5.1.2 Constitutive law of cohesive element in ABAQUS

The two-dimensional cohesive element from ABAQUS’ element library (COH2D4) is used as the interface element in this thesis. The constitutive response of cohesive elements using a traction-separation law is summarized in this section, i.e., the developments of $E_t$ and $E_n$ in Eq. (5.6). The traction-separation law is suitable for cohesive elements with negligibly small thickness. In this approach, the constitutive response of the cohesive layer is defined by damage initiation and evolution in terms of traction versus separation. Readers interested in a comprehensive discussion should refer to the ABAQUS manual [18].

In the traction-separation law, there is a scalar damage variable, $D$, representing the overall damage in the material. The damage variable initially has a value of 0. After the initiation of damage, $D$ monotonically evolves from 0 to 1 upon further loading. The stress components of the traction-separation model are affected by the damage according
\[ \tau = (1 - D)E_0^t \delta_t, \quad (5.13a) \]

\[ \sigma = \begin{cases} 
(1 - D)E_0^n \delta_n, & \delta_n \geq 0 \\
E_0^n \delta_n, & \delta_n < 0 
\end{cases}, \quad (5.13b) \]

where \( E_0^t \) and \( E_0^n \) are the initial stiffness (no damage) in the tangential and normal directions, respectively. It can be noted that, there is no damage to compressive situation.

Several damage initiation criteria are available in ABAQUS, such as the maximum nominal stress criterion, the maximum nominal strain criterion, the quadratic nominal stress criterion, and the quadratic nominal strain criterion. In the maximum nominal stress criterion, damage initiates when the maximum nominal stress ratio reaches a value of one:

\[ \max \left\{ \frac{\langle \sigma \rangle}{\sigma^0}, \frac{\tau}{\tau^0} \right\} = 1, \quad (5.14) \]

where \( \sigma^0 \) and \( \tau^0 \) represent the peak values of the normal stress and shear stress, respectively, and the symbol \( \langle \rangle \) represents the Macaulay bracket to signify that a pure compressive stress state does not initiate damage. Likewise, maximum nominal strain criterion specifies the peak values of the nominal strains \( \delta_0^n \) and \( \delta_0^t \) in the normal and tangential directions, respectively. Damage initiates when the maximum nominal strain ratio reaches one. In the quadratic nominal stress criterion, damage initiates when a quadratic interaction function involving the nominal stress ratios reaches a value of one:

\[ \left\{ \frac{\langle \sigma \rangle}{\sigma^0} \right\}^2 + \left\{ \frac{\tau}{\tau^0} \right\}^2 = 1. \quad (5.15) \]

Similarly, in the quadratic nominal strain criterion, damage initiates if the quadratic function of the nominal strain ratios reaches one.

There are three common ways in ABAQUS to define the softening behavior of the cohesive element, i.e., the damage evolution of the traction-separation response: linear,
exponential, and tabular. The schematics of these three laws (normal direction) are shown in Fig. 5.4. The cohesive element will not be damaged if it is in compression, thus the response in Fig. 5.4 is straight when $\delta_n < 0$. In all defining approaches, the maximum nominal stress $\sigma^n$ or maximum nominal strain $\delta_n^o$, and the initial stiffness $E_n^0$ should be specified. In the linear damage evolution, the damage variable $D$ monotonically evolves from 0 to 1 with the increase of the separation $\delta_n$ following:

$$D = \frac{\delta_n^f (\delta_n^{max} - \delta_n^o)}{\delta_n^{max} (\delta_n^f - \delta_n^o)}, \quad (5.16)$$

where $\delta_n^f$ and $\delta_n^{max}$ represent the separation at complete failure, and the maximum separation during the loading, respectively. For an exponential evolution, the damage variable $D$ develops with the separation $\delta_n$ following:

$$D = 1 - \left( \frac{\delta_n^o}{\delta_n^{max}} \right) \left\{ 1 - \exp \left[ -\alpha \left( \frac{\delta_n^{max} - \delta_n^o}{\delta_n^f - \delta_n^o} \right) \right] \right\}, \quad (5.17)$$

where $\alpha$ is the exponential coefficient. For the tabular softening, the evolution of $D$ is directly defined in a tabular form. Essentially speaking, it piece-wisely specifies the damage values related to different separations.

### 5.2 Modeling schemes

This section demonstrates the modeling schemes for the weak interfaces and the post-tensioned anchors embedded in the structure. The schemes are employed in ABAQUS using cohesive elements. For problems involving multiple domains, it is practical to discretize different domains independently in ABAQUS, which may result in non-matching meshes. Thus, the demonstration is based on non-matching meshes for the purpose of generality.
5.2.1 Modeling of weak interfaces

The stability of concrete gravity dams is substantially affected by the strength of the dam-foundation interface and the lift joints [4, 75]. The tensile strength commonly reaches its lowest value at such interfaces [4], thus the tensile behavior at weak interfaces should be considered since the cracks occurring there affect the overturning stability. Besides, sliding most likely occurs at these interfaces, where the peak shear strength depends on both cohesion and friction for bonded contacts [4].

A cohesive-frictional contact scheme is utilized for the simulation of such weak interfaces by inserting a zero-thickness cohesion layer between two contact blocks. Figure 5.5 demonstrates the modeling schematic at the dam-foundation interface, which is also used for the dam lift. General contact with a surface-to-surface (STS) scheme, embedded in ABAQUS, is utilized for the contact modeling. Cohesive elements (COH2D4) are connected with the contact interfaces at each side. With the proposed approach, the contact is activated when there is a penetration, while the cohesive elements provide tractions.
to resist cracking when the interfaces are separated, and damage initiates if the tensile stresses reach a threshold value. As mentioned before, the shear strength of a bonded weak joint depends on both friction and cohesion [4]. Shear stresses develop when relative slippage occurs at the interface, and they are comprised of contributions from the contact pair (friction) and cohesive elements (cohesion).

The selection of a friction model and the parameters governing the behavior of the cohesive-frictional contact are explained in the following. In the tangential direction, the mobilization of friction requires larger slippage, while the degradation of cohesion occurs at small strain [75]. To realistically reflect the mobilization property of the friction, the exponential decay friction model [18] is employed by specifying static \( \mu_s \) and kinetic \( \mu_k \) friction coefficients in conjunction with a decay coefficient \( d_c \). The effective friction coefficient \( \mu_e \) decays exponentially from \( \mu_s \) to \( \mu_k \), if the slippage reaches a static friction limitation \( s_e \):

\[
\mu_e = \mu_k + (\mu_s - \mu_k) \exp(-d_c \dot{\gamma}_\text{eq}),
\]

where \( \dot{\gamma}_\text{eq} \) is the slip rate. The cohesive element has an initial Young’s modulus \( E_n \) and \( E_t \) in normal and tangential directions, respectively. It has a tensile strength of \( \sigma_{\text{max}} \) and a shear strength of \( \tau_{\text{max}} \). The constitutive response of cohesive element is determined by a traction-separation response with linear damage evolution [18]. The parameters governing the constitutive response with linear damage evolution have been in explained in

Figure 5.5. General schematic of cohesive-frictional contact modeling
Section 5.1.2.

As a summary, the parameters used to determinate the cohesive-frictional contact are $\mu_s$, $u_k$, $d_c$, $s_e$, $E_{n0}$, $E_{t0}$, $\sigma^0$, $\tau^0$, $\delta^f$. There are considerable variations of interaction properties within the wide body of literature for concrete-rock contact [328, 4, 174, 329]. The parameters used in this work are mainly related to data presented in the final report of the Electric Power Research Institute (EPRI) [4], which has investigated the interfacial behavior for various types of rock by both laboratory and in-situ tests.

5.2.2 Modeling of PTAs considering bond-slip

A PTA is usually grouted with the structure along a portion of the length, called bond length. At the grouting interface, the bond stress develops with the slippage between the anchor and structure, i.e., bond-slip interaction. Current design approaches assume uniform bond stress along the bond length, while different studies have shown that its distribution is markedly non-uniform [16, 81]. To fill this gap and investigate the effects of the bond length design on the seismic performance of post-tensioned gravity dams, realistic modeling of the bond length must be considered.

A modeling scheme is proposed to simulate the PTA system considering the bond-slip interaction occurring along the bond length, by inserting cohesive elements connected to the anchor and structure. Figure 5.6 shows a schematic of the PTA modeling approach, in which there are two PTAs included in a square structure. The length of a PTA consists of a bond length and a free length. The mesh generation of the structural domain is characterized by edge partitioning of the bond lengths and free lengths. By assigning mesh seeds along the partitioning edges, sufficient structural nodes along the anchor layout can be generated for an accurate interpolation of the displacement field of the borehole, especially for the bond lengths. The detailed modeling of the left PTA is exhibited. The anchor is represented by beam elements (B21). Zero-thickness cohesive elements are connected with the anchor and the structure at different sides. The connection has to be realized by employing tie constraints in the ABAQUS model, if the meshes are non-
matching. The anchor head and the structure are locked by tie constraint in the ABAQUS model, since they might belong to different “parts” if the model is created in ABAQUS/CAE. Cohesive elements have great stiffness in the normal direction (100 times that in the tangential direction) to prevent penetration between the anchor and structure [330]. At the bond length, the cohesive elements reflect the bond-slip interaction, and they yield shear stresses if slippage occurs between the anchor and structure. For the free length, the cohesive elements have negligible stiffness in the tangential direction, which allows the anchors to slip freely along the borehole layouts (neglecting friction).

The constitutive law of the cohesive elements associated with the bond length should be able to describe the local bond stress-slippage relationship. Among various models, a model proposed by Bertero, Popov, and Eligehausen (BPE) [6] (see Fig. 5.7a) and a trilinear model [78] (see Fig. 5.7b) have been widely used, especially for rebars embedding in concrete. In this work, a quadrilinear model [112] (see Fig. 5.7c) originally proposed for the ground anchor is utilized to simulate the bond-slip behavior at the rock-grout interface. This model is more straightforward compared to the BPE model, and it allows a yielding plateau comparable to the trilinear model. The bond stress-slippage relationship
and its damage evolution law of the quadrilinear model are given as

\[
\tau = \begin{cases} 
\tau_u \left( \frac{s}{s_1} \right) & s \leq s_1 \\
\tau_u & s_1 < s \leq s_2 \\
\tau_u - \left( \tau_u - \tau_r \right) \frac{s - s_2}{s_3 - s_2} & s_2 < s \leq s_3 \\
\tau_r & s > s_3 
\end{cases}, \\
D = \begin{cases} 
0 & s \leq s_1 \\
1 - \frac{s_1}{s} & s_1 < s \leq s_2 \\
1 - \frac{s_1}{s} + \frac{(\tau_u - \tau_r)(s - s_2)s_1}{\tau_u(s_3 - s_2)s} & s_2 < s \leq s_3 \\
1 - \frac{\tau s_1}{\tau_u s} & s > s_3 
\end{cases},
\]

in which \(\tau_u\) and \(\tau_r\) are the ultimate and residual bond stresses, respectively. The bond strength has considerable variations in the literature [331, 17, 79, 332, 112, 5], and thus the determination of the governing parameters in the quadrilinear model usually requires full-scale pull-out tests in dam engineering practice. Then the local bond-slip relationship can be derived from the pull force-displacement data following the theoretical analysis procedure proposed by Liu et al [80]. In this work, the used parameters are selected based on the recommendation of the Post-Tensioning Institute (PTI) [5] and several reported full-scale pull-out tests [17]. The damage evolution in cohesive elements in Eq. (5.19) is described in a tabular form, which defines the damage value during damage progress in a piece-wise fashion.

### 5.3 Mesh-conforming techniques

The modeling schemes mentioned in Section 5.2 can be established on conforming meshes, leading to several advantages. Firstly, conforming meshes improve the accuracy of contact modeling [86, 88, 89]. Moreover, with conforming meshes, cohesive elements can be formed by matching node pairs at the interfaces [89]. In this way, the cohesive elements are connected with surrounding elements naturally by sharing nodes, thus no interface constraints are required, and additional unknowns/constraints are avoided.

In this work, conforming meshes are used in the proposed framework for both the weak joint and bond length modeling. Suitable mesh-conforming procedures for convert-
Figure 5.7. Bond-slip models
ing existing non-matching interfacial meshes to matching ones are developed [86, 231, 12]. They are easily implemented based on the polygonal S-elements that allow arbitrary numbers of nodes and edges. This section explains the mete-conforming techniques to generate: i) surface conforming meshes for the weak joints and ii) nodal conforming meshes for the PTAs.

5.3.1 Surface conforming meshes

In this thesis, interfacial meshes that have both matching nodes and edges are called surface conforming meshes. They provide interface representations without initial gaps or penetrations for contact problems and improve the performance of ABAQUS’ STS contact scheme [89]. The cohesive-frictional contact modeling in Section 5.2.1 is established on surface conforming meshes in the proposed framework.

A remeshing algorithm operated on polygonal S-elements to convert non-conforming meshes to conforming ones is developed as depicted in Fig. 5.8. The original discretization of the structural domains is the same as that in Section 5.2.1, it can be still generated individually for different domains and may have non-conforming interfaces. The remeshing algorithm consists of three steps, and it creates the cohesive-frictional contact in a fully automatic manner:

1. Remeshing non-conforming meshes to conforming ones (Fig. 5.8a). This step obtains coarse conforming meshes by adding nodes (red) for the nodes originally have no matching nodes on the opposite interface. Note that if two nodes on opposite interfaces are close to each other but not strictly conforming, one can be shifted to be matching with the other one instead of inserting matching nodes [86]. The added nodes subdivide the edges into shorter edges, and they are matching at the interfaces.

2. Refining coarse conforming meshes to finer ones (Fig. 5.8b). To obtain accurate results, cohesive elements may have smaller sizes compared to the surrounding
3. Generating cohesive elements on conforming meshes (Fig. 5.8c). Two adjacent pairs of matching nodes form a cohesive element directly. The generated cohesive elements share nodes with the surrounding elements, thus no additional constraints are required to connect them.
5.3.2 Nodal conforming meshes

Nodal conforming meshes have matching nodes along the characteristic interfaces. Compared to surface conforming meshes, matching edges are unnecessary for nodal conforming meshes. They are suitable for the modeling of composite materials with fiber inclusions [12]. In this work, nodal conforming meshes are utilized for the simulations of PTAs in the proposed framework.

An automatic remeshing algorithm is developed to generate nodal conforming meshes based on the polygonal S-elements, it releases the modeling burdens of PTAs in standard FEM. As discussed in Section 5.2.2, the discretization of the structural domain requires partitioning at the characterized lengths to obtain a sufficient number of nodes along the borehole. If there are multiple inclined anchors interacting with each other, irregular partitioning may result in distorted quadrilateral shapes. In addition, several tie constraints are required for each anchor, for the connections of cohesive elements and anchors, cohesive elements and boreholes, and the anchor head locking in the ABAQUS model. In the framework of the proposed approach, the structural domain discretization is independent of the anchor layouts, and the connections are achieved by sharing nodes based on conforming meshes. This remeshing scheme consists of two major procedures to establish the PTA modeling: i) a remeshing procedure modifying existing meshes to obtain sufficient nodes along anchor layouts, and ii) a duplication procedure creating matching nodes, beam elements and cohesive elements to realize the PTA modeling. A schematic of the remeshing algorithm is presented in Fig. 5.9, which has the same structure and anchors (geometry) as the example in Section 5.2.2.

Figure 5.9a depicts the remeshing procedure modifying an existing structural mesh [12]. The structural mesh is different from the example in Section 5.2.2, its generation does not involve partitioning edges along the bond lengths and free lengths. The remeshing procedure firstly sets three key points for each anchor identifying the characterized lengths, i.e., an anchor head (H) and two ends of the bond length (PE for proximal end and DE for distal end). Thereafter, by shifting nodes, inserting nodes, partitioning lines, and subdividing
elements, the original mesh (dashed dark lines) is modified to generate a new mesh (solid dark lines) that contains sufficient nodes along the anchor layout (colored solid circles). The remeshing consists of six strategies, in which the former strategies take precedence over the latter ones:

1. For a key point on the boundary, usually the anchor head (H1 and H2), the involved boundary edge is partitioned into two edges by inserting a node at the key point;

2. For a key point close to a node of the initial mesh (DE2 and PE2), the original node is shifted to the key point. The threshold for executing Strategy 2 is $d \geq e_2 l_e$, in which $d$ is the distance between the node and key point, $e_2$ is a predefined tolerance, and $l_e$ is the minimum length of the edges connected to the node;

3. For a key point close to an internal edge of the initial mesh (PE1), the edge is partitioned into two edges by the key point. The threshold for Strategy 3 is $d \geq e_3 l_e$, where $d$ is the perpendicular distance of the key point to the edge, $e_3$ is a predefined tolerance, and $l_e$ is the length of the edge;

4. For a key point within an element of the initial mesh (DE1), a new node is inserted at the key point, and the element is subdivided into $n$ new triangular elements by connecting the new nodes to the original nodes ($n$ is the initial number of edges, which is four in this example);
5. Initial nodes close to the anchor layout are shifted to the perpendicular feet on the layout (P1). The threshold for this strategy is \( d \geq e_5 l_e \), where \( d \) is the perpendicular distance of the node to the anchor layout, \( e_5 \) is a predefined tolerance, and \( l_e \) is the minimum length of the edges connected to the node;

6. At the intersection points of the anchor and initial mesh edges, additional nodes are inserted and subdivide the initial mesh edges into new edges (P2).

It is notable that, there are two node shifting (Strategies 2 and 5) and one edge partitioning (Strategy 3) operations that affect the shape of the original mesh. They are developed to avoid severely distorted polygonal elements [12]. The predefined tolerances \( e_2, e_3 \) and \( e_5 \) are set to be 0.15, 0.2, 0.3 in this thesis, respectively.

The duplication procedure takes the left anchor in Fig. 5.9a as an example and is demonstrated in Fig. 5.9b. The steps of realizing the same PTA modeling scheme as in Section 5.2.2 are explained as follows:

1. Recognizing the structural nodes along the borehole layout (colored solid circles: Nodes 1, 2, 3, 4, 5, 6);

2. Duplicating these nodes except for that one on the anchor head (Node 6). New anchor nodes (Nodes 1', 2', 3', 4', 5') are obtained with the same coordinates as the structural nodes;

3. Connecting two adjacent anchor nodes, additional beam elements (B1, B2, B3, B4) are generated. The anchor head node (Node 6) directly connects with the adjacent anchor node (Node 5') to form another beam element (B5);

4. Creating cohesive elements (C1, C2, C3, C4) using the conforming nodes between each two adjacent positions. The cohesive elements are naturally connected with the anchor and the structure;

5. Assigning cohesive elements with different properties for bond and free length parts of the anchor. If all nodes of a cohesive element are within the bond length, the
bond-slip model is assigned to this element. Otherwise, the element has negligible stiffness in the tangential direction. For example, cohesive element C3 (node ordering “3, 4, 4′, 3′”) should have negligible tangential stiffness since Node 4 belongs to the free length.

Generating mesh edges along the boreholes is unnecessary since the boreholes are not Neumann boundaries in two-dimensional problems. It might be argued that the lower edges of cohesive elements are not connected with the polygonal S-elements, which results in a slightly internal displacement difference along these lines, i.e., semi-analytical solutions for polygonal S-elements while linear interpolations for cohesive elements. Actually, the developments of cohesive elements only depend on nodal displacements regardless of the internal field, since its integration points locate at the ends of the element (the crosses in Fig. 5.9b) [18]. As long as cohesive elements are connected with the structure at the ends, the results are not affected by missing the edge connection. The proposed nodal conforming scheme is sufficient for the anchor-structure interaction.

5.3.3 Intersection situations

There are possible intersection scenarios when generating conforming meshes in post-tensioned gravity with multiple anchors and weak joints. The post-tensioned anchors used for controlling concrete cracking cross over the lift joints, and the ground anchoring passes through the dam-foundation interface. It is also reasonable to have planar intersections of anchors in two-dimensional modeling since they are the projections of spaced anchors in the longitudinal direction of dam engineering.

The schematic of inserting a bridging anchor into two contact domains with weak joint is presented in Fig. 5.10a. Two conforming nodes (P1 and P2) at the contact interfaces are inserted due to the intersections with the anchor. The nodal duplication procedure should generate one unique beam node (D1) for the two conforming nodes. These three nodes have the same coordinates, and they are separated for the sake of clarity. Two cohesive elements (COH2D4) for anchor-structure interaction (blue) are sharing nodes with each
polygonal S-element separately at the lower surfaces, while they are both associated with the Node D1 at the upper surface. In addition, the contact interfaces are subdivided by Node P1 and P2 into two segments at both sides, resulting in two cohesive elements (green) for the weak joint.

The modeling of two intersecting anchors is displayed in Fig. 5.10b. The two anchors intersect with the same polygon having eight nodes, in which four are the initial nodes (black), two are inserted nodes (red) to conform with one anchor, and two are for the other anchor. There is no node inserted at the planar intersection point for neither the polygonal S-element nor the beam elements, and thus, the two beam elements are essentially not intersected, and they interact with the polygonal S-element independently.

5.4 Numerical examples

Two numerical examples are presented in this section to verify the proposed modeling schemes of interfacial problems, and their performance on conforming meshes. The first example is a simulation of direct shear test on two contact blocks, which is used to examine the modeling scheme of weak interfaces. A pull-out test is simulated in the second example, in which a single anchor is grouted with the structure, and thus, the bond-slip interaction is considered.
Table 5.1. Cohesive-frictional contact parameters of concrete-sandstone contact [4]

<table>
<thead>
<tr>
<th>( \mu_s )</th>
<th>( \mu_k )</th>
<th>( d_c )</th>
<th>( s_c )</th>
<th>( E^0 _c )</th>
<th>( E^0 _s )</th>
<th>( \sigma^\circ )</th>
<th>( \sigma^\circ )</th>
<th>( \delta_f )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.14451</td>
<td>0.5543</td>
<td>10^8</td>
<td>1</td>
<td>18</td>
<td>18</td>
<td>0.9</td>
<td>1.8</td>
<td>0.5</td>
</tr>
</tbody>
</table>

5.4.1 Direct shear test simulation

A direct shear test simulation is performed on two contact blocks with conforming meshes at the interface. Two relative rigid blocks having the same dimension (0.5 × 1 m) are in contact, where a zero thickness cohesive element is inserted. The upper block is assumed as concrete with a Young’s modulus of \( E_c = 25 \) GPa and a Poisson’s ratio \( \nu_c = 0.2 \). The lower sandstone block has a Young’s modulus of \( E_s = 10 \) GPa and a Poisson’s ratio \( \nu_s = 0.2 \). The parameters for the concrete-sandstone cohesive-frictional contact are mainly referred to data presented in EPRI’s final report, and they are shown in Table 5.1. The definition of each parameter has been explained in Section 5.2.1.

The meshing and boundary conditions of this simulation is shown in Fig. 5.11. A pressure \( P \) is applied on the top surface, and a displacement-controlled loading \( d \) is imposed at the right surface of the upper block. The direct shear tests are simulated with different normal pressures \( P \) inducing different normal stresses at the contact interface. The reaction force \( F \) of the right surface during loading period is recorded and plotted in Fig. 5.12. The reaction-displacement curves have the same characters with that in Refs. [4, 75]. The peak and residual shear strengths at different levels of normal stress are shown in Fig. 5.13, and they are fit with the experimental results in Ref. [4] apparently.

5.4.2 Pull-out simulation

A pull-out test of a single anchor embedded in the foundation is simulated. This example examines the correctness of the modeling scheme and the bond-slip model as presented in Section 5.2.2. The anchor has 24 strands with a length of \( l = 6.2 \) m. Each strand has a diameter of 15.2 mm and a minimum breaking load (MBL) of 250 kN, resulting in a
Figure 5.11. Direct shear test simulation on two contact blocks

Figure 5.12. Reaction-displacement relationship of the direct shear test
Figure 5.13. Shear strength versus normal stress (adopted from Ref. [4])
Displacement-controlled loading

Figure 5.14. Pull-out simulation of ground anchor

Table 5.2. Parameters of quadrilinear model for sandstone-grout interface

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$\tau_u$ [MPa] [5]</th>
<th>$\tau_s$ [MPa] [16]</th>
<th>$s_1$ [mm] [17]</th>
<th>$s_2$ [mm] [17]</th>
<th>$s_3$ [mm] [17]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>4.2</td>
<td>0.42</td>
<td>1.5</td>
<td>2.25</td>
<td>4.5</td>
</tr>
</tbody>
</table>

total MBL of 6000 kN for the anchor. This anchor is fully grouted in a borehole with a diameter of $d_h = 215$ mm. The foundation is assumed as sandstone having a Young’s modulus of $E_s = 10$ GPa and a Poisson’s ratio $\nu_s = 0.2$. The mesh and boundary/loading conditions of this simulation are shown in Fig. 5.14. The anchor is represented by 20 beam elements. The foundation has a dimension of $12.4 \times 12.4$ m, which is discretized as squares with mesh size of 0.31 m. 20 zero thickness cohesive elements are inserted to connect the anchor and foundation. A quadrilinear bond-slip model is used to describe the constitutive response of the cohesive elements. The governing parameters (see Eq. (5.19)) of the quadrilinear bond-slip model for a sandstone-grout interface is shown in Table 5.2. A displacement controlled loading is assigned at the proximal end of the anchor. The anchor is pulled out until only residual stress left.

The pull force-extension at proximal end curve is plotted in Fig. 5.15a. An ultimate pull-out force of 6072 kN is obtained, which is slightly higher than the MBL (6000 kN) of the designed anchor. The bond stress distributions at three loading levels, i.e., 78% MBL, 90% MBL and 100% MBL are compared in Fig. 5.15b, they are obviously non-uniform
distributed along the bond length. Their distributions agree with the shapes in Ref. [5] for the initial, progressive and ultimate loading conditions, in which the peak bond stresses are obtained at the proximal end, shallow depth and deep depth, respectively.

5.5 Conclusion

In this section, the modeling of interfacial problems in post-tensioned gravity dams (non-linearity of the weak interfaces and bond-slip interaction at the anchor-structure grouting interfaces) is realized. A cohesive-frictional contact modeling scheme is proposed to simulate the weak interfaces. This scheme exploits the functions of ABAQUS. It inserts cohesive elements between contact surfaces employing the surface-to-surface (STS) contact modeling of ABAQUS. In this way, the cohesive fracture (normal direction) and the shear resistance consisting of friction and cohesion (tangential direction) of the weak interfaces are properly simulated. Besides, cohesive elements connecting the anchor and structure are utilized to model the bond-slip interaction at the bond length in a PTA system. They are assigned with a quadrilinear constitutive law in the tangential direction to model the bond-slip phenomena. This bond-slip model has a relative simple expression and features with a yielding plateau. Its damage evolution can be defined in a tubular form available in ABAQUS.

Automatic mesh-forming techniques are developed based on the polygonal S-elements to facilitate the modeling schemes. These techniques convert non-conforming meshes to conforming ones at the interfaces. Cohesive elements can be generated using the node pairs at the interfaces, which avoids introducing interface constraints for the connection between the cohesive elements and surrounding elements. Surface conforming meshes (both nodes and edges are matching) are obtained by shifting/inserting nodes and spiting edges of polygonal S-elements at the interfaces. They are used for the cohesive-frictional contact modeling. Nodal conforming meshes (matching nodes along desired lines) are generated by inserting additional nodes into polygonal S-elements along the anchor lay-
Figure 5.15. Results of pull-out test simulation
outs and duplicating the inserted nodes. This mesh-forming technique automatically in-
serts anchors into pre-existing structural meshes. It shows significant advantages for the
PTA modeling involving multiple inclined anchors.

Through two benchmark tests, the correctness and performance of the proposed mod-
eling schemes on conforming meshes are demonstrated. A direct shear test on two contact
blocks obtains a force-slippage curve featuring with a peak shear strength and a residual
shear strength. The selection of governing parameters for the cohesive-frictional contact
model is examined by comparing the shear strength-normal stress (both peak and residual)
relationships with the experimental results. A pull-out simulation of a single embedded
anchor gives a progressive damaged pull force-extension relationship: i) pseudo-elastic
behavior at small deformations, ii) de-bonding development at moderate displacements,
and iii) residual behavior at larger displacements. The shear stress distributions along the
the bond length under the initial, progressive and ultimate loading conditions have the
same characteristics reported in the literature.
Chapter 6

Seismic analysis of post-tensioned gravity dams

This chapter presents the proposed numerical framework for seismic analysis of post-tensioned concrete gravity dams based on the scaled boundary finite element method (SBFEM). The framework is implemented in the commercial FEM software ABAQUS by introducing the SBFEM as user elements (UEL), as explained in Chapter 4. The characters of post-tensioned concrete gravity dams, such as the weak interfaces and anchor-structure grouting interfaces are simulated using the modeling schemes in Chapter 5. To this end, comprehensive considerations have been taken in the numerical simulations, including: 1) dam-reservoir-foundation interaction, 2) near-field/far-field coupling, 3) post-tensioned anchors, 4) weak interface simulation, 5) bond-slip interaction. Some material of this chapter has been incorporated in a journal paper titled as 'Seismic analysis of post-tensioned concrete gravity dams using scaled boundary finite elements implemented as ABAQUS UEL' accepted by Soil Dynamics and Earthquake Engineering. Co-authors of this accepted paper are acknowledged.

The outline of this chapter is as follows: A brief introduction is provided in Section 6.1 to review the research histories and explain the advantages of the SBFEM for seismic analysis of post-tensioned gravity dams. Section 6.2 describes the modeling
of post-tensioned gravity dam-reservoir-foundation interaction system. In Section 6.3, the numerical performance, i.e., robustness, accuracy, and efficiency of the proposed approach, is demonstrated through two numerical examples featuring concrete gravity dams with weak interfaces and multiple inclined post-tensioned anchors. Finally, conclusions are stated in Section 6.4.

6.1 Introduction

Seismic analysis of post-tensioned gravity dams has been carried out by several researchers. The tensile stresses within the dam body are significantly reduced by installing PTAs [58, 189, 26], which is beneficial for controlling concrete cracking. Near the dam heel, PTAs result in compressive stresses that eliminate a substantial part of the tensile stresses caused by the overturning moment [233]. Moreover, the induced compressive stresses improve the friction resistance of the dam-foundation interface, which is favorable with regard to the dam sliding stability [333, 188]. The seismic performance of post-tensioned gravity dams is strongly influenced by the layout of PTAs. Studies suggest that only installing vertical PTAs near the upstream face might be inefficient [234] or insufficient [58]. Slightly-inclined PTAs often result in larger improvements on the seismic performance of the post-tensioned dams [234], and downstream PTAs are beneficial if the cracking initiates from the downstream face [58].

The failure of a PTA system should be considered in the seismic analysis of post-tensioned gravity dams [20]. The principal failure mechanisms are a) tensile failure of the tendon, b) shear failure of the surrounding rock, and c) bond failure at the tendon-grout and rock-grout interfaces [25, 5]. Among various numerical studies, only a few of them investigated the failure of PTA systems and mainly focused on the tensile/shear failure of the tendon [26, 188]. A perfect bond at the anchor-structure grouting interfaces is usually assumed in the literature, despite the fact that an interfacial failure of the rock-grout bond, i.e., pull-out failure, is deemed to be critical to the capacity of a PTA system.
in many applications [55, 17, 2]. This might be related to the fact that the current anchor design approaches simply assume a uniform bond stress distribution along the bond length [30, 25, 5] and may result in a conservatively designed bond length such that the bond failure will not easily occur [2]. However, this empirical method heavily relies on the selection of the ultimate bond stress, while commonly recommended values [5] have been indicated to be inconsistent with measurements [334] and are outdated due to the evolution of construction technologies [335, 336]. Moreover, evidence shows that the bond stress distribution is markedly non-uniform, while an exponential decay from the proximal end to zero at the distal end of the bond length is observed [16, 81]. Consequently, a partial debonding often occurs near the proximal end [30], which will affect the load-carrying capacity of a PTA system [337] and the fundamental frequency of the structure [338].

Seismic safety evaluation of dams is a very challenging problem containing several complex sub-problems such as fluid-structure (FSI) and soil-structure interactions (SSI). Although simplified procedures are readily available to take these interactions into account, such as Westergaard’s added mass approach [61] for FSI and massless foundation models [62] for SSI, a rigorous and efficient computational framework to model dam-reservoir-foundation interaction systems for realistic problems has usually been utilized based on the finite element method (FEM) [21, 63, 64, 65, 66]. Figure 1.6 depicts a typical dam-reservoir-foundation system of a concrete gravity dam anchored by a vertical PTA. The overall system is usually partitioned into a near-field (dam and adjacent reservoir and foundation) and a far-field (reservoir and foundation extending to infinity) domains. The foundation and reservoir, as unbounded domains with dimensions extending to infinity, should satisfy the so-called radiation conditions [67, 68] absorbing outgoing waves. At the near-field/far-field interfaces, either local or global artificial boundary conditions (ABC) [69, 70, 65, 71] are constructed to capture the dynamic properties of the unbounded domains. Besides, there are also interactions within the dam body for concrete gravity dams. The concreting of a gravity dam usually involves construction joints and
separates the dam into several lifts (horizontal dashed lines in the dam body in Fig. 1.6). The interactions between dam lifts should be considered, as the lift joints are relatively weak compared to the concrete blocks [4], and the non-linear behavior of these weak joints (cracking/sliding) have a significant influence on the seismic safety evaluation of concrete gravity dams [339, 340, 341].

The scaled boundary finite element method (SBFEM), a semi-analytical numerical method originally developed by Wolf and Song [108, 109], is used for the seismic analysis of post-tensioned gravity dams due to its salient features. Firstly, the unbounded foundation can be accurately modeled by the SBFE approach by employing displacement unit-impulse response matrices [23]. An unbounded element formulated by the SBFEM (unbounded S-element) only requires the discretization of the near-field/far-field interface. It can rigorously capture the radiation condition at infinity and transfer the free-field motions as equivalent seismic inputs [166, 66]. Moreover, polygonal elements derived by the SBFEM (polygonal S-elements) offer great flexibility regarding spatial discretization for the dam and near-field foundation. They are highly complementary with quadtree meshes [84, 295] allowing smooth and rapid transitions in element size. Local mesh refinement can be easily performed at the regions where high-stress gradients are expected, such as the regions near the anchor head and the bond length, to obtain accurate solutions. Last but not least, polygonal S-elements provide the convenience of generating conforming interfacial meshes for the simulations of the structural interfaces [231] and bond-slip interaction. The non-linear behavior of the weak interfaces can be simulated by a cohesive-frictional scheme, i.e., inserting cohesive elements in conjunction with establishing contact pairs at the interfaces. This scheme has been proved to be a reliable tool to model weak interfaces and can be conveniently operated on conforming meshes [86, 89]. The bond-slip interaction can be represented by linkage [90] or interface elements [91, 92, 93] that connect the anchor and structure. These elements have to be implemented on conforming meshes [87].
6.2 Modeling of post-tensioned dam-reservoir-foundation system

Two types of models for post-tensioned gravity dams are considered in this chapter, one model using the proposed S-elements for the dam and foundation (both near-field and far-field), and one model using the ABAQUS’ built-in element library for comparison. In the following, the model employing user-defined S-elements is named as SBFEM model, while the model using ABAQUS’ built-in elements is referred as ABAQUS model. However, keep in mind that both models utilize the solvers provided by ABAQUS. The modeling considerations for dam-reservoir-foundation interaction systems are described in this section. The advantages of unbounded S-elements have been shown in many other researches [166, 269, 342, 66]. Basically, an unbounded S-element only requires a small truncation area to rigorously satisfy the radiation condition of an unbounded medium. In this section, the benefits resulting from the use of polygonal S-elements to generate conforming interfacial meshes are highlighted.

A sketch of the models for the dam-reservoir-foundation interaction system based on ABAQUS’ built-in elements or the SBFEM elements implemented by means of a UEL is depicted in Fig. 6.1. Plane-strain conditions are assumed for both models. The main differences between the two models are: i) the dam and near-field foundation are constructed by plain strain elements (CPE4) in the ABAQUS model, while in the SBFEM model they are discretized as polygonal S-elements with smooth transitions in element size; ii) the far-field is represented by a classical spring-dashpot boundary [140, 70] in the ABAQUS model, and unbounded S-element in the SBFEM model. The ABAQUS model has a relatively large near-field foundation that extends three times the dam height ($h$) in the upstream, downstream, and downward directions, respectively. The foundation size is selected for a reasonable truncation boundary employing spring-dashpot elements [118]. In the SBFEM model, the extended length in the upstream direction is $3h$ to consider the reservoir-foundation interaction. In the downstream and downward directions, the
extended lengths can be reduced to $h$ and $1.5h$ owing to the unbounded S-elements [66]. An accurate representation of the far-field foundation can be provided by the unbounded S-element that only requires a small number of surface finite elements at the near-field/far-field interface [23].

The spring-dashpot boundary in the ABAQUS model consists of parallel-connected spring and dashpot elements in both normal and tangential directions as shown in Fig 6.1a. The values of the stiffness coefficient of each spring element and the damping coefficient of each dashpot element can be determined using Eq. (4.15). The equivalent seismic forces due to free field motions acting on the truncated boundary are given in Ref. [280], and therefore, they are not repeated here for the sake of brevity.

The reservoir modeling is the same for the ABAQUS and SBFEM models. Acoustic elements (AC2D4) are utilized for representing the near-field reservoir, and the same boundary conditions are applied [59]. At the fluid-structure interface, tie constraints are introduced to fulfill the compatibility between hydrodynamic pressures $p$ and normal displacements. In this thesis, the absorption effect due to the existence of a sediment layer at the bottom of the reservoir [343] is neglected. The free surface of the reservoir is imposed by setting zero hydrodynamic pressure $p = 0$ ignoring gravity waves. The radiation condition at infinity is satisfied by employing an improved planar nonreflecting acoustic boundary embedded in ABAQUS at the far end of the reservoir.

The anchor is represented by ABAQUS’ built-in beam elements (B21) for both models. The prestressing force in a PTA is imposed by assigning initial stress and detailed modeling considerations of the PTA system are presented in Section 5.2.2.

The interfacial problems existing in post-tensioned gravity dams are simulated using the modeling schemes presented in Chapter 5 exploiting the features of ABAQUS. It is worth to mention that, non-conforming meshes at the interfaces are inevitable if different domains are independently discretized. The automatic mesh-conforming techniques proposed in Section 5.3 are utilized to convert the non-conforming meshes to conforming ones in the SBFEM models.
Figure 6.1. Modeling of dam-reservoir-foundation interaction system
6.3 Numerical examples

Two numerical examples are presented in this section to verify the accuracy and to demonstrate the advantages of the proposed approach. The first example is the Manly dam with a single vertically configured PTA. For such post-tensioned gravity dams with a simple geometry, conforming meshes are relatively easy to generate in ABAQUS. The results obtained with ABAQUS are used for verification. In the second example, a dam that has three lift joints and four PTAs (vertical and inclined) is analyzed, through which the advantages of the proposed approach in handling complex problems are demonstrated.

6.3.1 Manly dam

Manly dam is a concrete gravity dam constructed in 1892 in a northern suburb of Sydney, Australia. In 1981, due to a higher PMF and the non-inclusion of uplift in the original design, the dam was strengthened by installing PTAs, which was considered to be the optimal solution in comparison with alternative techniques [56]. The dam has a height $h$ of 19 m and length of 250 m, along which 46 PTAs with different numbers of strands and dimensions were installed. Each strand has a minimum breaking load (MBL) of 250 kN, and the PTAs were post-tensioned to 78% MBL and finally locked off at 70% MBL. The PMF leads to a maximum reservoir depth of 18.52 m.

Figure 6.2 depicts the dimensions of the dam’s cross section and the PTA configuration, in which the relative coordinates of the anchor end to the dam heel are presented. The largest PTA system among all anchors is chosen in the following analysis, which has 24 strands with 15.2 mm diameter, a free length $l_f$ of 33.8 m, a bond length $l_b$ of 6.2 m, and a borehole diameter $d_h$ of 215 mm. The material properties of the dam body, foundation, and anchor are listed in Table 6.1. Rayleigh damping coefficients $\alpha_R$ and $\beta_R$ are
determined by [344]

\[
\begin{bmatrix}
\alpha_R \\
\beta_R
\end{bmatrix} = \frac{\chi}{\pi (f_1 + f_2)} \begin{bmatrix}
4\pi^2 f_1 f_2 \\
1
\end{bmatrix},
\]

where, \( \chi \) is the average Rayleigh damping, and \( f_1 \) and \( f_2 \) are the lower and upper bounding frequencies of the dominant frequency interval. In this example, a Rayleigh damping of \( \chi = 5\% \) is assumed, and \( f_1 = 3.24 \) Hz and \( f_2 = 19.97 \) Hz are obtained by performing modal analysis. \( E \) and \( B \) denote the Young’s and bulk moduli, respectively. The cohesive-frictional contact parameters of the dam-foundation interface (sandstone-concrete contact) are based on the EPRI’s report [4], and their values are listed in Table 6.2. The parameters of the bond-slip (quadrilinear) model are mainly taken from the PTI recommendation [5] on the sandstone grouting interface, and their values are given in Table 6.3.

The meshes of the ABAQUS and SBFEM models for the dam-reservoir-foundation systems are depicted in Fig. 6.3. The modeling considerations have been explained in
Table 6.1. Material properties of Manly dam-reservoir-foundation system

<table>
<thead>
<tr>
<th>Material</th>
<th>Concrete (dam)</th>
<th>Sandstone (foundation)</th>
<th>Steel (anchor)</th>
<th>Water (reservoir)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus [GPa]</td>
<td>25 ($E$)</td>
<td>10 ($E$)</td>
<td>210 ($E$)</td>
<td>2.07 ($B$)</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.2</td>
<td>0.2</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>Density $\rho$ [kg/m$^3$]</td>
<td>2500</td>
<td>2400</td>
<td>7900</td>
<td>1000</td>
</tr>
<tr>
<td>$\alpha_R$ [1/s]</td>
<td>1.76</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$\beta_R$ [s]</td>
<td>0.000685</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 6.2. Cohesive-frictional contact parameters of concrete-sandstone joint [4]

<table>
<thead>
<tr>
<th>$\mu_s$</th>
<th>$\mu_k$</th>
<th>$d_c$ [mm]</th>
<th>$s_c$</th>
<th>$E_n^0$ [GPa]</th>
<th>$E_t^0$ [GPa]</th>
<th>$\sigma^0$ [MPa]</th>
<th>$\tau^0$ [MPa]</th>
<th>$\delta_f$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.14451</td>
<td>0.5543</td>
<td>100</td>
<td>1</td>
<td>18</td>
<td>1.8</td>
<td>0.9</td>
<td>1.8</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Section 6.2. Therefore, here we only describe the meshes of the two models. In the ABAQUS model, there are 2,578 elements for the dam, 8,667 elements for the foundation, 2,668 acoustic elements for the reservoir, and 119 beam elements for the anchor. There are 20 and 98 cohesive elements (COH2D4) associated with the bond and free lengths, respectively, and 253 cohesive elements for the dam-foundation interface. The far-field foundation is represented by 542 spring and 542 dashpot elements. At the dam-foundation interface, the element size is about 0.3 m. The bond length of the anchor is discretized into 20 beam elements. To create a conforming mesh, the elements are extruded laterally, leading to the two strips of finer element size in the foundation. The element size for the remaining area is about 1.5 m to guarantee a reasonable aspect ratio. The SBFEM model is discretized through a quadtree-based meshing algorithm [295, 345]. The dam domain has minimum and maximum element sizes of 0.125 m and 1 m, respectively. The near-field foundation has minimum and maximum element sizes of 0.3125 m and 5 m,

Table 6.3. Parameters of quadrilinear model for sandstone-grout interface

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$\tau_u$[MPa] [5]</th>
<th>$\tau_t$[MPa] [16]</th>
<th>$s_1$[mm] [17]</th>
<th>$s_2$[mm] [17]</th>
<th>$s_3$[mm] [17]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>4.2</td>
<td>0.42</td>
<td>1.5</td>
<td>2.25</td>
<td>4.5</td>
</tr>
</tbody>
</table>
respectively. The smallest element sizes are applied for the zones with possible high-stress gradients, such as the anchor head, dam-foundation interface, and bond length. In this way, without increasing the mesh sizes in those featured zones, there are only 1,209 and 964 polygonal S-elements for the dam and foundation, respectively, since quadtree meshes offer smooth and rapid transitions in element size. The far-field foundation is represented by one 41-node unbounded S-element that has a scaling center at the dam heel. The other elements (taken from ABAQUS’ element library) used in the SBFEM model are 133 beam elements for the anchor, 2,376 acoustic elements for the reservoir, 197 cohesive elements used for cohesive-frictional contact, and 21 and 111 cohesive elements associated with the bond length and free length, respectively.

To reflect the anchor spacing in a two-dimensional model, the thickness of each element should be adjusted. With an assumption of a 5 m anchor spacing, plane strain state with 5 m thickness is employed for the elements of the dam, foundation, and reservoir, which is also adopted for the cohesive elements used for contact. The beam elements are assigned with a circular section with 37.2322 mm radius leading to a total cross section area same with 24 strands (15.2 mm diameter). The thickness of cohesive elements for anchor-structure interaction, is calculated by \( \pi d_h \) as 0.67544 m, where \( d_h \) denotes the borehole diameter.

To evaluate the accuracy of the proposed approach, the SBFEM model is compared to the ABAQUS model in aspects of:

1. Verification of the unbounded S-element and seismic input implementations on a linear elastic dam-reservoir-foundation system subjected to a Ricker wavelet excitation;

2. Preliminary evaluation by performing pseudo-static analysis on a non-linear modeling considering the cohesive-frictional contact and bond-slip;

3. Comprehensive evaluation through transient analysis imposing a real recorded seismic wave.
Figure 6.3. Meshes of dam-reservoir-foundation interaction system
6.3.1.1 Verification of unbounded S-element

A linear elastic model, i.e., the dam-foundation interface and the bond length are perfectly bonded, is analyzed to verify the unbounded S-element implementation. The dam-reservoir-foundation system is only subjected to a vertically propagating S-wave from the bottom boundary. No other external forces, such as gravity, prestressing, etc., are applied. The acceleration history of the S-wave is a Ricker wavelet expressed as

\[ A(t) = A_{\text{max}} \left\{ 1 - 2 \left[ \pi f_p (t - t_0) \right]^2 \right\} e^{-\left[ \pi f_p (t - t_0) \right]^2}, \]  

(6.2)

where \( A_{\text{max}} \), \( f_p \) and \( t_0 \) are the maximum amplitude, predominant frequency of the Fourier spectrum and the time of peak acceleration, respectively. In this example, those parameters are set as \( A_{\text{max}} = 0.5g \), \( f_p = 5 \) Hz and \( t_0 = 0.5 \) s, which yields a wave with frequencies approximately in the range of \( 0 - 15 \) Hz. The time history of the Ricker wavelet acceleration and its Fourier transform are shown in Fig. 6.4.

The transient analyses are performed with the default ABAQUS dynamic, implicit solver (HHT-\( \alpha \) method with \( \alpha = -0.05 \)) with a maximum time increment \( \Delta t_{\text{max}} = 0.0005 \) s. The discretized time interval \( \Delta \bar{t} \) used for calculating \( S^T_j \) is also chosen as 0.0005 s. The truncation time \( n\Delta \bar{t} \) is set as 0.1 s. The chosen element sizes and time increments meet the requirements [346, 66] for modeling wave propagation problems. The imposed Ricker wavelet with an maximum frequency 15 Hz yields a shear wavelength of 87.84 m \(( c_s = 1317.62 \) m/s), which allows 70 and 17 nodes per wavelength for the ABAQUS and SBFEM models, respectively. The maximum time increment allowing the waves to travel through one element for the ABAQUS and SBFEM models are 0.0008 s and 0.0038 s, respectively. These requirements are also satisfied for the following transient analyses with a recorded seismic wave.

The time histories of the horizontal relative displacement between the dam crest and heel, and the force in the cable are compared in Fig. 6.5. An excellent agreement between the seismic responses obtained from the ABAQUS model and the SBFEM model is ob-
Figure 6.4. Ricker wavelet
served. The correctness of the unbounded S-element implementation is verified through this example. Besides, the unbounded S-element needs a smaller truncation area compared to the spring-dashpot boundary, which further reduces the near-field foundation size requirement and saves computational resources.

### 6.3.1.2 Pseudo-static analysis

Pseudo-static analysis, also known as seismic coefficient method, is conventionally used to evaluate the seismic stability of structures [169]. In this method, earthquake loadings are simply treated as inertial forces applied statically to the structure, i.e., the inertia of concrete and reservoir. The inertia forces are computed by the product of structural mass, added mass of water [61] times the earthquake acceleration. The ratio of the earthquake acceleration to gravity is named as seismic coefficient $\alpha$.

In this example, the applied loadings consist of static forces (black) and inertia forces (red), as shown in Fig. 6.6. The static forces are caused by the gravity of the dam, the prestressing of the anchor, the hydrostatic pressure, and the uplift. Initial stress of 1.07463 GPa is introduced in the beam elements of the free length, which yields a pre-tension of 4680 kN (78% MBL). The uplift pressure distribution along the dam-foundation interface is expressed as:

$$P_u(x) = (1 - E_d)H_s(1 - \frac{x}{W}),$$

(6.3)

in which $H_s$ is the hydrostatic pressure at the dam heel, $W$ is the width of the dam foundation, $E_d$ is the drain effectiveness, which is chosen as 25 percent [19], and $x$ is the distance to the dam heel. The structural inertia and hydrodynamic pressure are imposed in the horizontal direction only [19]. The hydrodynamic pressure distribution is derived from Westergaard added mass [61] as follow:

$$H(y) = \frac{7}{8} \rho_w \alpha \sqrt{h_r(h_r - y)},$$

(6.4)
Figure 6.5. Seismic responses of Manly dam subjected to a Ricker wavelet
in which \( \rho_w \) is the water density, \( h_r \) is the height of reservoir, \( \alpha \) is the seismic coefficient and \( y \) is the height from the bottom of reservoir.

The pseudo-static analyses are performed by increasing the seismic coefficient \( \alpha \) until the PTA is fully pulled out (only residual bond stress left). The structural responses obtained from the two models are compared in Fig. 6.7, including the horizontal dam/crest relative displacement and the crack opening at the dam heel. It can be observed that, before a specific value of \( \alpha \) (0.893 \( g \) for ABAQUS model, 0.867 \( g \) for SBFEM model) the structural response of each model is very small. At these values, sudden changes of the responses occur. For clarification, the displacement contours (ABAQUS model) before and after the sudden change are plotted in Fig. 6.8. With a minor seismic coefficient change of 0.0004 \( g \), noticeable cracking at the dam-foundation interface occurs, leading to significant dam deformation (rotation about the dam toe). After which moderate structural responses are obtained until the PTAs are pulled out. For both models, an ultimate seismic coefficient of \( \alpha = 1.1 \ g \) is obtained. Generally speaking, the results have good
agreements except for a small difference at the sudden structural response.

The development of the normal stress along the dam-foundation interface is examined to investigate the crack propagation, as shown in Fig. 6.9. At $\alpha = 0.6150 \, g$, the stress at dam heel reaches 0.9 MPa. This value is the tensile strength $\sigma_{\text{max}}$ of the dam-foundation interface, as specified in Table 6.2, which implies the cracking initiates at the dam heel. Afterward, the crack propagates gradually from the dam heel towards to the dam toe with the increase of the seismic coefficient. From $\alpha = 0.6150 \, g$ to $\alpha = 0.8935 \, g$, the crack propagation is relatively slow and only generates a completely cracked width of 2.4 m (zero tensile stress). During the progress of the sudden structural response, i.e., $\alpha \in (0.8935 \, g, 0.8939 \, g)$, the cracking rapidly propagates for a distance of 12.8 m. The crack propagation progress coincides with the structural responses presented in Figs. 6.7–6.8.

With a special focus on the PTA behavior, the development of the post-tension cable force and the extension of the proximal end are recorded, as presented in Fig. 6.10. The extension of the proximal end indicates the relative displacement between the bond length proximal end and the structure, and it is the integrated slippage of the whole bond length. Similarly, the post-tension cable forces and the proximal end extension have negligible variations before the sudden structural response, after which they moderately increase until the PTA is pulled out. The obtained ultimate cable forces are 6056kN and 6042kN for ABAQUS and SBFEM models, respectively. They are slightly higher than the MBL of the PTA system (6000kN), which implies the bond length is adequate.

### 6.3.1.3 Transient analysis

Transient analyses of the Manly dam-reservoir-foundation system are performed by imposing a modified Taft wave [66] with a scaled peak horizontal acceleration of 0.5 g. The acceleration history and Fourier transform are shown in Fig. 6.11. Before the transient analysis starts, the static forces, including the gravity of the dam, prestressing, uplift, and hydrostatic pressure, are assigned to the model through a quasi-static analysis step. The
Figure 6.7. Structural responses of Manly dam in pseudo-static analysis
Figure 6.8. Sudden change of structural response (unit: m, deformation scale factor: 25)

Figure 6.9. Normal stress along dam-foundation interface for different $\alpha$
Figure 6.10. PTA behavior of Manly dam in pseudo-static analysis
ABAQUS dynamic solver with an application of moderate dissipation ($\alpha = -0.41421$) is utilized, which is the default setting for models with contact [18].

The seismic responses obtained from the ABAQUS and SBFEM models are compared in Fig. 6.12, including the histories of the dam crest/heel relative displacement and hydrodynamic pressure near the dam heel. The results obtained from the two models exhibit an excellent agreement. During the earthquake, the peak relative displacement is 17.85 mm occurring at 7.86 s, and the maximum (absolute) hydrodynamic pressure is 0.147 MPa occurring at 6.83 s.
Figure 6.12. Seismic responses of Manly dam subjected to modified Taft wave.
The histories of the cracking and slippage at the dam heel are compared in Fig. 6.13. The cracking shows a negligible difference between these two models. A maximum cracking of 11.04 mm is obtained, which will result in a complete tensile failure of the dam-foundation bond at the dam heel. However, the stability of the dam against overturning is still guaranteed by the tensions of the PTAs. The slippages obtained from the two models are always well within the static friction limitation (1 mm in this example) during the earthquake. The slight difference for the slippage does not affect the overall accuracy. The final slippages are 0.15 mm and 0.10 mm for the ABAQUS and SBFEM models, respectively.
In the next step, the PTA behavior during the seismic event is examined, as shown in Fig. 6.14. Except for a small difference in the extension of the proximal end after 7.71 s, negligible differences between the two models are observed for the remaining parts. For both models, the post-tensioned cable force varies from 4489 kN (74.82% MBL) to 4765 kN (79.41% MBL), which meets the requirement of a minimum working load of 60% MBL [56]. After the earthquake, the increase of the proximal end extension are 0.39 mm and 0.33 mm, and the losses of post-tension cable force are 11.24 kN (0.19% MBL) and 9.32 kN (0.16% MBL) for the ABAQUS and SBFEM models, respectively.

The transient analysis results are further examined by comparing them with the pseudo-static analysis results obtained from the ABAQUS model. The curves of post-tensioned
cable force-dam crest/heel relative displacement and the proximal end extension-dam crest/heel relative displacement are depicted in Fig. 6.15. The dashed parts of the pseudo-static curves are supplementary results for negative seismic coefficients. For both models, the cable force-relative displacement curve is close to the pseudo-static result. The vast majority of the dynamic results are slightly below the pseudo-static curve, especially after the dam experiences peak deformation. It indicates that a partial debonding has occurred and that the PTAs have lost some of their pre-stressing. The proximal end extension-relative displacement curves have envelopes close to the pseudo-static curve. There are non-reversible proximal end extensions for both ABAQUS and SBFEM models.

To investigate the effects of the bond-slip phenomenon, another modeling that features a perfect bond condition at the bond length is analyzed based on the SBFEM approach. The results obtained from the perfect bond model and the SBFEM model with bond-slip consideration are compared in Fig. 6.16. Figures 6.16a–6.16b have shown that there are insignificant differences between the two models regarding the relative displacement and the cracking. However, the existence of bond-slip leads to a reduction of 129 kN lock-off cable force (initial force of the transient analysis) compared to the perfect bond condition, as shown in Fig. 6.16c. For a clear comparison, the real cable force history of the bond-slip model is shifted by 129 kN to have the same initial value with the perfect bond model. Generally speaking, the shifted cable force is lower than the force of the perfect bond model during the earthquake, especially after the moment of the peak deformation. In conclusion, the bond-slip phenomenon will result in irrecoverable stress losses in the PTA, but may have minor effects on the overall structural responses if the bond length is designed properly.
Figure 6.15. Verification of PTA behavior of Manly dam
(a) Dam crest/heel relative displacement

(b) Crack opening at dam heel

(c) Post-tensione cable force

Figure 6.16. Seismic response comparison of Manly dam with perfect bond and bond-slip
6.3.2 A dam with three joints

This example demonstrates the advantages of the proposed approach on the mesh generation for dams with complex PTA configurations, i.e., multiple inclined anchors. A concrete gravity dam with a height of \( h = 80 \text{ m} \) is depicted in Fig. 6.17. Three lift joints are considered in the transient analysis. The distance between two adjacent lift joints is 20 m. The reservoir has a depth of 65 m. There is a drainage system with a rectangular shape (2 × 2.5 m) near the dam bottom, the centroid of which has a distance to the dam heel of 7.7 m and 7.5 m in the horizontal and vertical directions, respectively.

Three different SBFEM models are compared in this example: a plain concrete dam with no PTAs and two post-tensioned dam models with two proposed anchoring schemes. Four typical PTAs are utilized in the anchoring schemes, as shown in Fig. 6.18a, and their positions are denoted by the relative coordinates of the anchor end to the dam heel. The sketches of these two post-tensioned dams are presented in Figs. 6.18b–6.18c. Scheme 1 contains the vertical PTAs only (Anchors 1 and 2), and Scheme 2 contains all the four anchors (Anchors 1, 2, 3, and 4). Anchors 1 and 2 are common ground anchors installed...
Table 6.4. Material properties of the dam-reservoir-foundation system for a dam with three lift joints

<table>
<thead>
<tr>
<th>Material</th>
<th>Concrete (dam)</th>
<th>Granite (foundation)</th>
<th>Steel (anchor)</th>
<th>Water (reservoir)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus [GPa]</td>
<td>35 (E)</td>
<td>27 (E)</td>
<td>210 (E)</td>
<td>2.07 (B)</td>
</tr>
<tr>
<td>Poisson’s ratio ν</td>
<td>0.2</td>
<td>0.2</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>Density [kg/m³]</td>
<td>2400</td>
<td>2650</td>
<td>7900</td>
<td>1000</td>
</tr>
<tr>
<td>αₐR [1/s]</td>
<td>0.67</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>βₐR [s]</td>
<td>0.0037</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

vertically near the upstream. Anchors 3 and 4 are inclined anchors. Anchor 3 is a ground anchor to resist downstream sliding [2], and Anchor 4 is used to control the cracking of lift joints [26]. Each vertical anchor consists of 65 strands (15.2 mm diameter), and its borehole has a diameter of 310 mm. Each inclined anchor consists of 36 strands (15.2 mm diameter), and its borehole has a diameter of 230 mm. The MBL of one strand is 250 kN. All PTAs have an anchor spacing of 1.5 m, and they are post-tensioned to 75% MBL.

The material properties of the dam-reservoir-foundation system are summarized in Table 6.4, in which the Rayleigh damping coefficients αₐR and βₐR are chosen to introduce an average Rayleigh damping of χ = 5% for the overall system following Eq. 6.1. Granite rock is selected as the foundation for this example. The dam-foundation interface consists of a granite-concrete contact, and the lift joints are modeled as concrete-concrete contact. The cohesive-frictional contact properties for these two types of weak joints have been suggested by EPRI [4], as listed in Table 6.5. Table 6.6 presents the parameters governing the quadrilinear models used for the bond-slip of granite-grout (Anchors 1, 2, and 3) and concrete-grout (Anchor 4) interfaces.

The mesh of the dam-foundation reservoir system (Scheme 2) is presented in Fig. 6.19. Again, the near-field foundation has extended lengths of 3h, h, and 1.5h in the upstream, downstream and downward directions, respectively. The quadtree meshing technique is used for generating the original meshes of the dam and near-field foundation. The minimum and maximum mesh sizes for the dam domain are 0.25 m and 2 m, respectively. For
Figure 6.18. Anchoring schemes for a dam with three lift joints
Table 6.5. Cohesive-frictional contact parameters for weak joints in a dam with three lift joints [4]

<table>
<thead>
<tr>
<th></th>
<th>Dam-foundation</th>
<th>Lift joint</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mu_s$</td>
<td>1.35142</td>
<td>1.53986</td>
</tr>
<tr>
<td>$\mu_k$</td>
<td>0.66189</td>
<td>1.11061</td>
</tr>
<tr>
<td>$d_c$ [mm]</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>$s_c$ [mm]</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>$E_0^0$ [GPa]</td>
<td>18</td>
<td>1.6</td>
</tr>
<tr>
<td>$E_i^0$ [GPa]</td>
<td>1.2</td>
<td>2.1</td>
</tr>
<tr>
<td>$\sigma^0$ [MPa]</td>
<td>0.9</td>
<td>1.3</td>
</tr>
<tr>
<td>$\tau^0$ [MPa]</td>
<td>1.2</td>
<td>2.1</td>
</tr>
<tr>
<td>$\delta_f$ [mm]</td>
<td>0.5</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Table 6.6. Parameters of quadrilinear models for bond length of different anchors for a dam with three lift joints [5, 16, 17]

<table>
<thead>
<tr>
<th>Anchor</th>
<th>$\tau_u$ [MPa]</th>
<th>$\tau_i$ [MPa]</th>
<th>$s_1$ [mm]</th>
<th>$s_2$ [mm]</th>
<th>$s_3$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Anchors 1,2 and 3</td>
<td>8.00</td>
<td>0.80</td>
<td>1.25</td>
<td>2.50</td>
<td>5.00</td>
</tr>
<tr>
<td>Anchor 4</td>
<td>4.20</td>
<td>0.42</td>
<td>1.50</td>
<td>3.50</td>
<td>7.50</td>
</tr>
</tbody>
</table>
the near-field foundation, they are 0.5 m and 8 m, respectively. As shown in the details of Fig. 6.19, the zones where high-stress gradients may exist are discretized into fine meshes (minimum size or two times minimum size), such as the dam-foundation interface, lift joints, bond lengths, zones near the anchor heads and the drainage systems. The remeshing procedures presented in Section 5.3 are then taken for the modeling of the weak joints and PTA systems. At the end, there are 3,871 and 2,264 polygonal S-elements for the dam and near-field foundation, respectively, one 88-node unbounded S-element (scaling center located at the dam heel), 508 beam elements for the anchors, 8,240 acoustic elements for the reservoir, and 1,092 cohesive elements (587 for the weak joints and 505 for the anchor-structure grouting interfaces).

Two analysis steps are involved in this example, i.e., a quasi-static analysis step to impose static forces, followed by a transient analysis step subjected to a free-field motion. The static forces contain the contributions from the dam gravity, hydrostatic, uplifts on weak joints, and initial stresses of the PTAs. The uplift pressure varies linearly following Eq. (6.3), in which $E_d$ equals to 0.25 for the dam-foundation interface, and 0.5 for the lift joints. The initial stresses of the PTAs (75% MBL) are 1.033 GPa. Transient analysis is performed by imposing the El Centro wave (vertically propagated S-wave). The
Figure 6.20. El Centro wave

acceleration history and Fourier transform of this wave are presented in Fig. 6.20. The maximum time increment $\Delta t^{\text{max}}$ of the transient analysis and the discretized time station $\Delta \bar{t}$ of unbounded S-element are set as 0.00025 s, and the truncation time $n\Delta \bar{t}$ is set as 0.1 s.

The dam crest/heel relative displacement (horizontal) histories of these three models are compared in Fig. 6.21. The plain concrete dam has a final relative displacement of 243.01 mm, which indicates that the dam has experienced irreversible displacements during the earthquake. The two post-tensioned dams have similar final relative displacements
Figure 6.21. Seismic response of a dam with three lift joints subjected to El Centro wave with their initial ones. During the earthquake, the displacement magnitude of the dam with anchoring Scheme 1 is slightly larger than that of the dam with anchoring Scheme 2. It meets the expectation since Scheme 2 has two more PTAs strengthening the dam compared to Scheme 1. Displacement contours of these three models at 5.00s are shown in Fig. 6.22. At this time, irreversible displacement has occurred in the plain concrete dam, and it is mainly caused by the sliding occurring at the dam lift joints. The sliding of a higher joint is larger than that of a lower joint. Besides, the highest block has substantial rotation around its lower-left corner. The weak bonds of Lift Joints 2 and 3 have been completely damaged. Both post-tensioned dams have no significant sliding at the lift joints. It is because the PTAs introduce additional compressive stresses and further increase the frictional resistances of the lift joints. However, cracking is observed at the right end of Lift Joint 2 for the dam anchored by Scheme 1.

To investigate the effects of PTAs on the weak joint behavior and to evaluate the two anchoring schemes, the slippage of Lift Joint 1 and the cracking of Lift Joint 2 (right ends) are examined, as shown in Fig. 6.23. Starting from 4.53 s, Lift Joint 1 of the plain concrete dam experiences several rapidly increasing slippages (kinetic friction) and finally has an accumulated sliding of 39.72 mm. This result again provides evidence that the irreversible relative displacement of the plain concrete dam (see Fig. 6.21) is caused by the downstream sliding of the lift joints. Both anchoring schemes are efficient to
Figure 6.22. Displacement contours of three models for a dam with three lift joints (time: 5.00s, unit: m, deformation scale factor: 25)

prevent the sliding because the slippage always keeps within the static friction limitation (1 mm) for both post-tensioned dams. However, Anchor 3 of Scheme 2 is unnecessary since even without it the downstream sliding of Lift Joint 1 does not occur (Scheme 1). The maximum cracking (right end of Lift Joint 2) during the earthquake is 27.14 mm, 66.89 mm and 20.92 mm for the plain concrete dam, the dam with Scheme 1 and the dam with Scheme 2, respectively. Besides, the dam anchored by Scheme 1 still has small crack after 35 s, while the other two dams have invisible crack openings. Generally speaking, the cracking is slightly reduced by Scheme 2, but increased by Scheme 1. It is not surprising since the cracking is mainly affected by the tensile stresses. The post-tensioned tendons of Scheme 1 are near the upstream face, and they will generate tensile stresses at the downstream face. Figure 6.24 compares the first principal stress contours of these three models before significant crackings occur (1.50s). It is clear that the dam anchored by Scheme 1 exhibits higher tensile stresses than the plain concrete dam near the downstream face. Since Anchor 4 produces compressive stresses near the downstream face, the tensile stresses of Scheme 2 are lower than that of the plain concrete dam in this area.

The seismic responses of Anchor 1 in the two anchoring schemes are depicted in Fig. 6.25, in which the post-tension cable force and the proximal end extension are com-
Figure 6.23. Lift joint behavior of a dam with three lift joints subjected to El Centro wave
During the earthquake, the cable force of Anchor 1 varies from 11,577 kN (71.24% MBL) to 12,198 kN (75.06%) for Scheme 1, and it varies from 11,936 kN (73.45% MBL) to 12,869 kN (79.19% MBL) for Scheme 2. Non-recoverable extensions always occur in accompany with significant force increases. For example, the cable force in Scheme 2 suddenly increases to 12,214 kN around 2.40 s (see Fig. 6.25a), which leads to that the proximal end extension increases from 9.30 mm to 10.32 mm (see Fig. 6.25b). The extension recovers to 10.00 mm afterwards, which indicates that an non-recoverable extension of 0.70 mm has occurred. At the end, the increase of the non-recoverable extension is 2.01 mm and 3.54 mm for Scheme 1 and Scheme 2, respectively. This difference mainly occurs at 5.11 s, at which time the proximal end extension has no significant changes in Scheme 1 but increases by around 1.50 mm in Scheme 2. The final post-tension losses of Anchor 1 are 61 kN and 88 kN for Scheme 1 and Scheme 2, respectively.

To conclude, both anchoring schemes are efficient for strengthening the dam with three lift joints as they have eliminated the sliding occurring at the lift joints in the plain concrete dam. The post-tension cable forces have reasonable variations during the El Centro earthquake for these two schemes, i.e., the maximum forces do not reach the MBL limit, and the minimum forces still guarantee adequate working capacities. The post-
Figure 6.25. PTA (Anchor 1) behavior of a dam with three lifts subjected to El Centro wave.
tension losses are comparably small, and the PTAs are not pulled out, which indicates that the bond length designs are appropriate. Compared to Scheme 1, Scheme 2 contains additional PTAs to resist the downstream sliding (Anchor 3) and to control the cracking near the downstream face (Anchor 4). It has been shown that Anchor 3 has minor effects on the sliding resistance, while Anchor 4 significantly reduces the tensile stresses near the downstream face, and it is beneficial for cracking control. An improved anchoring scheme containing Anchors 1, 2, and 4 might be suggested for this example.

6.4 Conclusion

This chapter presents an efficient numerical approach for the seismic analysis of post-tensioned gravity dams based on the scaled boundary finite element method. The analyses are performed on the ABAQUS platform, with introducing the SBFEM through user elements. Polygonal user elements provide greater flexibility in mesh generation for bounded domains. Local mesh refinement is easily realized for areas with high-stress gradients, such as the anchor head and the tendon bond length. Unbounded (infinite) user elements only require a small truncation area to model wave propagation in the far-field foundation and satisfy the radiation condition at infinity. This salient feature significantly reduces the number of elements of the near-field foundation in near-field/far-field coupling problems, and further improves the computational efficiency.

Comprehensive consideration has been directed towards the interfacial behavior existing in post-tensioned gravity dams, such as the non-linear behavior of weak joints and the bond-slip interactions at the tendon along the bond length. The advantages of ABAQUS and SBFEM are combined together on dealing with these interfacial problems. ABAQUS offers robust contact modeling schemes and sophisticated cohesive elements. The cohesive elements are utilized to simulate both the cohesion of weak joints and the bond of the anchor-structure grouting interface. With the availability of polygonal user elements, conforming meshes can be generated at the interfaces in a fully automatic manner. Co-
hesive elements can be included conveniently based on the matching nodes at interfaces, and their connections with surrounding domains are achieved naturally by sharing nodes, which avoids the additional unknowns in assigning interface constraints.

The accuracy of the proposed approach is verified by performing a seismic analysis of a post-tensioned gravity dam with a simple geometry, and comparing the results to a model using ABAQUS’ built-in elements. The advantages of this method in handling complex problems are demonstrated through a dam with three lift joints anchored by multiple inclined anchors.
Chapter 7

Implementation of a regularized damage model and application to damage analysis of dams

This chapter incorporates a regularized damage model of concrete into the proposed framework of seismic analysis of post-tensioned concrete gravity dams. The new Mazar’s damage model (μ model) [113] in the framework of continuum damage mechanics is selected to describe the constitutive behavior of the concrete. A damage variable ω (scalar) consistently includes the effects of crack opening and closure on the stiffness of the material, i.e., degradation and recovery, by separating the strains in tension and compression.

To address the mesh sensitivity issue existing in a damage analysis problem, a regularization model using fracture energy [347, 348] is introduced to enhance the μ model. In the formulation of a polygonal S-element, the damage variable evaluated at the scaling center is employed to represent the overall material degradation of the whole element. The proposed approach for damage analysis is implemented into ABAQUS in conjugation with the polygonal UELs. This damage model is verified by comparing the numerical results with the theoretical solutions of a single polygonal UEL under cyclic loading, and applied to seismic damage analysis of the Koyna dam (plain and post-tensioned).
The remaining part of this chapter is organized as follow: In Section 7.1, the basic concepts of the \( \mu \) model is presented. Section 7.2 demonstrates the regularization of the \( \mu \) model based on the fracture energy concept. Section 7.3 explains the implementation procedures of the proposed damage model into the polygonal UELs. Three numerical examples are provided in Section 7.4 to verify the implementation and demonstrate the performance of this approach on seismic analysis of dams.

7.1 Constitutive law of \( \mu \) model

The new Mazar’s damage model (\( \mu \) model) [113] is used to describe the constitutive behavior of the damaged concrete. This model belongs to the family of continuum damage mechanics. It models the concrete behavior through a coupling of elasticity and damage within an isotropic formulation. Compared to the classical Mazar’s damage model [253] which was proposed for monotonic loadings, this new model is capable of considering the effects of cracking opening and closure and suitable for cyclic loadings.

7.1.1 Equivalent strains

In the \( \mu \) model, two equivalent strains \( \varepsilon_t \) and \( \varepsilon_c \) are defined for tension and compression, respectively. The equivalent strains are expressed as:

\[
\varepsilon_t = \frac{I_\varepsilon}{2(1-2\nu)} + \frac{\sqrt{J_\varepsilon}}{2(1+\nu)},
\]

\[
\varepsilon_c = \frac{I_\varepsilon}{5(1-2\nu)} + \frac{6\sqrt{J_\varepsilon}}{5(1+\nu)},
\]

in which \( \varepsilon_1, \varepsilon_2 \) and \( \varepsilon_3 \) are the principal strains of a point within the element (usually integration points for standard FEM), \( \nu \) is the Poisson’s ratio of the material, \( I_\varepsilon \) is the first invariant of the strain tensor given by

\[
I_\varepsilon = \varepsilon_1 + \varepsilon_2 + \varepsilon_3,
\]
and \( J_e \) is directly related to the second invariant of the deviatoric part of the strain tensor and expressed as

\[
J_e = 0.5 \left[ (\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_1 - \varepsilon_3)^2 \right].
\] (7.3)

For a two-dimensional element, \( \varepsilon_1 \) and \( \varepsilon_2 \) are the in-plane principal strains and \( \varepsilon_3 \) is the out-of-plane strain. They can be derived based on strains \( \varepsilon_x, \varepsilon_y \) and \( \gamma_{xy} \), following

\[
\varepsilon_1 = \frac{\varepsilon_x + \varepsilon_y}{2} + \sqrt{\left( \frac{\varepsilon_x - \varepsilon_y}{2} \right)^2 + \frac{\gamma_{xy}^2}{4}},
\] (7.4a)

\[
\varepsilon_2 = \frac{\varepsilon_x + \varepsilon_y}{2} - \sqrt{\left( \frac{\varepsilon_x - \varepsilon_y}{2} \right)^2 + \frac{\gamma_{xy}^2}{4}},
\] (7.4b)

\[
\varepsilon_3 = \begin{cases} 
\nu \frac{\varepsilon_x + \varepsilon_y}{2} & \text{plane stress} \\
0 & \text{plane strain}
\end{cases}.
\] (7.4c)

Two thermodynamic variables \( Y_t \) and \( Y_c \) are defined to describe the maximum equivalent strains reached during the loading history in tension and compression, respectively:

\[
Y_t = \max \{ \varepsilon_t(t) \} \geq \varepsilon_{0t},
\] (7.5a)

\[
Y_c = \max \{ \varepsilon_c(t) \} \geq \varepsilon_{0c},
\] (7.5b)

where \( \varepsilon_{0t} \) and \( \varepsilon_{0c} \) are specified strains related to the thresholds of cracking and crushing, respectively.

### 7.1.2 Triaxial effect

To evaluate the strain space of an element in a general loading state, a variable \( Y \) is introduced considering the triaxial effect:

\[
Y = rY_t + (1 - r)Y_c,
\] (7.6)
where $r$ is the triaxial factor expressed as:

$$ r = \frac{\sum_{i=1}^{3} \langle \bar{\sigma}_i \rangle_+}{\sum_{i=1}^{3} |\bar{\sigma}_i|} , $$  \hspace{1cm} (7.7)

in which $\langle x \rangle_+$ indicates the positive part of $x$, $|x|$ denotes the absolute value of $x$, and $\bar{\sigma}_i$ is the component of the stress tensor $\bar{\sigma}$ related to the principal strains. In this way, the triaxial loading is considered by coupling the tensile and compressive behavior with introducing the triaxial factor $r$. In a two-dimensional problem, the stress tensor $\bar{\sigma}$ is calculated by

$$ \bar{\sigma} = \left\{ \begin{array}{c} \bar{\sigma}_1 \\ \bar{\sigma}_2 \\ \bar{\sigma}_3 \end{array} \right\} = \frac{E}{(1+\nu)(1-2\nu)} \left[ \begin{array}{ccc} 1-\nu & \nu & \nu \\ \nu & 1-\nu & \nu \\ \nu & \nu & 1-\nu \end{array} \right] \left\{ \begin{array}{c} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{array} \right\} . $$ \hspace{1cm} (7.8)

For an element in pure tension, the triaxial factor $r$ equals to 1 and the variable $Y$ equals to $Y_t$, while for an element in pure compression, the triaxial factor $r$ equals to 0 and the variable $Y$ equals to $Y_c$.

### 7.1.3 Damage evolution

The evolution of the effective damage $\omega$ is defined through the variable $Y$:

$$ \omega = 1 - \frac{(1-A)Y_0}{Y} - A \exp \left(-B(Y - Y_0)\right) , $$ \hspace{1cm} (7.9)

where $Y_0$ is the threshold for damage given as

$$ Y_0 = r\varepsilon_{0t} + (1-r)\varepsilon_{0c} . $$ \hspace{1cm} (7.10)

The variables $A$ and $B$ determine the shape of the effective damage evolution law by governing the residual stress at the complete failure and the slope of the softening branch,
respectively. They are defined as:

\[
A = A_t \left[ 2r^2 (1 - 2k) - r (1 - 4k) \right] + A_c \left( 2r^2 - 3r + 1 \right), \quad (7.11a)
\]

\[
B = r^{2-2r+2}B_t + \left( 1 - r^{2-2r+2} \right) B_c, \quad (7.11b)
\]

in which \(A_t, B_t\) are material parameters related to the tensile behavior, \(A_c, B_c\) are related to the compressive behavior, \(k\) is used to calibrate the asymptotic stress value at large displacement in shear. Those parameters should be chosen to fit the shapes of stress-strain curves under uniaxial tensile and compressive loadings, respectively. It should be noted that: when \(r = 0\) (compressive stress domain), \(A = A_c\) and \(B = B_c\); conversely, when \(r = 1\) (tensile stress domain), \(A = A_t\) and \(B = B_t\). For an element under shear loading, the following relations hold: \(r = 0.5, A = kA_t\), and \(B = 0.42B_t + 0.58B_c\).

Figure 7.1 depicts the typical stress-strain curves of the \(\mu\) model under uniaxial tensile and compressive loadings. The initial Young’s modulus (undamaged material) is \(E_0\). Under uniaxial tensile loading, when the strain reaches \(\varepsilon_{0t}\), the maximum stress value (tensile strength \(f_t\)) is obtained, and the stress softening immediately occurs with the final stress tending to zero (or considerably small) at large strains. On the other hand, under uniaxial compressive loading, there is a stress hardening region (after the strain reaches \(\varepsilon_{0c}\)) until the compressive strength \(f_c\) is reached. At larger strains, there are still residual compressive stresses. The stress-strain curves of the \(\mu\) model coincide with the softening behavior of concrete under uniaxial tension [349] and compression [350, 351].

### 7.2 Regularization of \(\mu\) model

The numerical prediction of cracking strongly depends on the mesh of finite element discretization [261, 242, 220]. It is because the local damage model is used to describe the constitutive behavior of the material at the integration points. In FE analysis, the localization of the damage often occurs on a row of integration points due to stress softening [352], and thus the numerical solutions are sensitive to the mesh size, mesh orientation.
and element type, i.e., mesh sensitivity.

To address the mesh sensitivity issue mentioned above, the Mazar’s damage model has been regularized by a non-local approach using Gauss weight functions [353] or a crack band approach [113]. The non-local approach evaluates the damage variable of an element through a weighted spatial average over a representative volume determined by the material characteristic length [354]. Although the non-local approach is robust, it is not optimal for parallel computation since it necessitates establishing a dialogue between the Gauss points considering neighboring elements [352]. Parallel computing usually introduces mesh partitioning [355], leading to that the elements at the partitioning boundaries need special treatments to guarantee their integration volumes are truncated. The crack band approach hypothesizes that the concrete’s fracture occurs in the form of a blunt smeared crack band due to the heterogeneous aggregates [263]. The crack band width \( w_c \) can be evaluated from the fracture energy, tensile strength, Young’s modulus, and the tangent softening modulus. Within the crack band, the progressive microcracking (randomly non-homogeneous) is represented by the macroscopic continuum damage that exhibits strain-softening. Consequently, the mesh size in the damaged regions can be enforced to be consistent with the crack band width to avoid mesh-dependent solutions. This approach might not be applicable to the seismic analysis of concrete dams because: i) the crack band width is usually small compared to large-scale structures like dams resulting
in too fine meshes; ii) it is not easy to predict the damage region.

In this work, a simple and efficient regularization technique based on the fracture energy concept [13] is introduced to enhance the $\mu$ model. This method adjusts the softening branch of the $\mu$ model under uniaxial tension according to the element size. It essentially modifies the parameter $B_t$ in Eq. (7.11b) according to the element size to guarantee that the required energy for complete tensile damage remains a constant value for elements with different sizes.

### 7.2.1 Fracture energy

The fracture energy $G_f$ is the absorbed energy per unit area of a cracking surface from its unloaded state to a state of complete separation [13]. Figure 7.2a demonstrates the schematic of this crack model (opening mode). In this approach, the crack propagates when the tensile stress $\sigma_t$ at the crack tip reaches the tensile strength. The stress does not fall to zero immediately, but decreases with increasing crack width $w$. After the crack width reaches an ultimate value of $w_1$, the stress falls to zero, and the crack faces are completely separated. Thus the fracture energy $G_f$ of a material is expressed as

$$G_f = \int_0^{w_1} \sigma_t(w) dw,$$

in which $\sigma_t$ develops with the cracking width $w$. Figure 7.2b shows an example of the stress-crack width curve. The fracture energy corresponds to the area between the curve and the coordinate axes.

In this work, the fracture is represented by the isotropic damage of an element due to tension. It is referred to as smeared crack model in the literature [264]. Consequently, the required energy to complete the tensile damage of an element (dissipated energy) should be the same as the fracture energy over the crack area splitting the element. Figure 7.3 compares the energy dissipation mechanisms under the crack and damage models. Under the assumption of the crack model [13], the total dissipated energy $W_d$ of an element for
fracture should be

\[ W_d = G_f A_{ck}, \]  

(7.13)
in which \( A_{ck} \) is the crack area splitting the element. An element characteristic length \( l_{ch} \) is introduced to represent the computational width of the fracture in the FEM framework [13, 356]. If the crack is “smeared” as tensile damage over the whole element, the total dissipated energy \( W_d \) can be calculated as

\[ W_d = g_t V_e, \]  

(7.14)

where \( g_t \) is the dissipated energy for tensile damage per unit of volume, and \( V_e = A_{ck} l_{ch} \) is the volume of the element. Comparing Eq. (7.13) and Eq. (7.14), the following relation should hold:

\[ G_f = l_{ch} g_t. \]  

(7.15)
The upper bound of the available energy in the volume (in tension) can be defined as \( g_t \), given by [357]

\[ g_t = \int_0^\infty \sigma_t d\varepsilon_t, \]  

(7.16)

which corresponds to the area covered by the tensile stress-strain curve like Fig. 7.1a.
7.2.2 Regularization using fracture energy

The $\mu$ model in purely tension ($r = 1$, $A = A_t$ and $B = B_t$) should be regularized to guarantee the condition mentioned in Eq. (7.15). In this work, the compressive behavior of the $\mu$ model ($r = 0$, $A = A_c$ and $B = B_c$) does not change with the element size, and it is governed by the specified parameters $A_c$ and $B_c$. For a better explanation, here two definitions $\omega_t$ and $\omega_c$

$$\omega_t = 1 - \frac{(1 - A_t) \varepsilon_{0t}}{Y_t} - A_t \exp(-B_t(Y_t - \varepsilon_{0t})),$$

$$\omega_c = 1 - \frac{(1 - A_c) \varepsilon_{0c}}{Y_c} - A_c \exp(-B_c(Y_c - \varepsilon_{0c})),$$

are introduced to distinguish the damage evolution in purely tension and compression, respectively.

The damage evolution in purely tension $\omega_t$ is modified by adjusting $B_t$ according to the element size. Assuming there is no residual stress at the complete damage due to tension, i.e., $A_t = 1$, the tensile stress $\sigma_t$ develops with the strain $\varepsilon_t$ following

$$\sigma_t = \begin{cases} E_0\varepsilon_t & \varepsilon_t \leq \varepsilon_{0t} \\ E_0\varepsilon_t \exp(-B_t(\varepsilon_t - \varepsilon_{0t})) & \varepsilon_t > \varepsilon_{0t} \end{cases}. \quad (7.18)$$
Figure 7.4. Tensile stress-strain relationship of $\mu$ model regularized by fracture energy

Substituting Eq. (7.18) into Eq. (7.16), $g_t$ is expressed as

$$g_t = E_0 \left( \frac{\varepsilon_0^2}{2} + \frac{1}{B_t^2} + \frac{\varepsilon_0}{B_t} \right).$$

(7.19)

Consequently, the parameter $B_t$ is modified using Eq. (7.15) and Eq. (7.19), which leads to

$$B_t = \frac{\varepsilon_0 + \sqrt{\varepsilon_0^2 + 4g_s/E_0}}{2g_s/E_0}. 

(7.20)$$

in which

$$g_s = \frac{G_t}{l_{ch}} - \frac{E_0\varepsilon_0^2}{2} > 0$$

(7.21)

is the dissipated energy during the softening procedure. Figure 7.4 shows the schematic of the constitutive behavior of concrete using a fracture energy-based regularized $\mu$ model. A larger characteristic length $l_{ch}$ leads to a larger $B_t$ and more brittle behavior. On the other hand, a smaller characteristic length $l_{ch}$ reduces $B_t$ and defers the softening progress of the element. The value of parameter $B_t$ should govern that the softening branch locates in the zone between the two dash curves in Fig. 7.4. If $g_s = 0$, the parameter $B_t$ tends to infinity, and then a sudden drop of stress will occur upon damage (blue dash curve). To avoid a strain hardening (exceeding the red dash curve), the parameter $B_t$ should satisfy:

$$B_t \leq \frac{1}{\varepsilon_0}. 

(7.22)$$
To guarantee that the softening branch locates in the reasonable range (between the two dash curves in Fig. 7.4), a characteristic length $l_{ch}$ should satisfy the conditions in Eqs. (7.21–7.22). Consequently, the reasonable range of the characteristic length is

$$\frac{2G_f}{5E_0\varepsilon_{0t}^2} \leq l_{ch} < \frac{2G_f}{E\varepsilon_{0t}^2}. \quad (7.23)$$

As a summary, the material properties (parameters) used to determine the constitutive law of the $\mu$ model are: the initial Young’s modulus $E_0$, strains $\varepsilon_{0t}$ and $\varepsilon_{0c}$ related to the thresholds of damage, ratios between the residual stress and strength $A_t$ and $A_c$, fracture energy $G_f$, initial slope of the strain softening in compression $B_c$, and parameter $k$ governing the strain softening in shear. The parameter $B_t$ is unnecessary since it can be derived based on the element characteristic length and fracture energy using Eq. (7.20).

### 7.3 Regularized $\mu$ model in polygonal UEL

This section demonstrates the application of the regularized $\mu$ model in the polygonal S-elements, as well as the implementation procedures realized in ABAQUS (UEL).

#### 7.3.1 Application in polygonal S-element

The regularized $\mu$ model is based on an isotropic formulation, and thus, for an element has an original stiffness matrix $K$, the matrix for the damaged material is given by

$$K_d = (1 - \omega)K, \quad (7.24)$$

where $\omega$ is a scalar value denotes the effective damage of the whole element. The damage variable should be evaluated at a point within an element to represent the overall material degradation.

In this work, the strains ($\varepsilon_x$, $\varepsilon_y$ and $\gamma_{xy}$) at the scaling center of a polygonal S-element are used to calculate the effective damage. The fundamental formulations of the strains
at the scaling center are briefly summarized. Readers are referred to the monograph by Song [22] for more detailed derivations. Besides, the calculation of the area of a polygonal S-element $A_c$ is presented, which is used to estimate the element characteristic length $l_{ch}$.

7.3.1.1 Strains at the scaling center

The nodal displacement functions $u(\xi)$ of a polygonal S-element are similar to that of a polyhedral S-element (Eq. (3.81a)), expressed as

$$u(\xi) = \phi_{u1} \xi^+ c_1,$$  (7.25)

in which all the variables have the same definitions as in Eq. (3.81a). After obtaining the nodal displacement vector $d = u(\xi = 1)$, the integration constants can be determined as

$$c_1 = \phi_{u1}^{-1}d.$$  (7.26)

The strain field of a polygonal S-element is evaluated within an individual sector scaled from a line element at the boundary. It has a similar expression with that of a polyhedral S-element as well (Eq. (3.42)), given by

$$\varepsilon(\xi, \eta) = B_1(\eta)u^c(\xi, \xi) + \frac{1}{\xi}B_2(\eta)u^c(\xi),$$  (7.27)

where $u^c(\xi)$ are the displacement functions related to the DOFs of the line element. They are extracted from $u(\xi)$ and expressed as

$$u^c(\xi) = \phi_{u1}^c \xi^+ c_1,$$  (7.28)

in which, in the same way as $u^c(\xi)$, the displacement modes of the line element $\phi_{u1}^c$ are extracted from the displacement modes $\phi_{u1}$ of the polygonal S-element. Substituting
Eq. (7.28) into Eq. (7.27), the strain field can be explicitly expressed as

\[
\varepsilon = \sum_{i=1}^{n-2} c_i \xi^{\lambda_i^{-1}} (\lambda_i B_1 + B_2) \phi^{(u1)e}_i, \tag{7.29}
\]

in which \( n \) is the number of DOF of the polygonal S-element, \( c_i \) and \( \lambda_i \) are the \( i \)-th entry of \( c_1 \) and \( \lambda^+ \), respectively, \( \phi^{(u1)e}_i \) is the \( i \)-th column of \( \phi^{e}_{u1} \). The last two terms \( (i = n - 1, n) \) are excluded since they correspond to the transnational rigid motions and do not produce strains. The argument \((\xi, \eta)\) has been dropped for the sake of clarity. The strain field in each sector are

The strains \((\varepsilon_x, \varepsilon_y \text{ and } \gamma_{xy})\) at the scaling center \((\xi = 0)\) are considered. For a polygonal S-element, there are four eigenvalues \( \lambda_i = 1 \) \((i = n - 5, n - 4, n - 3, n - 2)\) that results in \( \xi^{\lambda_i^{-1}} = \xi^0 = 1 \) (constant strain modes). The real part of all other eigenvalues \( \lambda_i \) \((i = 1, 2, \ldots, n - 6)\) is greater than 1 leading to \( \xi^{\lambda_i^{-1}} = 0 \) at the scaling center \((\xi = 0)\). Consequently, the strains at the scaling center are equal to

\[
\varepsilon = \sum_{i=n-5}^{n-2} c_i (\lambda_i B_1 + B_2) \phi^{(u1)e}_i. \tag{7.30}
\]

It should be note that, the strains at the scaling center obtained from all sectors with an arbitrary \( \eta \) are numerically identical.

### 7.3.1.2 Characteristic length

The characteristic length \( l_{ch} \) of an element essentially represents the computational width of the fracture [358] as explained in Section 7.2.1. In a two-dimensional problem using the isotopic damage model, assuming the cracking occurs along a line across the element with a length of \( l_{cr} \), the width of the fracture should be \( A_e / l_{cr} \) \((A_e \text{ is the element area})\). Since the crack orientation in an isotopic damage model can not be detected, the characteristic length \( l_{ch} \) should be determined according to the general condition (random crack orientation). In this work, the characteristic length \( l_{ch} \) is estimated through the square root

239
of the element area $A_e$: [347]

$$l_{ch} = \sqrt{\frac{4A_e}{\pi}}. \quad (7.31)$$

The element area $A_e$ of a polygonal S-element can be calculated by summarizing the area of each sector scaled from each line element at the boundary. For a first-order polygonal S-element formed by $n_e$ line elements, there are $n_e$ triangular sectors (see Fig. 1.11a). Consequently, the element area $A_e$ is the sum of the areas of all triangles, formulated by

$$A_e = \frac{1}{2} \sum_{i=1}^{n_e} |\hat{x}^i_1 \hat{y}^i_2 - \hat{x}^i_2 \hat{y}^i_1|, \quad (7.32)$$

in which $\hat{x}^i_j$ and $\hat{y}^i_j$ $(j = 1, 2)$ are the relative Cartesian coordinates with respective to the scaling center (origin) of Node $j$ of Line $i$.

## 7.3.2 ABAQUS Implementation

This implementation is based on the programming in Section 4.2.4. A subroutine MAZARD is added between Step 6 and Step 7 in Fig. 4.5. This subroutine introduces the effective damage variable $\omega$ to compute the damaged stiffness matrix $K_d$ in Step 7 by performing Eq. 7.24. The remaining procedures to implement the damaged polygonal UEL are the same as that in Fig. 4.5. The parameters to describe the material properties, see the last paragraph of Section 7.2, can be specified in the ABAQUS input file (*.inp). They are passed into the UEL subroutine through the array PROPS (as explained in Section 4.2.4) and utilized in the subroutine MAZARD.

The flowchart of the subroutine MAZARD incorporated in UEL is shown in Fig. 7.5. The functions and specific implementations of each process are explained as follows:

1. Calculation of parameter $B_t$: In the first increment, the parameter $B_t$ should be determined to describe the damage evolution in the following steps. This parameter can be derived through Eqs. (7.20), (7.21) and (7.31). The element area $A_e$ in Eq. (7.31) are calculated by summarizing the area of each sector of a polygonal S-element
using Eq. (7.32).

2. Calculation of equivalent strains $\varepsilon_t$ and $\varepsilon_c$: This step calculates the equivalent strains $\varepsilon_t$ and $\varepsilon_c$ in the current increment using Eqs. (7.1–7.3). The principal strains used in Eqs. (7.2–7.3) can be derived by Eq. (7.4). The strains $\varepsilon_x$, $\varepsilon_y$ and $\gamma_{xy}$ are calculated at the scaling center of each polygonal UEL following the mathematical procedures presented in Section 7.3.1.1.

3. Update of thermodynamic variables $Y_t$ and $Y_c$: The initial values of $Y_t$ and $Y_c$ are specified as $\varepsilon_{0t}$ and $\varepsilon_{0c}$, respectively. If the computed equivalent strains $\varepsilon_t$ and $\varepsilon_c$ in Step 2 are larger than the current values of $Y_t$ and $Y_c$, the values of $Y_t$ and $Y_c$ should be updated by $\varepsilon_t$ and $\varepsilon_c$. In the following increments, the equivalent strains $\varepsilon_t$ and $\varepsilon_c$ are comparing with the updated values of $Y_t$ and $Y_c$. This procedure makes $Y_t$ and $Y_c$ being thermodynamic variables.

4. Consideration of triaxial effect $r$: The triaxial factor $r$ are computed following Eq. (7.7) based on the stresses $\bar{\sigma}_i$ ($i = 1, 2, 3$) in Eq. (7.8). Consequently, the variables $Y$, $Y_0$, $A$ and $B$ that couple the tensile behavior and compressive behavior are calculated by Eq. (7.6), Eq. (7.10) and Eq. (7.11), respectively.

5. Calculation of effective damage variable $\omega$: The effective damage variable $\omega$ of the current increment is computed following Eq. (7.9). This damage variable will be returned to the main subroutine, i.e., UEL, to calculate the effective stiffness matrix $K_d$ for a trial iteration.

### 7.4 Numerical examples

Three examples are presented in this section to verify the implementation of the proposed concrete damage model. A single five-node polygonal UEL is analyzed under cyclic loading in the first example, as presented in Section 7.4.1. This example is designated to examine the correctness of the implementation, by comparing the numerical results with
1. Calculation of parameter $B_t$

MAZARD subroutine starts

First increment of analysis

Yes

No

2. Calculation of equivalent strains $\varepsilon_t$ and $\varepsilon_c$

3. Update of thermodynamic variables $Y_t$ and $Y_c$

4. Consideration of triaxial effect $r$

5. Calculation of effective damage variable $\omega$

Figure 7.5. Flowchart of subroutine MAZARD incorporating concrete damage model into UEL

the theoretical solutions. In Section 7.4.2, seismic damage analysis is performed on the Koyna dam with plain concrete. The proposed concrete damage model is compared with the concrete damaged plasticity (CDP) model from ABAQUS built-in material library, by comparing the tensile damage contours of the dam. Section 7.4.3 presents the seismic damage analysis of the post-tensioned Koyna dam, which incorporates the proposed concrete damage model into the proposed framework in Chapter 6.

7.4.1 A single element under uniaxial cyclic loading

A model of one simple five-node polygonal S-element (square shape) under uniaxial cyclic loading is used to validate the implementation and examine the stiffness recovery in compression of the $\mu$ model. The dimension of this element is indicated in Fig. 7.6 with $b = 0.5$ m and $h = 1$ m, which leads to a characteristic length of $l_{ch} = 1.128$ m. The bottom of the element is constrained in the $x$-direction, and the middle node at the bottom is constrained in the $y$-direction. A cyclic displacement controlled loading $d$ is im-
posed on the top surface to generate designated strain $\varepsilon_y = d/h$. The parameters of the $\mu$ model in this example are listed in Table 7.1. A parameter $B_t = 19153$ is derived through Eq. (7.20). The setting of this model is fitted to concrete with strengths of $f_t = 2$ MPa and $f_c = 28$ MPa.

The cyclic displacement controlled loading contains four cycles that generate both tensile and compressive strains, as shown in Fig. 7.7. The maximum strains (absolute value) in Cycle $i$ are $\varepsilon_{yi}^t$ (at $T_i$) and $\varepsilon_{yi}^c$ (at $C_i$) in tension and compression, respectively. Table 7.2 lists the values of $\varepsilon_{yi}^t$ and $\varepsilon_{yi}^c$ in this example. According to Eq. (7.9) and the parameters listed in Table 7.1, the theoretical effective damage variable $\omega$ at each state is calculated and listed in Table 7.2.

The responses of the element, i.e., the developments of variable $Y$ and effective dam-

Table 7.1. Material properties of uniaxial test

<table>
<thead>
<tr>
<th>$E_0$ [GPa]</th>
<th>$\nu$</th>
<th>$\varepsilon_{0t}$</th>
<th>$\varepsilon_{0c}$</th>
<th>$A_t$</th>
<th>$G_t$ [J/m$^2$]</th>
<th>$A_c$</th>
<th>$B_c$</th>
<th>$k$</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>0.2</td>
<td>8.0e-5</td>
<td>8.0e-4</td>
<td>1</td>
<td>285</td>
<td>0.68</td>
<td>400</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 7.2. Characteristic values of strains for uniaxial test

<table>
<thead>
<tr>
<th>Time</th>
<th>$T_1$</th>
<th>$T_2$</th>
<th>$T_3$</th>
<th>$T_4$</th>
<th>$\varepsilon_{yi}^t$</th>
<th>$\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_1$</td>
<td>1.2e-4</td>
<td>1.8e-4</td>
<td>2.5e-4</td>
<td>1.0e-3</td>
<td>0.535</td>
<td>0.535</td>
</tr>
<tr>
<td>$T_2$</td>
<td>0.853</td>
<td>0.961</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$T_3$</td>
<td></td>
<td></td>
<td>1.000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$T_4$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Time</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$C_3$</th>
<th>$C_4$</th>
<th>$\varepsilon_{yi}^c$</th>
<th>$\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_1$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-2.5e-3</td>
<td>0.553</td>
</tr>
<tr>
<td>$C_2$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7e-3</td>
<td>0.906</td>
</tr>
<tr>
<td>$C_3$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-1.5e-2</td>
<td>0.981</td>
</tr>
<tr>
<td>$C_4$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-2.5e-2</td>
<td>0.990</td>
</tr>
</tbody>
</table>
age variable $\omega$ are plotted in Fig. 7.7. It is clear that $Y$ equals to $Y_t$ if $\varepsilon_y \geq 0$ (tension), and $Y$ equals to $Y_c$ if $\varepsilon_y < 0$ (compression). The thermodynamic values $Y_t$ and $Y_c$ never decrease during the entire loading progress. The effective damage variable $\omega$ has reached to 0.535 when $\varepsilon_y = Y = Y_t = \varepsilon_{y1}^{\text{I}}$ (at $T_1$), however, reduces to 0 when the strain turns to be negative (compressive). This result indicates the stiffness recovery for concrete in compression. With continuously deceasing $\varepsilon_y$, the effective damage variable $\omega$ exponentially increases from 0 ($|\varepsilon_y| = Y = Y_c = \varepsilon_{0c}$) to 0.553 ($|\varepsilon_y| = Y = Y_c = \varepsilon_{c1}^{\text{I}}$ at $C_1$). The developments of $\omega$ in the following cycles are similar to Cycle 1. In this example under an uniaxial cyclic loading, the development of $\omega$ can be described as: $\omega$ equals to $\omega_t$ (Eq. (7.17a)) if $\varepsilon_y \geq 0$ and $\omega$ equals to $\omega_t$ (Eq. (7.17b)) if $\varepsilon_y < 0$.

The stress-strain relationship of the element under the cyclic loading is shown in Fig. 7.8. The numerical result coincides with the theatrical derivation. In both tension and compression domains, the envelopes of $\sigma_y$ strictly follow the stress-strain curves. The slopes of the stress-strain relationship for unloading procedures are the damaged stiffness $(1 - w)E_0$ in all cycles.

### 7.4.2 Plain Koyna dam

Koyna dam is a concrete gravity dam that experienced a strong earthquake (Koyna earthquake) in 1967. Horizontal concrete cracks were observed on non-overflow monoliths after the earthquake [48]. The cracks were mainly distributed near the dam neck from both the upstream and downstream faces. The ground acceleration of the Koyna earthquake (horizontal component, PGA: 0.47 g) and its Fourier transform are shown in Fig. 7.9.

In this section, seismic damage analysis is performed on a typical non-overflow monolith of the Koyna dam. The geometry of the monolith is illustrated in Fig. 7.10. It has a dam height of $h = 103$ m, and a width of 70 m at the dam base. The upstream is assumed to be vertical, which is slightly different from the real construction. The depth of the reservoir is assumed to be 91.75 m.

Two types of numerical models are created for the seismic damage analysis, i.e., a
Figure 7.7. Histories of $\varepsilon_y$, $Y$ and $\omega$ of an element under uniaxial cyclic loading
Figure 7.8. Stress-strain relationship of an element under uniaxial cyclic loading.
Figure 7.9. Horizontal component of Koyna earthquake

Figure 7.10. Geometry of Koyna dam (Unit: m)
simple ABAQUS model using ABAQUS built-in elements associated with the concrete damaged plasticity (CDP) model for comparison, and SBFEM models using polygonal UELs with the proposed concrete damage model. For both types of models, plane stress condition is assumed, gravity of dam and hydrostatic pressure are considered.

The ABAQUS model is modified from an example from the ABAQUS manual [18] for the seismic analysis of the Koyna dam, as shown in Fig. 7.11a. This model considers the dam-reservoir interaction through a simple added mass approach [61], but neglects the dam-foundation interaction. The earthquake is represented by imposing the horizontal component of the Koyna earthquake at the dam base. At the end, the ABAQUS model consists of 1,200 first-order, reduced-integration, plane stress elements (CPS4R) and 57 user elements [18] as added mass.

The SBFEM model considers both the FSI and SSI by including the dam-reservoir-foundation interaction system, as illustrated in Fig. 7.11b. The near-field foundation is extended to $2h$, $h$ and $h$ towards the upstream, downstream, and downwards directions, respectively. It is discretized as a quadtree mesh (1,056 elements, minimum element size: 2.2 m, maximum element size: 8.8 m) assigned with polygonal UELs. At the boundary of the near-field foundation, a 63-node unbounded UEL is constructed to simulate the unbounded foundation. Vertically propagated shear wave is imposed at the bottom of the near-field foundation in terms of displacement $u_s(t)$ and velocity $\dot{u}_s(t)$, and transferred as surface traction $F_{fb}(t)$ acting at the boundary of the foundation (see Section 3.3). The displacement and velocity are obtained by integrating the ground acceleration of the Koyna earthquake (Fig. 7.9a). To guarantee the same seismic input at the dam base, their values should be halved since the motion is doubled at the ground surface due to the reflected waves (see Eq. (3.101)). The reservoir is modeled by 2,944 acoustic elements (element size: 3 m) with a non-reflecting boundary at the far end. Zero hydrodynamic pressure is introduced at the free surface of the reservoir. Tie constraints are introduced between different domains. Linear elastic material (no damping) is assigned to the foundation domain with a Young’s modulus of $E_f = 31.027$ GPa, a Poisson’s ratio of $\nu_f = 0.2$ and mass
Figure 7.11. Numerical models of Koyna dam
Table 7.3. Material properties of Koyna dam concrete [14, 18]

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus $E_d$ [GPa]</td>
<td>31.027</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu_d$</td>
<td>0.2</td>
</tr>
<tr>
<td>Density $\rho_d$ [kg/m$^3$]</td>
<td>2643</td>
</tr>
<tr>
<td>Tensile failure stress $\sigma_{t0}$ [MPa]</td>
<td>2.900</td>
</tr>
<tr>
<td>Rayleigh damping coefficient $\beta_R$ (s)</td>
<td>0.00323</td>
</tr>
<tr>
<td>Compressive yield stress $\sigma_{cy}$ [MPa]</td>
<td>13.000</td>
</tr>
<tr>
<td>Compressive ultimate stress $\sigma_{cu}$ [MPa]</td>
<td>24.100</td>
</tr>
</tbody>
</table>

Table 7.4. Parameters of $\mu$ model for Koyna dam concrete

<table>
<thead>
<tr>
<th>Property $E_0$ [GPa]</th>
<th>$\epsilon_{0t}$</th>
<th>$\epsilon_{0c}$</th>
<th>$A_t$</th>
<th>$G_f$ [J/m$^2$]</th>
<th>$A_c$</th>
<th>$B_c$</th>
<th>$k$</th>
</tr>
</thead>
<tbody>
<tr>
<td>31.027</td>
<td>9.34670e-5</td>
<td>4.18989e-4</td>
<td>0.98</td>
<td>400</td>
<td>0.68</td>
<td>475</td>
<td>0.75</td>
</tr>
</tbody>
</table>

density of $\rho_f = 2643$ kg/m$^3$. The acoustic elements of the reservoir have a bulk modulus of $B_w = 2.07$ GPa and density of $\rho_w = 1000$ kg/m$^3$.

The material properties of the Koyna dam concrete have been extensively investigated by previous researchers [48, 115, 357, 240, 14]. In this example, the CDP model (used in the ABAQUS model) and the $\mu$ model (used in the SBFEM model) should be set to have the characteristics reported in Ref. [14], as listed in Table 7.3. The CDP model defines the softening behavior of the concrete by specifying a stress-cracking curve in conjunction with a tensile damage-cracking curve, as shown in Fig. 7.12. This setting results in a fracture energy of $G_f = 400$ J/m$^2$ according to Eq. 7.12. The stiffness degradation damage due to crushing of the concrete is assumed to be zero. A dilation angle of $\psi = 36.3^\circ$ is set to control the plasticity behavior under shearing [18]. The parameters of the $\mu$ model used for the Koyna dam are listed in Table 7.4. For an element that has a characteristic length $l_{ch} = 1$ m, a parameter $B_t = 17622$ is obtained. The stress-strain curves of an element ($l_{ch} = 1$ m) under uniaxial tensile and compressive loadings for the CDP model and $\mu$ model are plotted in Fig. 7.13. They both satisfy the requirements given in Table 7.3. The tensile behavior that governs the concrete damage is quite similar in these two models.

To examine the proposed regularization technique using fracture energy, a mesh sensi-
Figure 7.12. Cracking properties of Koyna dam concrete [14]

Figure 7.13. Stress-strain curves of CDP model and \( \mu \) model for Koyna dam concrete
A sensitivity study is performed on the SBFEM models by modeling the dam domain using three different quadtree meshes, as shown in Fig. 7.14. The meshes of the foundation and reservoir remain the same in these three SBFEM models. The mesh sizes (dam) are 2.20 m, 1.10 m and 0.55 m for Mesh 1, 2 and 3, respectively. They all meet the requirement about characteristic length $l_{ch}$ in Eq. (7.23). Based on the parameters given in Table 7.4 and Eq. (7.31), the characteristic length $l_{ch}$ should range from 0.59 m to 2.95 m, and the element size should range from 0.53 m to 2.64 m.

The horizontal relative displacements between the dam crest and heel obtained from the ABAQUS model and three SBFEM models are compared in Fig. 7.15. It is obvious that the numerical results obtained from three SBFEM models have excellent agreements, however, differ from the result from the ABAQUS model. Generally speaking, the displacements of the SBFEM models have larger amplitudes compared to the ABAQUS model. This difference is caused by the concrete plasticity in the ABAQUS model. Under cyclic loading, the existence of the plasticity leads to the hysteresis behavior of the concrete, i.e., the plastic deformation results in a residual displacement when the loading returns to zero and further reduces the overall response when the loading gets reversed.

The tensile damage contours of the ABAQUS and SBFEM models ($w_1$ in Eq. (7.17a))
are compared in Fig. 7.16. Both the ABAQUS and SBFEM models obtain reasonable results, i.e., the tensile damage distributions are close to the concrete cracks observed in Koyna dam after the earthquake [48]. The cracks are mainly located at the dam neck zone. From the upstream, multiple horizontal cracks are observed, while from the downstream, an inclined crack initiating from the shape corner is obtained. A horizontal crack propagates from the dam heel along the dam base. The lengths of this crack are 17.5 m, 13.2 m, 9.0 m, 8.8 m for the ABAQUS model, SBFEM models with Meshes 1, 2 and 3, respectively. The tensile damage contours of the SBFEM model are similar on different meshes, indicating the proposed regularization technique effectively addresses the mesh sensitivity issue.

### 7.4.3 Post-tensioned Koyna dam

Installing post-tensioned anchors into the non-overflow monoliths of the Koyna dam was one of the repairing techniques [48, 49]. Two rows of post-tensioned anchors passing through the cracks were installed from the dam top down to nearly 21.5 m below the cracks [48], as shown in Fig. 7.17a. They are near and parallel to the upstream and
Figure 7.16. Tensile damage contour comparison of Koyna dam
downstream faces, respectively. Each row contains five anchors in one monolith [49], which results in an anchor spacing of 3.1 m. The anchors were grouted with the structure along the full lengths and locked at the anchor heads. The vertical anchor is assumed to have 65 strands and a borehole with a diameter of 350 mm. The inclined anchor is assumed to have 105 strands and a borehole with a diameter of 500 mm. Each strand has a diameter of 15.2 mm and an MBL of 250 kN. The anchors are post-tensioned to 75% MBL.

Seismic damage analysis is performed on the post-tensioned Koyna dam using the proposed approach. A model of the post-tensioned Koyna dam-reservoir-foundation system is established. The modeling of the reservoir and foundation domains is the same with the example in Section 7.4.2. Figure 7.17b shows the mesh of the post-tensioned Koyna dam, which consists of 1,920 polygonal S-elements (dam) and 217 beam elements (anchors). The element sizes are 0.55 m near the anchor heads and along the anchor layouts, 1.10 m at the dam neck and boundary (upstream face, downstream face, and dam base), and 2.20 m for the remaining area. More detailed modeling considerations, such as
Table 7.5. Parameters of quadrilinear model for anchor-structure grouting interface of Koyna dam [5]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau_u$ [MPa]</td>
<td>8.00</td>
</tr>
<tr>
<td>$\tau_r$ [MPa]</td>
<td>0.80</td>
</tr>
<tr>
<td>$s_1$ [mm]</td>
<td>1.25</td>
</tr>
<tr>
<td>$s_2$ [mm]</td>
<td>2.50</td>
</tr>
<tr>
<td>$s_3$ [mm]</td>
<td>5.00</td>
</tr>
</tbody>
</table>

the PTA modeling, can be found in Section 6.2.

Two models with different bond conditions at the anchor-structure grouting interfaces are established. A model considers the bond-slip interaction by inserting cohesive elements connecting the beam and polygonal S-elements, while a model assumes a perfect bond between the anchor and structure by assigning tie constraint. In the following, they are referred as bond-slip model and perfect bond model, respectively. The constitutive law of the cohesive elements is described by the quadrilinear model (as explained in Section 5.2.2), for which the governing parameters are listed in Table 7.5.

The histories of the dam crest/heel relative displacement and tensile damage contours ($w_t$ in Eq. (7.17a)) obtained from the two post-tensioned Koyna dam models are compared in Figs. 7.18–7.19. The results show negligible difference between these two models, which indicates the anchor-structure grouting is sufficiently strong. Compared to the plain Koyna dam, the displacement response of the anchored Koyna dam is slightly reduced, especially after around 3.90 s. This is because the concrete cracking near the dam neck has been controlled by the PTAs. The tensile damage contours in Fig. 7.19 clearly show that the cracks stop at the anchor configurations.

### 7.5 Conclusion

In this chapter, a Mazar’s concrete damage model ($\mu$ model), allowing stiffness recovery under cyclic loading, is incorporated into the proposed framework for seismic analysis of concrete gravity dams. The quasi-brittle behavior of the concrete can be simulated using this model. To address the mesh dependent issue encountered in the damage analysis, the $\mu$ model is regularized based on the fracture energy by adjusting the softening branch.
Figure 7.18. Post-tensioned Koyna dam crest/heel horizontal relative displacement

Figure 7.19. Tensile damage contour comparison of Post-tensioned Koyna dam models
according to the element size. This regularized concrete damage model is implemented into the polygonal UELs by evaluating the effective damage variable at the scaling center.

Three numerical examples are presented to verify the computational implementation and examine the performance of the $\mu$ model. In the first example, the numerical results of a simple five-node element polygonal S-element under cyclic loading have excellent agreements with the theoretical solutions. The capacity of the $\mu$ model that allows stiffness recovery is clearly demonstrated. In the second example, seismic damage analysis is performed on the Koyna dam subjected to the Koyna earthquake. Similar tensile damage contours are obtained from an ABAQUS model using ABAQUS built-in elements and material and the SBFEM models using the proposed approach (polygonal UEL associated with $\mu$ model). The proposed regularization technique has been proved to be effective by comparing the tensile damage contours using three different meshes. Finally, the seismic damage analysis is performed again on post-tensioned Koyna dam models to show the effect of anchoring on controlling concrete cracking.
Chapter 8

Conclusions and recommendations

8.1 Concluding statements

This thesis presents an efficient numerical framework for seismic analysis of post-tensioned concrete gravity dams. The proposed framework is based on the scaled boundary finite element method (SBFEM), and implemented into commercial FEM software ABAQUS by introducing polygonal and unbounded S-elements as user elements (UEL). The modeling of dam-reservoir-foundation systems can be efficiently established by combing the UELs and ABAQUS built-in elements. The dam and near-field foundation are discretized as quadtree meshes assigned with polygonal UELs. The radiation damping conditions of the far-field foundation can be rigorously captured by an unbounded UEL. The reservoir is modeled by ABAQUS built-in acoustic elements (AC2D4) with a non-reflecting boundary at the far end.

Interfacial problems existing in post-tensioned concrete gravity dams are rigorously simulated in the numerical models, such as the cohesive-frictional behavior at the weak structural interfaces and the bond-slip interaction at the anchor-structure interface. The stability of gravity dams highly relies on the shear resistance at the dam-foundation interface, where the shear strength is smaller compared to the concrete. Besides, the concreting of a gravity dam is usually completed in stages, which may result in weak joints
between different dam lifts. Since the shear resistance of those weak interfaces depends on both friction and cohesion, a cohesive-frictional contact modeling scheme is proposed using ABAQUS contact modeling in conjunction with cohesive elements. This scheme also considers cohesive fracture in the normal direction. A post-tensioned anchor (PTA) is usually grouted with a portion of its length, i.e., bond length. Bond stress occurs at the grouting interface to transfer the post-tension into the structure. It develops with the slippage between the anchor and structure, called bond-slip interaction. Inadequate bond length may lead to the pull-out failure of a PTA system. The bond-slip interaction at the anchor-structure grouting interface is simulated by cohesive elements connecting the anchor and structure.

Automatic mesh-conforming techniques are developed to facilitate the modeling of interfacial problems. These techniques generate node-to-node matching (conforming) meshes at the interfaces. The cohesive elements are created based on matching node pairs, which avoids assigning constraints to connect the cohesive elements with the surrounding components. Moreover, conforming interfacial meshes can improve the accuracy of contact modeling. In particular, this technique automatically inserts PTAs into existing meshes, which makes the mesh generation of the structure to be independent of the anchor layouts. Additional nodes are inserted into existing structural meshes along the anchor layouts since polygonal S-elements can have an arbitrary number of nodes/edges. The inserted nodes are duplicated, and then the duplicated nodes are connected to form beam elements representing anchors. It has significant advantages for the PTA modeling involving multiple inclined anchors.

The quasi-brittle behavior of concrete under tensile loading is simulated using a new Mazar’s concrete damage model ($\mu$ model). This model defines different constitutive behavior under tension and compression and considers the triaxial effect by introducing a triaxial factor. Besides, the stiffness recovery under cyclic loading can be simulated. A regularization technique based on the fracture energy concept is developed for polygonal S-elements to enhance the $\mu$ model to avoid the mesh sensitivity issue in damage analysis.
The developments undertaken in this thesis are summarized as follows.

**ABAQUS implementation of SBFEM**

Polytope (polygonal in 2D and polyhedral in 3D) and two-dimensional unbounded elements derived through the SBFE approach are implemented into ABAQUS as user elements. The availability of polytope UELs significantly alleviates the meshing burden on i) generating conforming interfacial meshes and ii) handling complex geometries given in the form of digital images or stereolithography (STL) files. The flowchart and detailed implementation explanations are presented. A method to create element-based surfaces on the UELs is proposed to exploit the ABAQUS contact modeling capacity.

The implementation is verified through a patch test and a few benchmark tests (modal and transient analyses) comparing with ABAQUS built-in elements. It is proved that the polytope UELs present results accurate up to the machine precision (patch test) and show a slightly faster convergence rate compared to ABAQUS built-in elements. Through a series of numerical examples of interfacial problems benefiting from polytope UELs, the performance of ABAQUS is augmented on conforming meshes. The advantages of polytope UELs offering greater flexibility on mesh generation are demonstrated through two examples using octree meshes. A numerical example of an elastic half space subjected to a pressure impulse clearly shows that the unbounded UEL can rigorously capture the dynamic properties of unbounded media using a small truncation area.

**Modeling of interfacial problems**

Based on the polygonal UELs, automatic mesh-forming techniques are developed to convert non-conforming meshes to conforming ones at the interfaces. Surface conforming meshes (both matching nodes and edges) are obtained by shifting/inserting nodes and spiting edges of polygonal UELs at the weak structural interfaces. Nodal conforming meshes (matching nodes along desired lines) are generated by inserting additional nodes into polygonal UELs along the anchor layouts and duplicating the inserted nodes.
A cohesive-frictional contact modeling scheme is proposed to simulate the weak structural interfaces by combining the ABAQUS contact modeling using the kinetic friction model and the cohesive elements. Cohesive elements employing a quadrilinear constitutive law is used to simulate the bond-slip interaction at the anchor-structure grouting interface. For both two types of interfacial problems, the cohesive elements are generated based on matching node pairs at the interfaces.

Through a direct shear test on two contact blocks, the proposed cohesive-frictional contact modeling scheme shows a reasonable shear stress-slippage curve featuring a peak shear strength and a residual shear strength. A pull-out simulation of a single embedded anchor clearly demonstrates the progress of the pull-out failure: i) pseudo-elastic behavior at small deformations, ii) de-bonding development at moderate displacements, and iii) residual behavior at larger displacements. The shear stress distributions along the bond length under the initial, progressive and ultimate loading conditions present the same characteristics reported in the literature.

**Seismic analysis of post-tensioned concrete gravity dams**

The seismic analysis of post-tensioned concrete gravity dams is performed on the ABAQUS platform using the implemented polygonal and unbounded UELs. Quadtree meshes providing fast and smooth transition elements are directly utilized to discretize the structural domains in the dam-reservoir-foundation interaction systems. The unbounded foundation is represented by constructing an unbounded UEL at the boundary of the near-field foundation. ABAQUS built-in acoustic elements (AC2D4) are used to model the reservoir, and a non-reflecting boundary embedded in ABAQUS is set at the far end of the reservoir. The proposed cohesive-frictional contact modeling and automatic PTA modeling schemes are adopted in the post-tensioned concrete gravity dam models.

The accuracy of the proposed approach is verified by performing a seismic analysis of a post-tensioned gravity dam with simple geometry, and comparing the results to a model using ABAQUS’ built-in elements. The advantages of this method in handling complex
problems are demonstrated through a dam with three lift joints anchored by multiple inclined anchors.

**Seismic damage analysis of post-tensioned concrete gravity dams**

A regularized Mazar’s concrete damage model (µ model) is incorporated into the proposed framework to consider the quasi-brittle behavior of concrete in tension. The µ model is suitable for seismic analysis since it consistently considers the effects of crack opening and closure on the stiffness of the material, i.e., degradation and recovery. This damage model is regularized by adjusting its softening branch (tension) according to the element size to guarantee specific fracture energy.

Through a numerical example of a simple five-node element polygonal UEL under cyclic loading, the proposed damage model is verified by comparing the numerical results with the theoretical solutions. The effectiveness of this model on simulating stiffness degradation and recovery is demonstrated. Seismic analysis of Koyna dam (plain) subjected to Koyna earthquake is performed to validate this damage model. Similar tensile damage contours have been obtained on the models using the proposed approach and an ABAQUS model using built-in concrete damaged plasticity model, and they coincide with the cracks observed in the Koyna dam after the earthquake. The proposed damage model is shown to be mesh independent through three models with different meshes. The same seismic damage analysis is performed on a post-tensioned Koyna dam model, and it is shown that the PTAs are efficient in controlling the cracks.

### 8.2 Recommendations for future work

**Arbitrary seismic input**

Vertically propagated shear wave is used as the seismic input in this thesis. In reality, seismic waves propagate in the foundation in terms of both compressional and shear waves,
and they can travel into the near-field foundation with an arbitrary direction. To make the proposed framework more realistic, seismic input models related to general wave propagation, i.e., compressional and shear waves with an arbitrary incident angle, are recommended to be developed. The current research has distinguished the free-field motion (wave propagation) from the scatter motion. Consequently, introducing new seismic input models can be conveniently realized by modifying the implementations related to the free-field motion only, such as the displacements, velocities, and surface traction at the near-field/far-field interface [166].

**Non-linear/anisotropic/non-homogeneous foundation**

In this thesis, the foundation is assumed to be a linear isotropic homogeneous medium. However, because of the natural sedimentation process, soils generally exhibit anisotropic and non-homogeneous behavior. For a rock foundation, there might be faults and other weak planes, which again makes the foundation to be a non-homogeneous medium. Besides, the rock mass grouted with ground anchors might experience shear or uplift failures [2]. The shear and/or tensile strength of the rock mass should be considered in the numerical simulation to examine the design of ground anchors against rock mass failure by uplift.

**Reservoir boundary absorption**

The effects of the reservoir boundary absorption on the hydrodynamic pressure and seismic response of post-tensioned gravity dams are neglected in this work. Studies have shown that a non-absorptive boundary may result in resonant hydrodynamic responses [359, 343]. Therefore, the hydrodynamic pressure might be overestimated in the seismic analysis of post-tensioned gravity dams, especially for dams subjected to vertical ground motions [359]. An interface element that introduces a wave reflection coefficient $\alpha$ at the reservoir boundary is suggested to be developed.
Three-dimensional seismic analysis of dams

The proposed framework can be extended to three-dimensional cases, such as post-tensioned arch dams and post-tensioned buttress dams. Seismic analysis of dams in this thesis is limited to two-dimensional cases, and only two-dimensional unbounded element is implemented into ABAQUS as user elements. However, three-dimensional finite element analysis is required for dams with high importance [20]. In Chapter 4, three-dimensional polytope (polyhedral) elements have been implemented into ABAQUS. The PTAs can be automatically embedded into existing polyhedral meshes as well, by inserting additional nodes on the surfaces of polyhedral UELs [12]. The implementations of three-dimensional unbounded element and its associated seismic input model [360, 342] need to be developed.

Simulation of randomly spread fiber-reinforced composite

The automatic anchor modeling technique can be extended to model composite materials with randomly spread fibers. The advantages for this technique on inserting multiple inclined anchors have been demonstrated through the example in Section 6.3.2. No special treatment, such as partitioning at the anchor/fiber layouts, is required to generate the mesh of the dam/matrix. Using a similar technique, the discrete modeling of composite materials with randomly spread fibers has been done in Ref. [12]. However, in that work, truss elements representing fibers are formed by directly connecting the inserted nodes, which essentially assumes a perfect bond between the fibers and matrix. The proposed anchor modeling technique can duplicate the inserted nodes and establish interactions between the fibers and matrix.
Bibliography


[71] L. Liu, J. Zhang, C. Song, C. Birk, W. Gao. An automatic approach for the acoustic analysis of three-dimensional bounded and unbounded domains by scaled bound-


[88] W. Xing, J. Zhang, C. Song, F. Tin-Loi. A node-to-node scheme for three-
dimensional contact problems using the scaled boundary finite element method.

[89] S. Ya, S. Eisenträger, C. Song, J. Li. An open-source abaqus implementation
of the scaled boundary finite element method to study interfacial problems using


interfaces and joints. *International journal for numerical and analytical methods

of soil-reinforcement interaction. *Computers and Geotechnics*, 7(1-2):133–151,
1989.

Kwiecień 2018.


[96] H. M. Hilber, T. J. R. Hughes, R. L. Taylor. Improved numerical dissipation for
time integration algorithms in structural dynamics. *Earthquake Engineering &


