Anura3D MPM Research Community

Anura3D MPM Software

Tutorial Manual

Version: 2025
January 2025

Anura3D MPM Software, Tutorial Manual

With the contributions of:

Amine Aboufirass	Deltares Delft, The Netherlands
Abdelrahman Alsardi	Virginia Tech, United States
Luis Angel Aviles	Universitat Politècnica de Catalunya, Barcelona, Spain
Francesca Ceccato	Università degli Studi di Padova, Italy
Alexander Chmelnizkij	Technische Universität Hamburg, Germany
Gaia Di Carluccio	Universitat Politècnica de Catalunya, Barcelona, Spain
James Fern	Amberg Engineering Ltd / EPFL, Switzerland
Vahid Galavi	Deltares Delft, The Netherlands
Veronica Girardi	Università degli Studi di Padova, Italy
Mateu Maglia	Universitat Politècnica de Catalunya, Barcelona, Spain
Pietro Marveggio	Politecnico di Milano, Italy
Mario Martinelli	Deltares Delft, The Netherlands
Miriam Mieremet	Deltares Delft, The Netherlands
Francesc Mirada	Universitat Politècnica de Catalunya, Barcelona, Spain
Jonathan Moore	Virginia Tech, United States
John W. Murphy	University of California Berkeley, United States
Yipin Peng	Pennsylvania State University, United States
Núria Pinyol	Universitat Politècnica de Catalunya, Barcelona, Spain
Grace Roberts	University of Cambridge, United Kingdom
Alexander Rohe	Deltares Delft, The Netherlands
Angela Saade	Virginia Tech, United States
Marc Stapelfeldt	Technische Universität Hamburg, Germany
Alba Yerro	Virginia Tech, United States
Kaleigh Yost	Pennsylvania State University, United States
Luis E. Zambrano-Cruzatty	University of Maine, United States
Tong Zhang	Universitat Politècnica de Catalunya, Barcelona, Spain
Bruno Zuada	Deltares Delft, The Netherlands

Many people contributed to the creation of this tutorial. If you feel that you are one of them but your name does not appear in this list, please contact us.

Published and printed by:
Anura3D MPM Research Community
c/o Stichting Deltares
Boussinesqweg 1
2629 HV Delft (The Netherlands)

Contact and information:
www.Anura3D.com

Contents

1 Introduction **5**
1.1 Large deformations and soil–water–structure interaction . . . 5
1.2 Anura3D MPM Research Community . . . 5
1.3 About Anura3D 2025 . . . 6

2 Getting started with Anura3D **9**
2.1 Calculation process with Anura3D . . . 9
2.1.1 Creating input data . . . 9
2.1.2 Performing calculation . . . 10
2.1.3 Visualisation of results . . . 10
2.2 Downloading Anura3D source code . . . 12
2.3 Compiling Anura3D . . . 12
2.3.1 Prerequisites . . . 12
2.3.2 Compiling Anura3D . . . 13
2.4 Installation of GiD (pre-processor) . . . 16
2.4.1 Installation instructions . . . 16
2.4.2 The Anura3D 2025 problemtype . . . 17
2.5 Installation of ParaView (post-processor) . . . 19

3 One-dimensional consolidation (oedometric compression test) **23**
3.1 Problem definition in 2D and 3D . . . 23
3.2 Creating input data in GiD . . . 24
3.2.1 Starting pre-processor and selecting problem type . . . 25
3.2.2 Defining geometry in 2D and 3D . . . 27
3.2.3 General data . . . 34
3.2.4 Defining calculation stages . . . 34
3.2.5 Defining material properties . . . 35
3.2.6 Specifying material points in 2D and 3D . . . 37
3.2.7 Defining boundary conditions in 2D and 3D . . . 39
3.2.8 Creating mesh in 2D and 3D . . . 46
3.2.9 Calculation data . . . 51
3.2.10 Results output . . . 53
3.3 Performing calculation in Anura3D . . . 54
3.4 Visualisation of results in ParaView . . . 55
3.4.1 Opening ParaView . . . 56
3.4.2 Loading the VTK output files . . . 56
3.4.3 Showing the mesh in 2D and 3D . . . 58
3.4.4 Change view settings to parallel projection (for 3D case only) . . . 59
3.4.5 Visualisation of liquid pressure in 2D and 3D . . . 60
3.4.6 Visualisation of soil displacement in 2D and 3D . . . 64
3.4.7 Determining the material point ID . . . 66
3.4.8 Plotting time graphs for material points . . . 69
3.5 Visualization of results in GiD . . . 73
3.5.1 Opening GiD post-processor . . . 73
3.5.2 Loading results in GiD post-processor from Binary files . . . 74
3.5.3 Mesh and MPs visualization . . . 75
3.5.4 Change view settings . . . 76
3.5.5 Visualisation of liquid pressure in 2D and 3D . . . 76
3.5.6 Determining the material point ID . . . 78

3.5.7 Plotting time graphs for material points . . . 79
3.6 Visualisation of text output files . . . 80
3.7 Variations . . . 81
3.7.1 Variation: Effect of local damping . . . 82
3.7.2 Variation: Small deformation problem . . . 84
3.8 Discussion . . . 85

4 Wave propagation **87**
4.1 Problem definition . . . 87
4.2 Creating input data . . . 87
4.2.1 Starting pre-processor . . . 87
4.2.2 Defining geometry . . . 87
4.2.3 Defining boundary conditions . . . 88
4.2.4 Defining loading conditions . . . 89
4.2.5 Creating mesh . . . 90
4.3 Calculation data . . . 91
4.4 Generating input data and performing calculation . . . 91
4.5 Visualisation of results . . . 92
4.6 Variations: Bulk viscosity . . . 92
4.7 Variations: Absorbing Bounadries . . . 94
4.8 Variations: Coupled wave propagation . . . 97

5 Triaxial compression test **103**
5.1 Problem definition . . . 103
5.2 Creating input data . . . 104
5.2.1 Starting pre-processor . . . 104
5.2.2 Defining geometry . . . 104
5.2.3 Defining material properties . . . 105
5.2.4 Specifying material points . . . 105
5.2.5 Defining initial conditions . . . 106
5.2.6 Defining boundary conditions . . . 107
5.2.7 Creating mesh . . . 108
5.2.8 Initialization of calculation data . . . 109
5.2.9 Generating input files . . . 111
5.3 Performing calculation in Anura3D . . . 111
5.4 Visualisation of text output files . . . 111
5.5 External Soil Model . . . 113
5.5.1 Preparation of the FORTRAN File . . . 113
5.5.2 Include the Module into the code . . . 117
5.5.3 Definition of the pointer to the ESM in "ReadMaterialData.for" . . . 118
5.5.4 Definition of a External Soil Model in Gid . . . 118

6 Sliding blocks **121**
6.1 Problem definition . . . 121
6.1.1 Starting pre-processor . . . 122
6.1.2 Defining geometry . . . 122
6.1.3 Defining general data . . . 127
6.1.4 Defining calculation stages . . . 127
6.1.4.1 Stage 0: Gravity loading . . . 127
6.1.4.2 Stage 1: Sliding of the blocks . . . 133
6.1.5 Creating mesh . . . 140

6.2 Performing calculation in Anura3D . . . 142
6.3 Visualization of results in ParaView . . . 143
6.4 Variations . . . 146
6.5 Discussion . . . 146

7 Column collapse 147
7.1 Problem description . . . 147
7.2 Creating input data in GiD . . . 148
7.2.1 Starting pre-processor and selecting problemtype . . . 148
7.2.2 Defining geometry . . . 149
7.2.3 Defining calculation stages . . . 149
7.2.4 Definition of the first stage (gravity loading) . . . 150
7.2.4.1 Defining material properties . . . 150
7.2.4.2 Specifying material points . . . 151
7.2.4.3 Defining boundary conditions . . . 151
7.2.4.4 Creating mesh . . . 152
7.2.4.5 Definition of calculation data . . . 154
7.2.4.6 Define results output . . . 156
7.2.5 Definition of the second stage (column collapse) . . . 156
7.3 Performing calculation in Anura3D . . . 158
7.4 Visualisation of results in ParaView . . . 158
7.4.1 Gravity loading . . . 159
7.4.2 Column collapse . . . 160
7.4.3 Visualization of results of text files . . . 161

8 Shallow foundation 163
8.1 Problem definition . . . 163
8.2 Creating input data . . . 165
8.2.1 Starting pre-processor . . . 165
8.2.2 Defining units and dimensions . . . 165
8.2.3 Defining geometry . . . 165
8.2.4 Defining material properties . . . 166
8.2.5 Specifying material points . . . 168
8.2.6 Defining initial conditions . . . 168
8.2.7 Defining boundary conditions . . . 168
8.2.8 Defining a surface for computation of reaction forces . . . 172
8.2.9 Defining the moving mesh . . . 172
8.2.10 Creating mesh . . . 176
8.3 Approach 1: External load A and use of load multipliers . . . 176
8.3.1 Initialisation of calculation data . . . 176
8.3.2 Generating Input Files . . . 178
8.3.3 Performing calculation in Anura3D . . . 178
8.3.4 Visualisation of results in ParaView . . . 179
8.3.5 Visualisation of text output files . . . 179
8.3.6 Approach 1 - Variation: add load system B to simulate surcharge on soil 183
8.4 Approach 2: Prescribed velocity . . . 184
8.4.1 Initialisation of calculation data . . . 185
8.4.2 Generating input files . . . 186
8.4.3 Performing calculation in Anura3D . . . 186
8.4.4 Visualisation of results . . . 187
8.5 Variations . . . 188

9 Impact problem **189**
9.1 Problem definition . . . 190
9.2 Creating input data . . . 190
9.2.1 Starting pre-processor . . . 191
9.2.2 Defining geometry . . . 191
9.2.3 Defining material properties . . . 191
9.2.4 Specifying material points and assigning materials . . . 192
9.2.5 Defining initial conditions . . . 192
9.2.6 Defining boundary conditions . . . 194
9.2.7 Defining initial velocity on material points . . . 195
9.2.8 Defining a surface to record reaction forces . . . 195
9.2.9 Defining the moving mesh . . . 196
9.2.10 Creating mesh . . . 197
9.2.11 Calculation parameters . . . 197
9.2.12 Performing calculation in Anura3D . . . 199
9.2.13 Visualisation of results in ParaView . . . 200
9.2.14 Visualisation of text output files . . . 201
9.3 Variation: Simulation of the impacting body as a rigid body . . . 202
9.3.1 Starting pre-processor . . . 202
9.3.2 Defining geometry . . . 202
9.3.3 Defining material properties . . . 202
9.3.4 Specifying material points . . . 203
9.3.5 Defining boundary conditions . . . 203
9.3.6 Defining initial velocity on material points . . . 204
9.3.7 Defining contact properties . . . 204
9.3.8 Defining a surface for computation of reaction forces . . . 204
9.3.9 Defining the moving mesh . . . 204
9.3.10 Creating the mesh . . . 205
9.3.11 Calculation parameters . . . 205
9.3.12 Performing calculation in Anura3D . . . 205
9.3.13 Visualisation of results in ParaView . . . 205
9.3.14 Visualisation of text output files . . . 205
9.4 Variation: Simulation in 3D cylindrical axisymmetric conditions . . . 205
9.4.1 Starting pre-processor . . . 205
9.4.2 Defining geometry . . . 206
9.4.3 Defining problem dimensions . . . 207
9.4.4 Defining material properties . . . 208
9.4.5 Specifying material points . . . 208
9.4.6 Defining boundary conditions . . . 208
9.4.7 Defining initial velocity on material points . . . 210
9.4.8 Defining a surface for computation of reaction forces . . . 210
9.4.9 Defining the moving mesh . . . 210
9.4.10 Creating mesh . . . 210
9.4.11 Calculation parameters . . . 211
9.4.12 Performing calculation in Anura3D . . . 211
9.4.13 Visualisation of results in ParaView . . . 211
9.5 Other variations . . . 212

10 Excavation **213**
10.1 3D Column excavation . . . 213
10.1.1 Problem definition . . . 213

10.1.2 Creating input data . . . 214
10.1.2.1 Starting pre-processor . . . 214
10.1.2.2 Defining geometry . . . 214
10.1.2.3 Defining material properties . . . 216
10.1.2.4 Specifying material points . . . 217
10.1.2.5 Defining boundary conditions . . . 218
10.1.2.6 Creating mesh . . . 219
10.1.2.7 Defining excavation steps . . . 219
10.1.2.8 Initialisation of calculation data . . . 221
10.1.2.9 Stage0: Gravity loading . . . 221
10.1.2.10 Stage1: Excavation . . . 222
10.1.3 Performing calculation in Anura3D . . . 222
10.1.4 Visualisation of results in ParaView . . . 222
10.1.5 Visualisation of text output files . . . 224
10.1.6 Excavation with Mohr-Coulomb material . . . 226
10.2 2D Strutted excavation . . . 230
10.2.1 Problem definition . . . 230
10.2.2 Creating input data . . . 230
10.2.2.1 Starting pre-processor . . . 230
10.2.2.2 Defining geometry . . . 231
10.2.2.3 Defining material properties . . . 232
10.2.2.4 Specifying material points . . . 233
10.2.2.5 Defining boundary conditions . . . 234
10.2.2.6 Creating mesh . . . 234
10.2.2.7 Defining excavation steps . . . 234
10.2.2.8 Initialisation of calculation data . . . 236
10.2.2.9 Stage0: Gravity loading . . . 236
10.2.2.10 Stage1: Excavation . . . 237
10.2.3 Performing calculation in Anura3D . . . 237
10.2.4 Visualisation of results in ParaView . . . 238

11 Construction **241**
11.1 2D Column construction . . . 241
11.1.1 Problem definition . . . 241
11.1.2 Creating input data . . . 242
11.1.2.1 Starting pre-processor . . . 242
11.1.2.2 Defining geometry . . . 242
11.1.2.3 Defining material properties . . . 243
11.1.2.4 Specifying material points . . . 244
11.1.2.5 Defining boundary conditions . . . 244
11.1.2.6 Defining construction steps . . . 245
11.1.2.7 Creating mesh . . . 247
11.1.2.8 Initialisation of calculation data . . . 247
11.1.2.9 Generating input files . . . 248
11.1.3 Performing calculation in Anura3D . . . 248
11.1.4 Visualisation of results in ParaView . . . 248
11.1.5 Visualisation of text output files . . . 249
11.1.6 Variation: filling empty elements during construction . . . 250
11.1.6.1 Visualisation of results in ParaView . . . 252

12 Submerged slope collapse **253**

12.1 Problem definition . . . 253
12.2 Creating input data . . . 253
12.2.1 Starting pre-processor . . . 254
12.2.2 Defining geometry . . . 254
12.2.3 Defining general data . . . 255
12.2.4 Defining calculation stages . . . 256
12.2.4.1 Stage 0: Gravity loading . . . 256
12.2.4.2 Stage 1: Slope failure . . . 260
12.2.4.3 Stage 2: Slope stabilization . . . 263
12.2.5 Creating mesh . . . 265
12.3 Performing calculation in Anura3D . . . 266
12.4 Results . . . 266
12.4.1 Stage 0: Gravity Loading . . . 267
12.4.2 Stage 1: Slope Failure . . . 267
12.4.3 Step 2: Slope Stabilization . . . 269

13 Slope failure under transient hydraulic loads 271
13.1 Problem definition . . . 271
13.2 Rapid Drawdown . . . 272
13.2.1 Creating input data . . . 272
13.2.1.1 Starting pre-processor . . . 272
13.2.1.2 Defining general data . . . 272
13.2.1.3 Defining geometry . . . 273
13.2.1.4 Defining stages . . . 274
13.2.1.5 Creating mesh . . . 285
13.2.2 Performing calculation in Anura3D . . . 286
13.2.3 Visualisation of results in ParaView . . . 286
13.2.4 Variation on initial conditions: maximum suction threshold . . . 288
13.3 Rainfall infiltration . . . 288
13.3.1 Creating input data . . . 289
13.3.1.1 Starting pre-processor . . . 289
13.3.1.2 Defining general data . . . 289
13.3.1.3 Defining geometry . . . 289
13.3.1.4 Defining calculation stages . . . 290
13.3.1.5 Creating mesh . . . 301
13.3.2 Performing calculation in Anura3D . . . 302
13.3.3 Visualisation of results in ParaView . . . 303
13.3.4 Variation on triggering action: application of zero suction . . . 304
13.4 Stress initialization from file . . . 304
13.4.1 Creating input data . . . 305
13.4.1.1 Preparation of input file for stress initialization . . . 305
13.4.1.2 Defining calculation stages . . . 306
13.4.1.3 Defining calculation data . . . 307
13.4.2 Performing calculation in Anura3D . . . 307
13.4.3 Visualisation of results in ParaView . . . 308

References 310

A Shallow foundation in 3D-cylindrical coordinates 313
A.1 Creating input data . . . 313
A.1.1 Defining problem dimensions . . . 313

A.1.2 Defining geometry 313
A.1.3 Defining material properties 316
A.1.4 Specifying material points 316
A.1.5 Defining boundary conditions 317
A.1.6 Defining a surface for computation of reaction forces 319
A.1.7 Defining the moving mesh 319
A.1.8 Creating mesh 322
A.1.9 Generating Input Files 322
A.1.10 Performing calculation in Anura3D 322
A.1.11 Visualisation of results in ParaView 323

Anura3D - is a software for the numerical modelling and simulation of large deformations and soil–water–structure interaction using the material point method (MPM). Copyright (C) 2020 Members of the Anura3D MPM Research Community.

Anura3D is free software: you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation <https://www.gnu.org/licenses/>, either version 3 of the License, or (at your option) any later version.

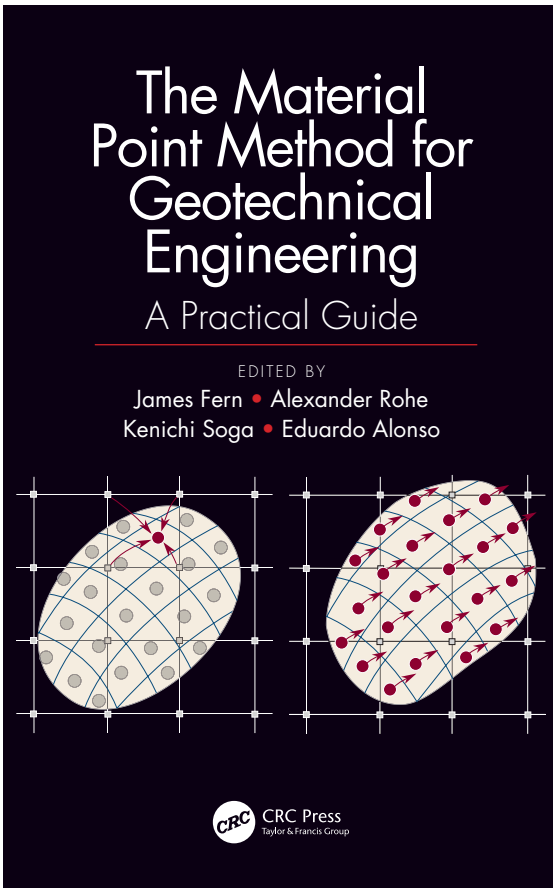
Anura3D documentation is furnished under the License and may be used only in accordance with the terms of such license. It is advised to consult the manuals before applying the software.

Anura3D is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details.

Anura3D MPM Research Community
c/o Stichting Deltares
Boussinesqweg 1
2629 HV Delft (The Netherlands)

Contact and information
e-mail: info@Anura3D.com
web: www.Anura3D.com

Complementary reading: [1] Fern E.J., Rohe A., Soga K., Alonso E.E. (2019) The Material Point Method for Geotechnical Engineering – A Practical Guide. CRC Press, London.



1 Introduction

Authors: Alba Yerro; Alexander Rohe; James Fern; Francesca Ceccato; Veronica Girardi; Kaleigh Yost

1.1 Large deformations and soil–water–structure interaction

Interactions between soil, water, and structures play an important role in geotechnical, environmental and civil engineering problems, such as landslides induced by seepage and rain, liquefaction or fluidisation, and sedimentation processes in submerged slopes, internal erosion in dykes, scouring around offshore structures, consolidation-induced settlement or pile installation in saturated soils. In many cases, the material involved can experience large deformations, which can lead to dramatic events. The numerical modelling of these processes is challenging because the treatment of soil-water coupling, soil-structure interaction, and large deformations is not straightforward.

MPM is a point-based numerical method capable of modelling large deformations, which has been extended and implemented in the Anura3D software to cope with advanced soil-water-structure interaction problems. This tutorial manual provides instructions on how to perform numerical simulations with Anura3D . More detailed information on the material point method and its applications is given by Fern et al. 2019 [1]. A list of publications and the Anura3D Scientific Manual are also good references available on the Anura3D website (www.anura3D.com).

1.2 Anura3D MPM Research Community

The Anura3D Software is being developed by the Anura3D MPM Research Community. In this international collaboration the complementary expertise of different groups that carry out research on numerical modelling of large deformations and soil–water–structure interaction is brought together. For more information visit the website www.anura3d.com of the Anura3D MPM Research Community.

The Anura3D MPM Research Community is collaborating with CIMNE (International Center for Numerical Methods in Engineering), who is granting free temporal licences of GiD for the academic purposes of the Anura3D MPM Research Community. The support of CIMNE is greatly acknowledged. The Anura3D MPM Research Community uses GiD to develop the Anura3D "problemtype". GiD may also be used as a pre-processing software for the preparation of the required files to perform calculations with Anura3D . However, the interface between Anura3D and the pre- and post-processing software is completely open and the user may choose other software.

1.3 About Anura3D 2025

Anura3D is a research-oriented software developed by the Anura3D MPM Research Community (Sec. 1.2). The code has been validated and tested with a limited number of problems. The Anura3D MPM Research Community shall not be responsible for losses of any kind, resulting from the use of this program. The Anura3D is a free open source software under the GNU Lesser General Public License version 3.

The most relevant publications related to the development and application of the Anura3D software are available online where direct links to the respective journals or theses for download are provided. The development of the current code started in 2008 and is ongoing.

The features implemented in the open source are briefly summarized in Table 1.1. While some of them have been tested, others are still under developemnt. The examples provided in this tutorial provide guidance to the user through the available tested features.

The current state of the Anura3D has the following identified limitations concerning the tested features:

- 3D-Cylindrical: y-axis is the axis of symmetry, gravity can only be applied in y-axis direction.
- Multiphase formulations: a combination of different material types can cause problems for certain combinations (undrained total stress and undrained effective stress), no water flow is transferred between saturated and dry materials.
- Contact algorithm: the maximum number of contact materials is four (4), the maximum number of master materials is one (1). Contact is fixed on the mesh nodes.
- Moving mesh: works only for prismatic bodies in 3D and trapezium areas in 2D, the moving mesh direction is constrained only in one direction. the moving mesh needs an extension and/or compression mesh.
- Excavation: limited to 30 excavation stages.
- Rigid body: can be applied only to one body in the system, only works together with the contact algorithm, only moves in one direction, it can't rotate.
- Fixities and traction boundary conditions: only aligned with the coordinate axes
- K_0 stress initialization: one homogeneous material (otherwise, use gravity together with local damping and stress initialization with quasi-static convergence criteria).
- Infiltration/evaporation boundary condition: only one value of infiltration rate can be applied, it is not possible to assign different values at different part of the domain.
  If two-phase single point unsaturated formulation is used, K_0 initialization works if a phreatic surface and a soil surface are specified using options available in Anura3D_2025 Problem Type. Phreatic surface and soil surface are assigned to lines, thus are only limited to 2D geometries.

The user should also take into account the following warnings:

- Multiple materials: the use of several drainage material types on the same model is not recommended.
- Absorbing boundaries: do not prevent material points from leaving the mesh.

Table 1.1: *List of features available in Anura3D 2025*

Name of the feature	Under development	Tested
Geometrical dimensions		
2D - plane strain		X
2D - axisymmetric	X	
3D - cartesian		X
3D - cylindrical		X
Computational methods		
MPM-MP		X
MPM-MIXED		X
UL-FEM		X
FEM		X
Integration schemes		
Explicit		X
Implicit	X	
Multiphase formulations		
1-phase (dry material, undrained, drained)		X
2-phase single-point for saturated soil		X
2-phase single-point for unsaturated soil		X
2-phase double-point	X	
3-phase single-point	X	
Constitutive models		
Linear elastic		X
Mohr-Coulomb		X
Bingham fluid		X
External soil model (user defined)		X
Other features		
Contact algorithm		X
Moving mesh		X
Excavation		X
Construction		X
Rigid body		X
Bulk viscosity		X
Submerged analysis		X
Output of reaction forces at surfaces		X
Boundary and initial conditions		
Traction/pressure		X
Infiltration & evaporation		X
Total hydraulic head		X
Seepage face		X
Absorbing boundary		X
Remove fixity		X
Prescribed velocity		X
Initial velocity on material points		X
K_0 stress initialization (including phreatic surface)		X
Stress initialization with quasi-static convergence		X
Stress initialization from file		X

2 Getting started with Anura3D

Authors: Alba Yerro; Alexander Rohe; James Fern; Francesca Ceccato; Luis E. Zambrano-Cruzatty; Kaleigh Yost

Anura3D is a software tool for numerical analysis of geotechnical problems developed by the Anura3D MPM Research Community and distributed as open-source code programmed using Fortran. This software is a 2D/3D implementation of the material point method (MPM) and it is used for simulating the physics involved in soil-water-structure interaction and large deformation problems. There is no warranty or liability in any sense for the use of the provided software or results (see "Copyright and Liability Anura3D" at the beginning of this document). The list of implemented and under development features is provided in Table 1.1.

This chapter explains how to get started with Anura3D. The basis to perform a numerical simulation is provided in Section 2.1. Instructions for downloading Anura3D are given in Section 2.2. Then, the process to compile the code is explained in Section 2.3. Subsequently, the installation instructions for both pre- and post-processing software are presented in Sections 2.4 and 2.5. In both cases, the use of third-party software is required.

2.1 Calculation process with Anura3D

The process to perform a numerical simulation consists of three parts as follows.

- ◇ Creating input data with the pre-processor (e.g., GiD software)
- ◇ Performing calculation with the Anura3D software (compiled executable)
- ◇ Visualisation of results with the post-processor (e.g., ParaView software)

In this tutorial manual, we will use GiD as the pre-processor and ParaView as the post-processor, though other pre- and post-processors may be used (the interface between Anura3D and the pre- and post-processing software is completely open). A schematic description of the calculation procedure is presented in Figure 2.1, and each step is explained below in this chapter.

Figure 2.1: *Procedure for performing a numerical simulation with Anura3D including pre- and post-processing.*

2.1.1 Creating input data

Anura3D requires three input files – the *GOM* file, the *CPS* file, and the *OPD* file. All files can be generated with the GiD pre-processing software (the installation instructions of GiD are given in Section 2.4). All files are text files (ASCII), which can be modified manually by opening them in a text editor and changing the text. This is typically done for parametric analyses for which the value of a specific parameter is changed. Users could create their own *GOM*, *CPS*, or *OPD* files with alternative pre-processors (different than GiD) if desired.

Definition of problem: *GOM* file
The *GOM* file describes the geometry of the problem, the material data sets, initial and boundary conditions, loading conditions, excavation phases, contact properties, number of material points, and computational mesh. When a model is created, GiD writes this information into the *GOM* file.

Definition of calculation data: *CPS* file
The *CPS* file describes the calculation data such as the computation method, calculation time step data, gravity data, load multipliers, quasi-static convergence, damping, strain smoothing, output data, etc. Unlike the *GOM* file, a *CPS* file is required for each calculations phase. GiD is used to define the initial *CPS* file required to initiate the calculation (i.e., *CPS*_001 file), whilst the subsequent CPS files are generated at the end of each calculation phase.

Definition of desired output data: *OPD* file
The user selects the desired frequency for writing the output data, as well as which variables should be included in the output files, in order to reduce the size of the output files. These parameters are set in the *OPD* file which is generated by GiD when a model is created.

2.1.2 Performing calculation

In Anura3D 2025 , users can start the calculation from the GiD interface directly by navigating to <Anura3D> in the upper toolbar and clicking on <Calculate>.

Note that previous versions of Anura3D required the program to be executed from the Windows command line or by running the `calculate.bat` file in the project folder `projectname.A3D`.

2.1.3 Visualisation of results

Anura3D creates results by writing several output files (Figure 2.1). These files have to be treated differently depending on the file type.

VTK files
Anura3D stores most of the output information in a set of files with *VTK* extension. The ParaView post-processing software is recommended to visualise the results stored in those output files (the installation instructions of GiD are given in Section 2.5). There are four different types of *VTK* files depending on the type of data stored in them, and are as follows.

- ◇ `projectname_MeshData_SSSTTTTTT.VTK`
- ◇ `projectname_MPScalar_SSSTTTTTT.VTK`
- ◇ `projectname_MPVector_SSSTTTTTT.VTK`
- ◇ `projectname_MPTensor_SSSTTTTTT.VTK`

where `projectname` is the user-defined name of the project and `SSS` and `TTTTTT` are the automatically assigned calculation step and time step numbers, respectively. The variables of the VTK files are listed in Figure 2.2.

VTK files	Variables	Symbol	Dimension
projectname_MeshData_SSSTTTTTT.VTK	Geometry information		
projectname_MPScalar_SSSTTTTTT.VTK	deviatory_strain_solid	ε_q	
	deviatory_stress	q	
	incremental_deviatoric_strain_solid	$\Delta\varepsilon_q$	
	incremental_volumetric_strain_solid	$\Delta\varepsilon_{vol}$	
	mean_effective_stress	p	
	porosity	n	scalar
	pressure_liquid	σ_L	
	volumetric_strain_liquid	ε_{vol_L}	
	volumetric_strain_solid	ε_{vol}	
	weight_liquid	W_L	
	weight_solid	W_S	
projectname_MPVector_SSSTTTTTT.VTK	acceleration_solid	a_s	
	displacement_liquid	x_L	
	displacement_solid	x_S	vector (3)
	velocity_liquid	v_L	
	velocity_solid	v_S	
projectname_MPTensor_SSSTTTTTT.VTK	eff_stress_solid	σ'_S	
	strain	ε	tensor (3x3)
	stress_liquid	σ_L	

Figure 2.2: *Scalar, vector and tensor components in the VTK files.*

The mesh data, such as the mesh location and active elements, is stored in `MeshData`. The material point data is stored in:

- `MPScalar`: collects all scalar quantities such as material point number, pore water pressure, mean and deviatoric stress, etc.
- `MPVector`: collects all vector quantities such as velocity, displacement, etc. The components of the vector are

  $$(X, Y, Z)$$

  in global coordinate system. For 2D analysis the third component (Z) is zero.
- `MPTensor`: collects tensorial quantities such as stress and strain. The tensor has 9 components, that ParaView enumerate from 0 to 8 starting from the top left corner of the matrix, as indicated in Figure 2.3.

$$\begin{array}{ccc} \textit{Soil mechanics} & & \textit{Paraview equivalent} \\ \boldsymbol{\sigma} = \begin{pmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_{zz} \end{pmatrix} & = & \begin{pmatrix} 0 & 1 & 2 \\ 3 & 4 & 5 \\ 6 & 7 & 8 \end{pmatrix} \quad \textit{in 3D} \\ \boldsymbol{\sigma} = \begin{pmatrix} \sigma_{xx} & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_{yy} & 0 \\ 0 & 0 & \sigma_{zz} \end{pmatrix} & = & \begin{pmatrix} 0 & 1 & 2 \\ 3 & 4 & 5 \\ 6 & 7 & 8 \end{pmatrix} \quad \textit{in 2D} \end{array}$$

Figure 2.3: *Components of the stress tensor in 2D and 3D in Paraview.*

Other output files

Any text editor or spreadsheet software can be used to visualise the other output files such as the **.PAR* and **.ENG* files. The *PAR* file stores data from a given material point, which has to be specified in the *CPS* file, `$$OUTPUT_MATERIAL_POINTS`. The *ENG* files gives the values

of kinetic energy, internal work of the system, and time step information. Chapter 3.6 explains how to create and read text output files.

2.2 Downloading Anura3D source code

The Anura3D source code is stored in the Anura3D GitHub repository in the *src* folder. You can visualize the files by clicking on each file in the *src* folder. You can download all the files by clicking on "Code" and then select "Download ZIP".

We recommend that external users and developers fork the code. Information on how to fork a repository in GitHub can be found here.

2.3 Compiling Anura3D

Before performing any simulation with Anura3D, it is required to compile the code and create an executable file.

2.3.1 Prerequisites

Anura3D is written in Fortran language and requires a text editor and a compiler to visualize the code and create the executable. The current version (*Anura3D 2025*) is supported only on Windows operating systems and requires Visual Studio (VS) and a Fortran Compiler, distributed in the oneAPI package (Base + HPC Toolkits). Since VS is a dependency for the oneAPI software, not all combinations of VS and oneAPI are compatible. Check the compatibility requirements using this link. Make sure to include the Intel oneAPI Math Kernel Library (oneMKL) during the installation of the Intel oneAPI Base Toolkit.

The VS Community version 2022 is recommended and is freely distributed. Similarly, the oneAPI release is suggested. Intel offers academic licenses, provided an active *.edu* email address. Visit the sites shown bellow to download VS Community 2022 release and the oneAPI base and high performance computing toolkits.

https://visualstudio.microsoft.com/vs/community/

https://software.intel.com/content/www/us/en/develop/tools/oneapi/base-toolkit/download.html

The following steps are recommended for the successful installation of VS and oneAPI.

(1) Check that the minimum hardware requirements for VS and oneAPI are fulfilled.
(2) Download and **install VS first**.
(3) When installing VS, the user is prompt to select custom *workloads*. Select *"Desktop development with C++"* as shown in Figure 2.4.
(4) For different versions of VS the corresponding C++ components must be installed. Consult this site's information to get help.
(5) Continue with the installation process following the instructions provided in the VS installer.
(6) Download and execute the installation file for the oneAPI (base and high performance computing toolkit). Check that oneAPI is compatible with the previously installed VS version.
(7) Depending on the oneAPI version, the installation manager will indicate if VS was installed successfully. If that is not the case, instructions are provided by the oneAPI installation

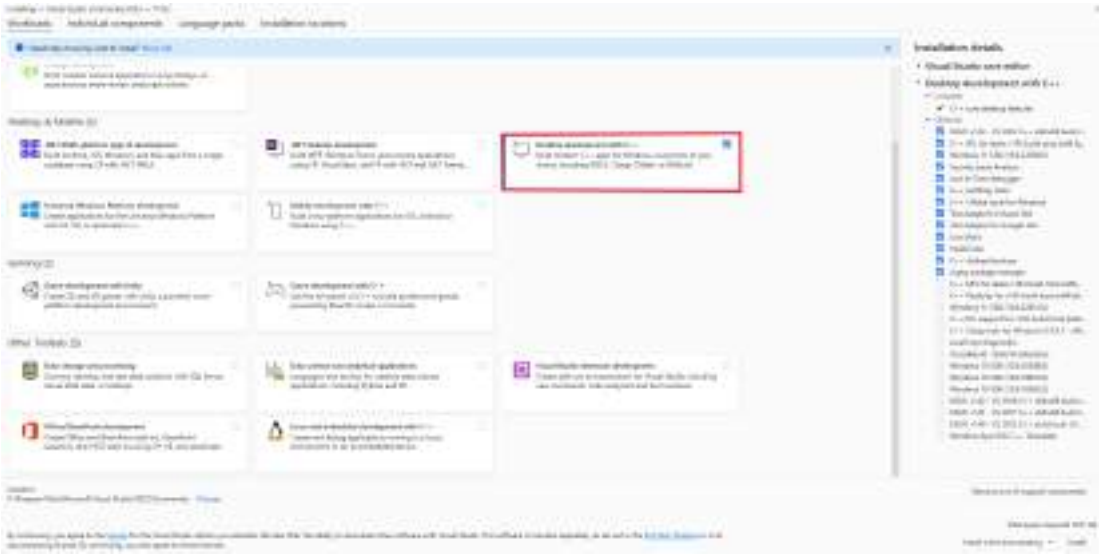


Figure 2.4: *Visual Studio installer- Workload tab.*

manager.

(8) In the case that VS was installed successfully, proceed to install oneAPI following the instructions on the screen.

(9) Once the installation is completed, open a new project in VS as shown in Figure 2.5.

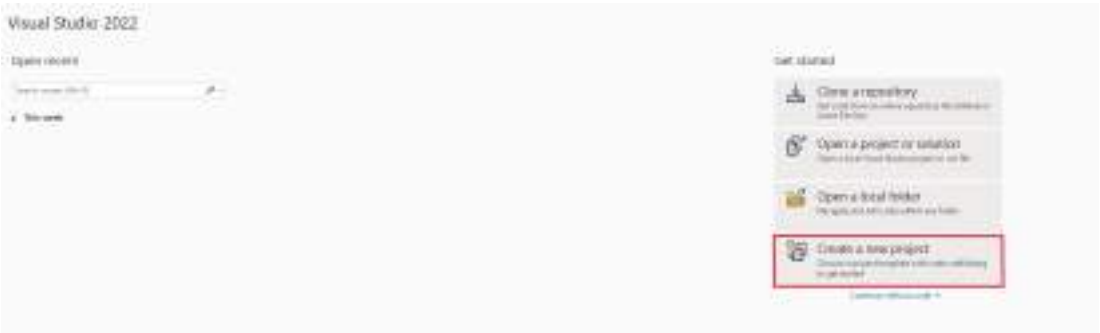


Figure 2.5: *Visual Studio 2022: Creating a new project.*

(10) If *Fortran* appears under the *All languages* drop-down list as it is shown in Figure 2.6, the installation of VS and oneAPI is successful, and they are integrated correctly.

Figure 2.6: *If Visual Studio 2022 and oneAPI are integrated correctly, Fortran should be available in the dropdown menu.*

2.3.2 Compiling Anura3D

Once the installation of VS and oneAPI is completed, the Anura3D code can be compiled. The following steps show how to build the code together with the included constitutive models.

(1) Navigate the code folder to the `"../Anura3D/VS"` folder.
(2) Find and double click on the `Anura3D.sln` file.
(3) VS should initialize. If the screen observed in Figure 2.7 doesn't automatically appear, open the *Anura3D.for* file from the *Solution Explorer* window. Select "Release" from the solution configurations dropdown menu.

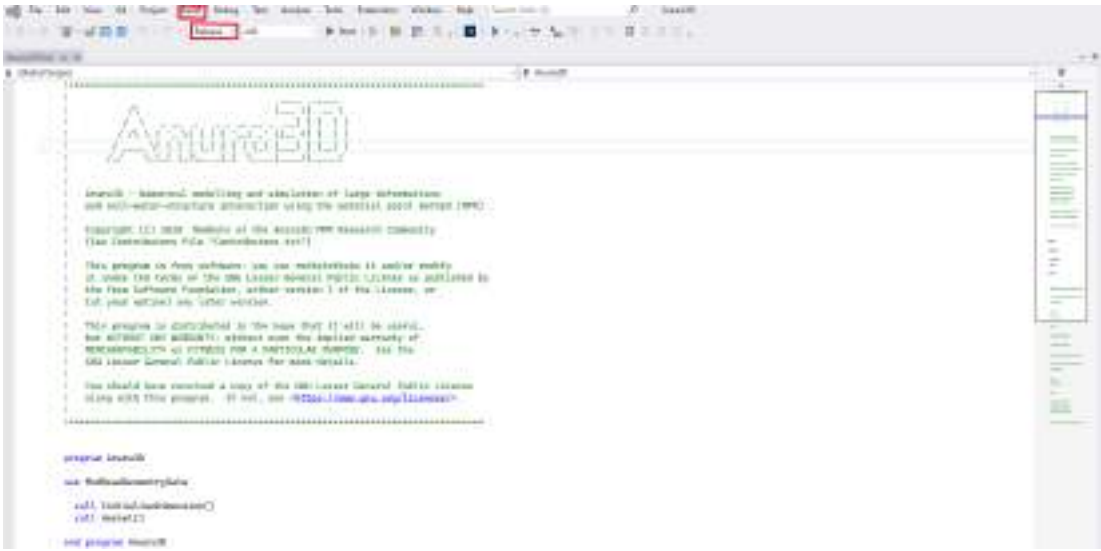


Figure 2.7: *Anura3D code in Visual Basic 2022.*

(4) Click on the *Build* menu (Figure 2.7).
(5) Click on *Build Solution* (Figure 2.8). This will compile the Anura3D executable.

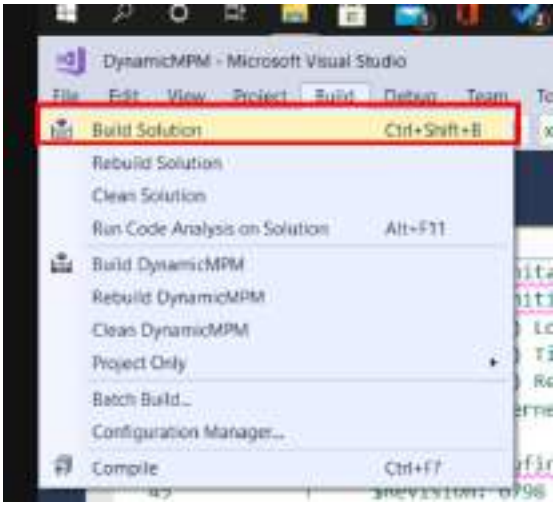


Figure 2.8: *The code will be compiled after clicking on Build Solution.*

(6) If the compilation is completed successfully, the executable is stored in the `"../Anura3D/VS/Release"` folder.

(7) For troubleshooting look at the error messages shown in the *Error List* (select View/Error List from the top menu) or the *Output* window as it is shown in Figure 2.9 and direct your questions to the forum at www.Anura3D.com specifying your VS and oneAPI versions with the error message displayed a the *Error List* or *Output* window.

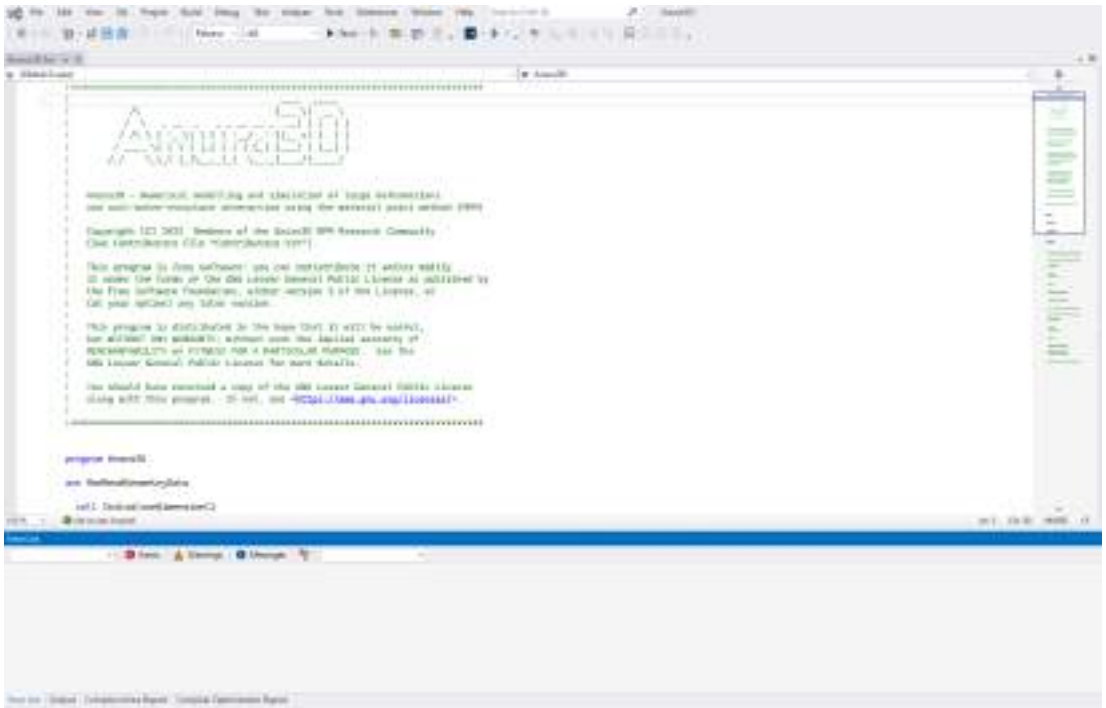


Figure 2.9: *Error List window in Visual Studio Community 2022.*

2.4 Installation of GiD (pre-processor)

GiD is a licensed software with free one-month license for first-time user, and can be renewed for free up to three times. The installation process is as follows.

2.4.1 Installation instructions

(1) Download the installation files (i.e. v17) at

https://www.gidsimulation.com/gid-for-science/downloads/

(2) Check the version number and operating system, and click on the "Download" button as shown in Figure 2.10.

Figure 2.10: *Download GiD v16 installation files.*

(3) Start the installation and follow the on-screen instructions.

(4) Obtain the one-month free license at:

http://www.gidhome.com/purchase/passwords

(5) Select the version number, license type, and time period as shown in Figure 2.11.

Figure 2.11: *Obtain one-month free password for GiD.*

(6) Follow the instructions on the website and provide the required information. A one-month free password will directly be provided (at the bottom of the website).

(7) Enter the password in the "Enter password window" in GiD as shown in Figure 2.12, or navigate to Help/Register GiD/Local/Enter Password in the menu.

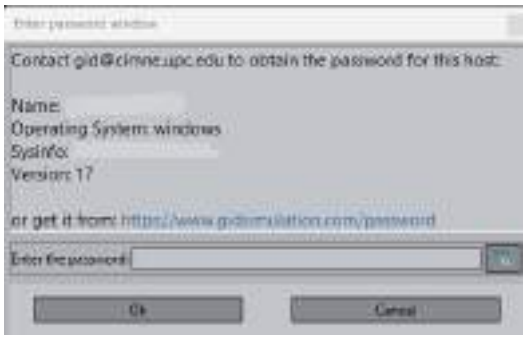


Figure 2.12: *Enter password in GiD.*

Anura3D 2025 problemtype is meant to be used with GiD v17.0, however, GiD v16.0 is also compatible. The GiD community and the Anura3D MPM Research Community do not support the use of different versions.

2.4.2 The Anura3D 2025 problemtype

The Anura3D 2025 "problemtype" folder is needed to use the Anura3D Software features in the GID pre-processor (more in section 2.1). The folder is partially pre-assembled and distributed with the Anura3D code. The "problemtype" folder must have the structure shown in Figure 2.13 to be completely functional, which includes the executable and other dependencies.

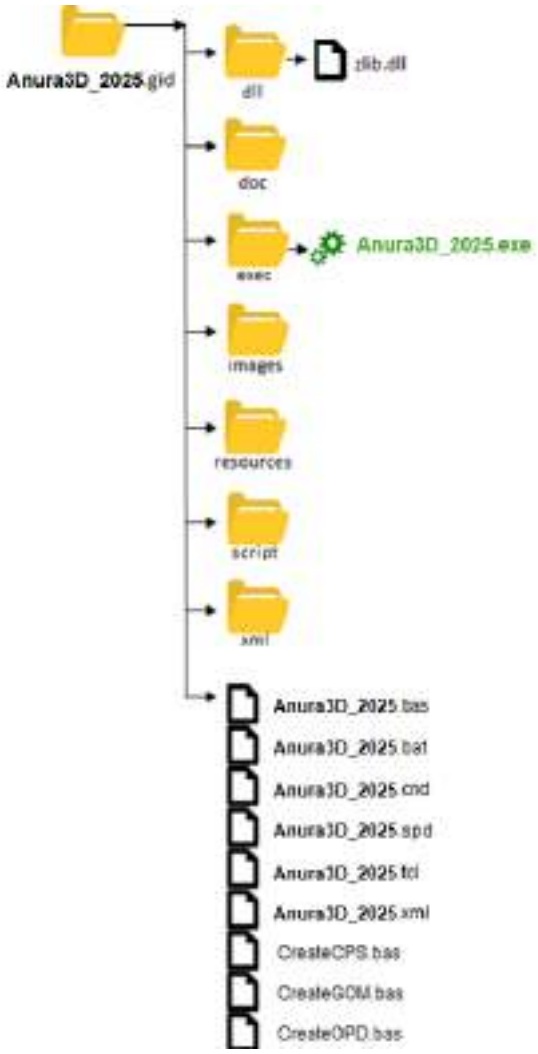


Figure 2.13: *"Problemtype" folder structure. The green file represent the path where the Anura3D_2025.exe must be pasted.*

To complete the assembling of the problemtype file and ensure its functionality proceed with the following steps:

(1) Copy the executable file from `"../Anura3D/VS/Release/"` to `"...Anura3D_2025.gid/exec/"`.
(2) Finally, copy the Anura3D 2025 "problemtype" into the GiD installation directory as shown in Figure 2.14.

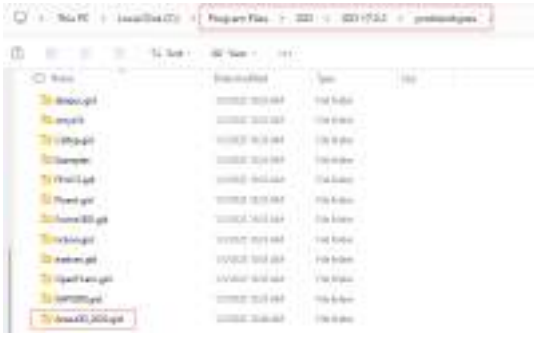


Figure 2.14: *Copy Anura3D 2025 problemtype into GiD installation directory.*

2.5 Installation of ParaView (post-processor)

ParaView is an open-source software under a permissive BSD license and is, thus, free to use with Anura3D. The original ParaView software will be used without any modifications of its source code. The following steps have to be performed to install ParaView.

(1) Download the installation files of the latest supported ParaView version (i.e. v5.13) at:

http://www.paraview.org/download

(2) Select the version number, download type and operating system, and click on the <Download> button as shown in Figure 2.15.

Figure 2.15: *Download ParaView v5.13 installation files.*

(3) Start the installation executable and follow the on-screen installation instructions.

After installation ParaView is ready for use to post-process Anura3D calculation results.

Important note for the first time use of ParaView
If you use ParaView for the first time after installation, the following changes are recommended to secure an improved visualisation of the results.

(1) In the ParaView main menu, choose <Edit> <Settings...> as indicated in Figure 2.16 and the "Settings" dialogue box will open.

Figure 2.16: *Change settings in ParaView.*

(2) Select the tab "Color Palette" as indicated in Figure 2.17

(3) Click on <Select palette to load...> drop down menu and select the option <White Background> as shown in Figure 2.17.

(4) The colours should be updated. Click on <OK> to save the changes.

Figure 2.17: *Change of "Color Palette" in ParaView.*

(5) Choose <View> from the menu and select the options of the drop down menu as indicated

in Figure 2.18.

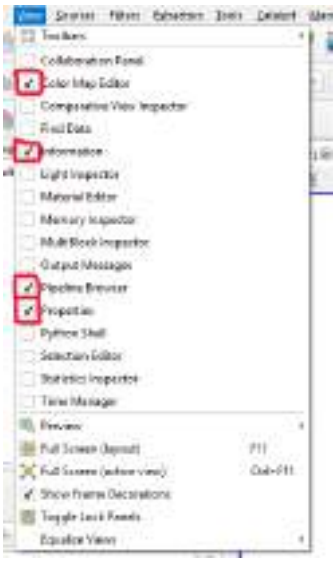


Figure 2.18: *Select "View" options in ParaView.*

These are one-time actions and the choices will be stored as default for any further use of ParaView.

3 One-dimensional consolidation (oedometric compression test)

Authors: Alba Yerro; Alexander Rohe; Alexander Chmelnizkij; James Fern; Francesca Ceccato; John Murphy; Grace Roberts; Pietro Marveggio; Mateu Maglia; Gaia Di Carluccio

The aim of this tutorial is to simulate a one-dimensional (1D) consolidation problem, also known as an oedometric compression test, using the two-phase single-point MPM formulation in Anura3D . This means that a single set of material points is generated and are representative for both the solid and liquid phases of the continuum. More information on this modelling approach is available in Chapter 2 of Fern et al. [1]. In Anura3D , the continuum can be represented by a two-dimensional (2D) or a three-dimensional (3D) discretization. Both cases are explained in this chapter.

The following steps are required for the simulation of a 1D compression and consolidation test (oedometer problem).

- ◇ Create the geometry in 2D (or 3D);
- ◇ Choose the material model and definition of the material properties;
- ◇ Definition of loading and boundary conditions in 2D and 3D;
- ◇ Generation of the mesh in 2D (or 3D);
- ◇ Execution of calculation;
- ◇ Output of results.

Several variations of the problem are proposed in Section 3.7 in order to investigate the effects of local damping for the dynamic explicit formulation. This is done by changing the local damping coefficient. In addition, it is suggested to decrease the external load in order to perform a simulation with small deformations for comparison with analytical results.

3.1 Problem definition in 2D and 3D

A saturated soil column of 1m height is loaded with a vertical load of q = 400 kPa, which will be suddenly applied on the upper boundary and maintained throughout the calculation. The geometry of the problem in 2D and 3D is shown in Figure 3.1 and the dimensions are given in Table 3.1. The bottom boundary will be defined as impervious and, therefore, the drainage length is H = 1 m. Gravity is neglected. The bottom boundary is fixed and the soil column can only deform in the vertical direction. Note that an empty space (T) is included at the top of the column to allow for vertical movement of the material (points) at the top of the column.

The material is considered elastic and the model parameters are given in Table 3.2. Note that the bulk modulus of the liquid is considered 100-times smaller than the physical one in order to decrease the computational time as a larger critical time step size is obtained. This does not have a significant influence on the results but provides a computationally faster simulation. The intrinsic permeability of $\kappa = 1.0214 \cdot 10^{-9} m^2$ is equivalent to a Darcy permeability of k = 0.01 m/s.

Table 3.1: *Dimensions of the oedometer problem in 2D and 3D.*

Dimension	Height H [m]	Width W [m]	Depth D [m]	Empty space T [m]
2D	1.0	0.1	–	0.1
3D	1.0	0.1	0.1	0.1

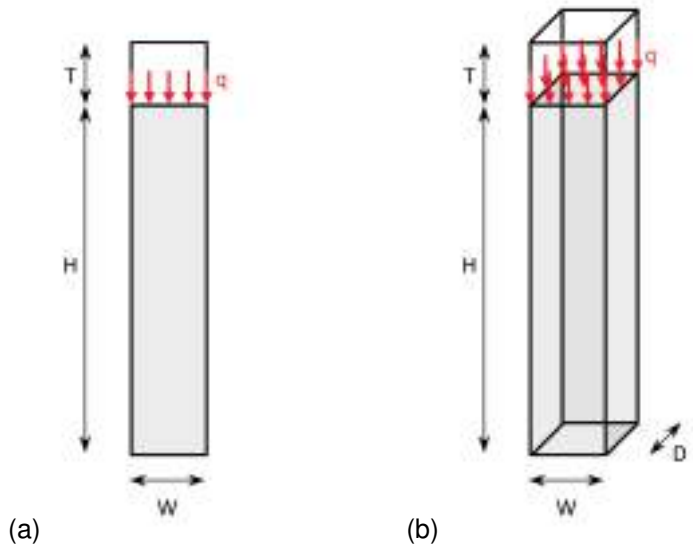


Figure 3.1: *Problem definition for the oedometer simulation including empty space (T) to allow for large deformation: (a) 2D case and (b) 3D case.*

Table 3.2: *Material properties of the oedometer problem.*

Parameter	Symbol	Unit	Value
Material type: saturated soil – fully coupled			
Initial porosity	n_0	–	0.4
Density solid	ρ_s	kg/m^3	2650
Density liquid	ρ_l	kg/m^3	1000
Intrinsic permeability	κ	m^2	$1.0214 \cdot 10^{-9}$
Bulk modulus liquid	K_l	kPa	$2.15 \cdot 10^4$
Dynamic viscosity liquid	μ_d	kPa/ s	$1.002 \cdot 10^{-6}$
Material model solid: linear elasticity			
Effective Poisson ratio	ν'	—	0.3
Effective Young's modulus	E'	kPa	1000

3.2 Creating input data in GiD

Most simulations in Anura3D consist of multiple calculation stages. To initiate the calculation for each stage, Anura3D requires three input files: one GOM-stageX and one OPD-stageX files, which are specific to each stage "X," and a CPS file, which follows an independent numbering system related to the number of calculation steps performed in each stage. The GOM-stageX file includes the geometry, material properties, boundary conditions, loading conditions, and mesh properties for a specific stage. The OPD-stageX file contains specification about output data, such as variables to plot, output writing frequency, and selected material points. The CPS file defines the calculation options. All input files for each calculation stage can be created using GiD (see Section 2.4 for software installation instructions), which is described in Sections 3.2.1 to **??**.

The Anura3D menu data-tree is available in GiD and it is recommended to follow each item top down (see Figure 3.2). The Anura3D menu appears after selecting the Anura3D problemtype as explained in the following Section 3.2.1.

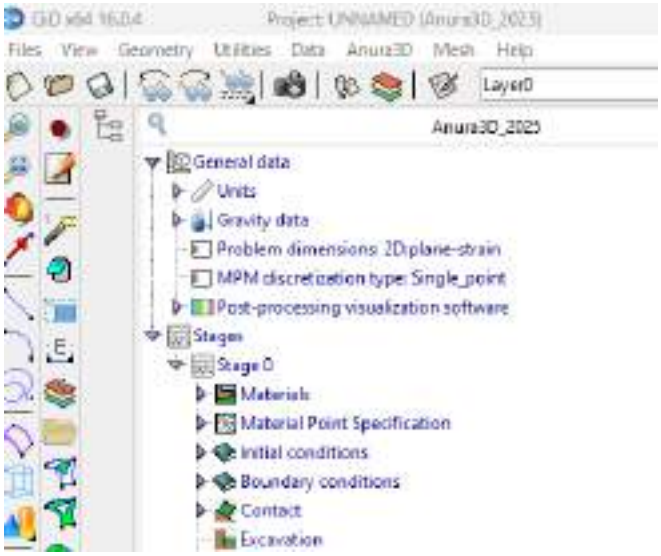


Figure 3.2: *GiD data-tree to prepare Anura3D calculation input files.*

3.2.1 Starting pre-processor and selecting problem type

(1) Start the GiD programme (version 16.0.7 or newer) by clicking on the Windows desktop icon (Figure 3.3).

Figure 3.3: *Starting the GiD pre-processor.*

(2) Choose the Anura3D problemtype:
Select <Data> <Problem type> from the menu.
Choose <Anura3D_2025> in the list of problemtypes as shown in Figure 3.4.

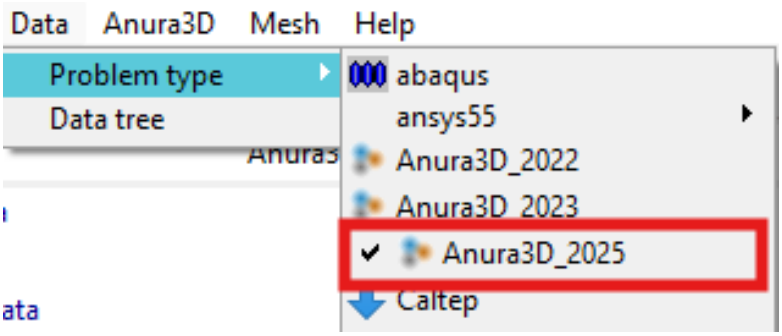


Figure 3.4: *Choosing the "Anura3D 2025" problemtype in GiD.*

(3) The disclaimer for using the Anura3D software appears as shown in Figure 3.5 and has to be accepted to be able to continue.

Figure 3.5: *Disclaimer for using the Anura3D software.*

(4) Save the project. Click on icon <Save GiD project> in the sidebar OR select <File> <Save as...> from the menu as shown in Figure 3.6 OR press <CTRL>+<S>.

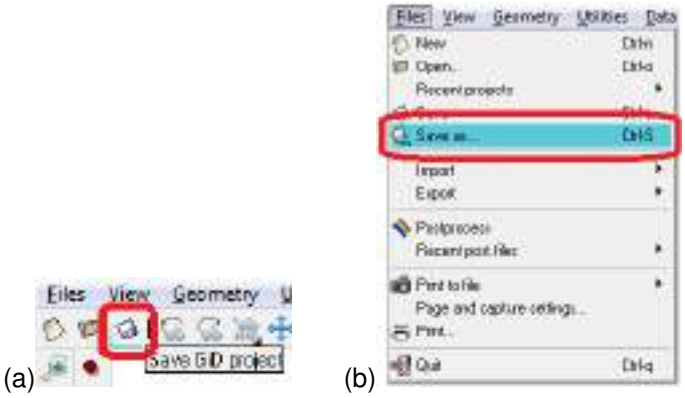


Figure 3.6: *Saving the project in GiD: (a) sidebar button or (b) drop-down menu.*

(5) Save the project in the desired directory on the hard disk with an appropriate name (e.g. "oedometer") as shown in Figure 3.7. **Make sure that the projectname does not contain any spaces.**

Figure 3.7: *Save project files in GiD for the oedometer problem.*

3.2.2 Defining geometry in 2D and 3D

The geometry consists of points, lines, surfaces, and in 3D also volumes. The approach is (1) to define lines between points, and then (2) to define surfaces from them. For the 3D case, volumes will be created from the surfaces. The geometry of the problem is shown in Figure 3.1 with dimensions as given in Table 3.1.

The 2D geometry will be created in the x-y-plane with z-coordinate equal to zero. In that case entering the z-coordinate is not required. GiD automatically assumes the z-coordinate equal to zero.

Creating lines (for 2D and 3D case)
The lines are created in the following way.

(1) Click on icon <Create Line> in the sidebar OR choose <Geometry> <Create> <Straight Line> from the menu as shown in Figure 3.8.

Figure 3.8: *Creating lines in GiD: (a) sidebar button or (b) drop-down menu.*

(2) In the command line, enter the coordinates of the first point. Note that there must be a space between numbers.

- ◇ 2D case: **0 0**, as shown in Figure 3.9(a) and confirm by pressing <ENTER>.
  The x- and y-coordinates are provided. For the 2D case, entering the z-coordinate is not required as GiD automatically assumes it equal to zero.
- ◇ 3D case: **0 0 0**, as shown in Figure 3.9(b) and confirm by pressing <ENTER>.
  The x-, y-, z-coordinates are provided.

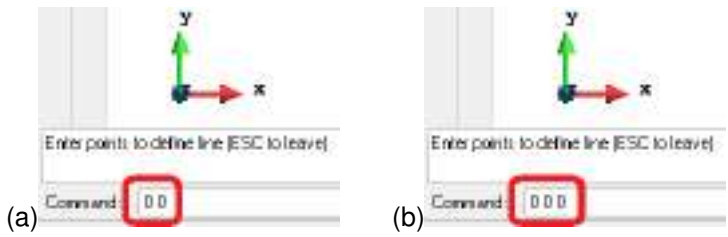


Figure 3.9: *Entering coordinates at command line in GiD: (a) 2D case and (b) 3D case.*

(3) Enter coordinates of the remaining points of the lower rectangle.

- ◇ 2D case: Second point: **0.1 0** <ENTER>
  Third point: **0.1 1** <ENTER>
  Fourth point: **0 1** <ENTER>
  Enter coordinates of the last point: **0 0** <ENTER>, to close the rectangle by connecting the line with the starting point.

◇ 3D case: Second point: **0.1 0 0** <ENTER>
Third point: **0.1 1 0** <ENTER>
Fourth point: **0 1 0** <ENTER>
Enter coordinates of the last point: **0 0 0** <ENTER>, to close the rectangle by connecting the line with the starting point.

(4) A dialogue box appears asking if the lines have to be joined. Confirm by clicking on <Join> as shown in Figure 3.10.

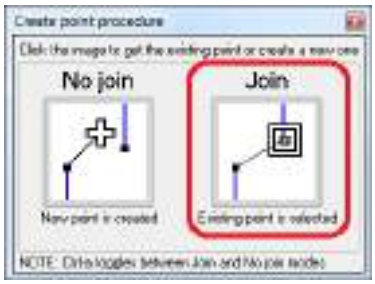


Figure 3.10: *Joining points in GiD.*

(5) Press <ESC> once to disconnect the cursor from the last point.
(6) Enter coordinates of the upper rectangle. In the dialogue box, always choose <Join> to connect to the existing geometry.

◇ 2D case: First point: **0.1 1** <ENTER>
Second point: **0.1 1.1** <ENTER>
Third point: **0 1.1** <ENTER>
Fourth point: **0 1** <ENTER>

◇ 3D case: First point: **0.1 1 0** <ENTER>
Second point: **0.1 1.1 0** <ENTER>
Third point: **0 1.1 0** <ENTER>
Fourth point: **0 1 0** <ENTER>

(7) Press <ESC> twice to quit the line creating process.

To resize the view of the geometry, click on the icon <Zoom frame> in the sidebar OR choose <View> <Zoom> <Frame> from the menu as shown in Figure 3.11.

Figure 3.11: *Resizing view in GiD: (a) sidebar button or (b) drop-down menu.*

The geometry of the oedometer problem after creating lines should look like as shown in Figure 3.12.

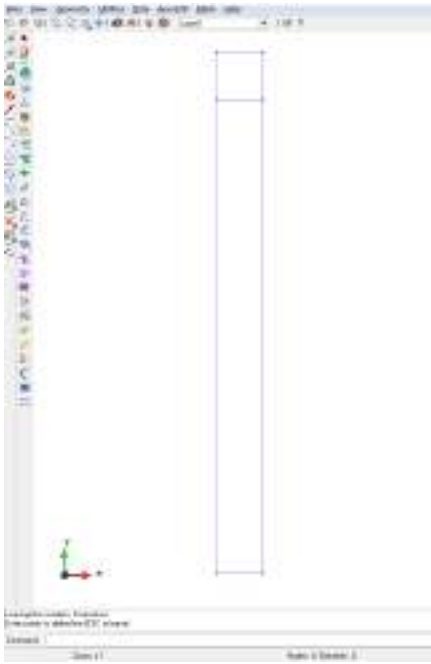


Figure 3.12: *Geometry of the oedeometer problem after creating lines (2D and 3D case).*

Creating surfaces (for 2D and 3D case)

The surfaces are created in the following way.

(1) Click on the icon <Create NURBS Surface> in the sidebar OR choose <Geometry> <Create> <NURBS surface> <By contour> from the menu as shown in Figure 3.13.

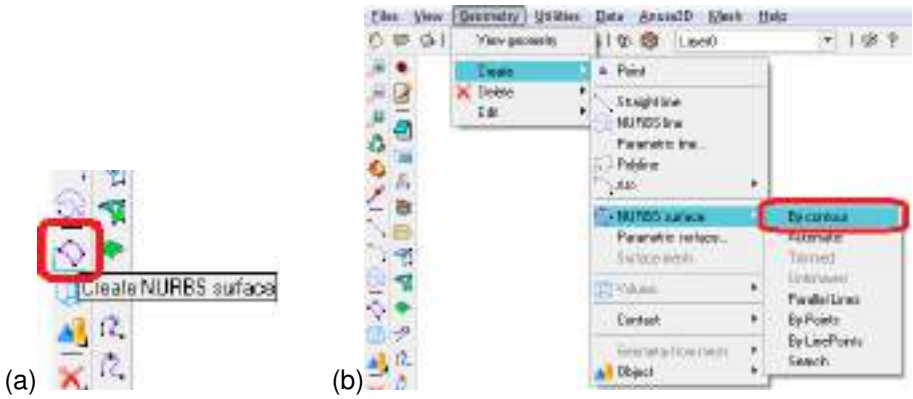


Figure 3.13: *Creating surfaces in GiD: (a) sidebar button or (b) drop-down menu.*

(2) Select the four boundary lines of the upper rectangle by clicking with the cursor on each line. After selection each line should turn red as shown in Figure 3.14a.

(3) Press <ESC> once to leave the selection of lines. The created surface appears in the upper rectangle as shown in Figure 3.14b.

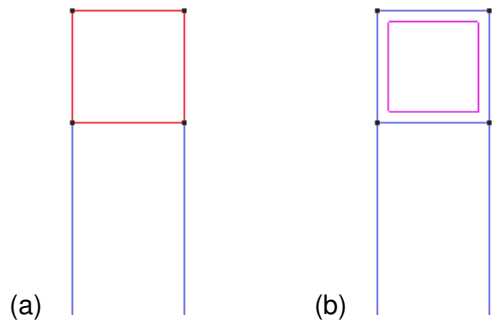


Figure 3.14: *Example of (a) selected lines and (b) created surface.*

(4) Repeat the previous two steps for the lower rectangle.
(5) Press <ESC> twice to quit the surface creating process.

The geometry of the oedometer problem after creating surfaces should look like as shown in Figure 3.15. For the 2D case, one can continue to define the material properties in Section 3.2.5. For the 3D case, volumes have to be created as described below.

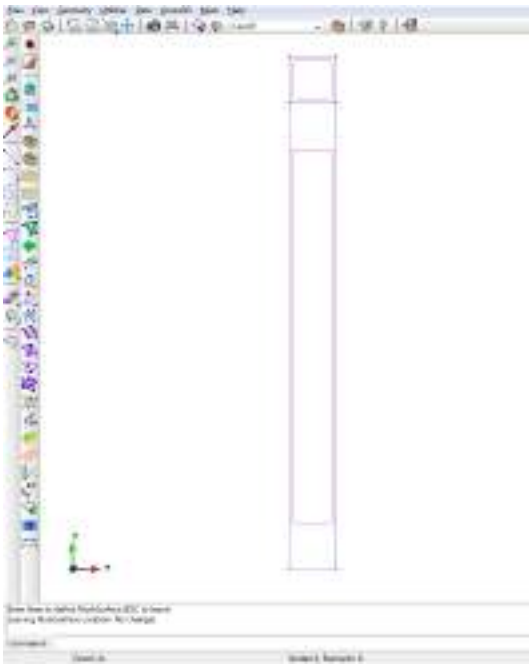


Figure 3.15: *Geometry of the oedeometer problem after creating surfaces (2D and 3D case).*

Creating volumes (only for 3D case, for the 2D case continue with Section 3.2.5)
Once the surfaces are created, the column volume can easily be created by extruding the surface in z-direction. The following steps should be taken in order to extrude the surface entities to volumes.

(1) Choose <Utilities> <Copy...> from the menu as shown in Figure 3.16.

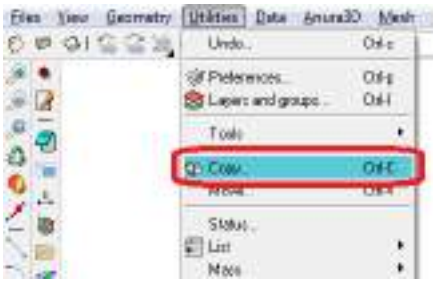


Figure 3.16: *Open dialogue box to copy entities in GiD.*

(2) In the "Copy" dialogue box make the following selections:

- ◇ Select "Entities type" as "Surfaces".
- ◇ Set "Transformation" to "Translation".
- ◇ Enter the extrusion vector. In this case the second point is (0.0, 0.0, 0.1).
- ◇ Select "Do extrude" as "Volumes".
- ◇ Keep the other options as selected by default.

The dialogue box should look like shown in Figure 3.17.

Figure 3.17: *Open dialogue box to copy entities in GiD.*

(3) Click on "Select" in order to select the surfaces that should be extruded.

(4) Select the surface of the upper rectangle by clicking with the cursor on the surface. After selection the surface should turn red as shown in Figure 3.18.

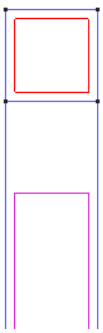


Figure 3.18: *Illustration of selected surface (upper surface) for creating volume entities.*

(5) Select the surface of the lower rectangle by clicking with the cursor on the surface. The surface should turn red once selected.
(6) Press <ESC> once to leave the selection of surface.
(7) Back in the "Copy" dialogue box click on <Cancel> to quit the volume creating process.

To rotate the view of the geometry, click on the icon <Rotate trackball> in the sidebar OR choose <View> <Rotate> <Trackball> from the menu as shown in Figure 3.19. It is also possible to press <shift> + right button on mouse (keep both pressed down) and move the mouse to navigate around.

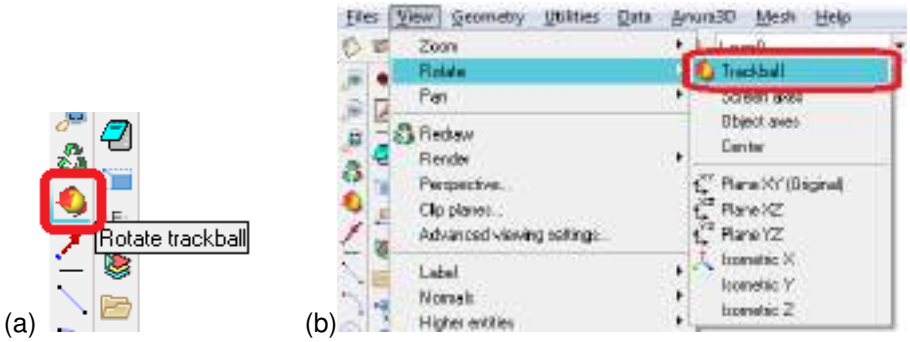


Figure 3.19: *Rotating view in GiD: (a) sidebar button or (b) drop-down menu.*

The geometry of the oedometer problem for the 3D case after creating volumes should look like shown in Figure 3.20.

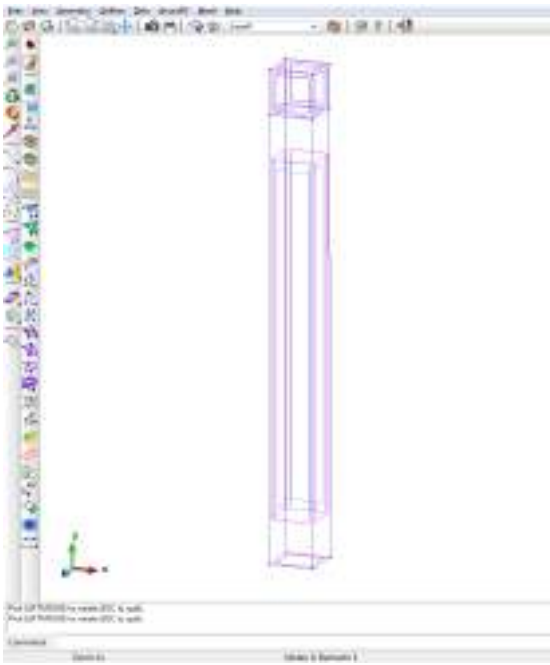


Figure 3.20: *Geometry of the oedometer problem after creating volumes (3D case).*

3.2.3 General data

Before starting defining materials and assign condition to the model, it is required to choose model dimensions and material point discretization type. The following steps are required.

(1) Choose <General data> from the data-tree as shown in Figure 3.21

Figure 3.21: *General data for oedometer problem.*

(2) The "Problem dimensions" option appears. Select either "2D:plane-strain" or "3D" according to the model dimension you are considering.

(3) In the "MPM discretization type" box define the discretization type you need. In this example we will consider "Single_point".

(4) Select "Paraview" and "GiD" as the post-processing visualization software by setting the value to "1" for their respective options.

3.2.4 Defining calculation stages

The oedometer example consists of a single stage in which the load is applied to the top surface. As a result, no additional stages has to be added, and all input information will be defined within the Stage 0 data tree.(Figure 3.22).

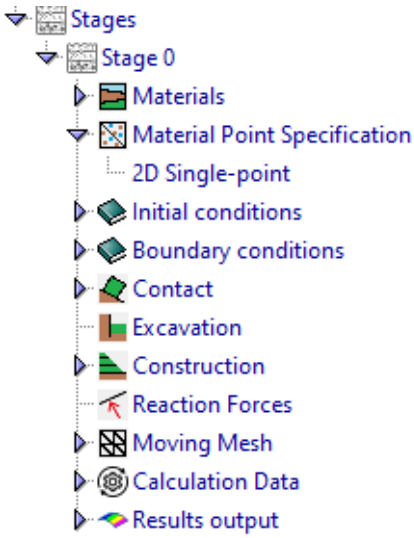


Figure 3.22: *Stage 0 data tree.*

3.2.5 Defining material properties

In this example, the oedometric compression of a saturated soil column will be analysed. The soil is represented by a two-phase material with fully coupled behaviour. The soil constitutive behaviour is linear elastic and the material parameters are given in Table 3.2. The following steps have to be taken in order to define the material parameters.

(1) Choose <Stages → Stage 0 → Materials> from the data tree.

(2) A standard material "Material 1" will appear. Select it and press "F2" or right click "Rename" and enter an appropriate name for the material, as shown in Figure 3.23. Confirm by clicking on the <Enter> button.

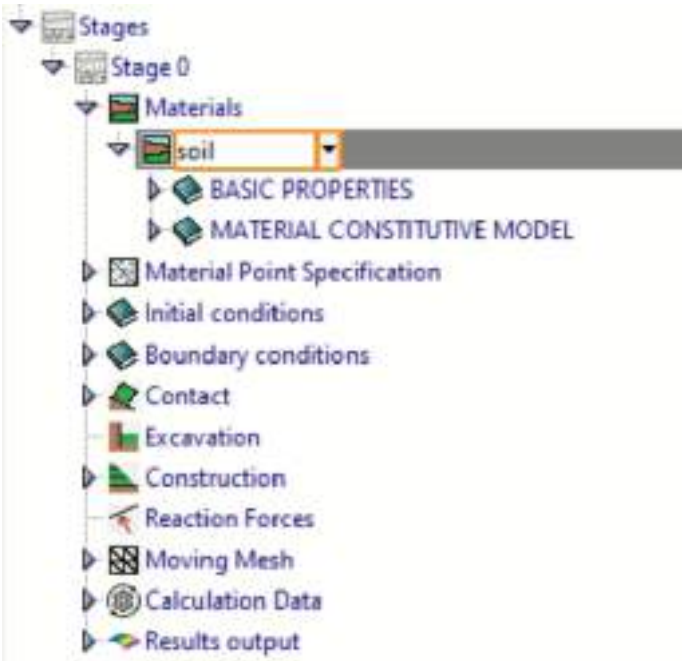


Figure 3.23: *Defining material name in GiD.*

(3) Enter the values for the material parameters as given in Table 3.2 and shown in Figure 3.24.

- ◇ Select "Material type" as "saturated material – fully coupled" from the drop down menu.
- ◇ Set "initial porosity" to "0.4".
- ◇ Set "density solid" to "2650".
- ◇ Set "density liquid" to "1000".
- ◇ Keep "K0-value" as "0.0". Note that this value is not used for the current example.
- ◇ Set "intrinsic permeability liquid" to "1.0214e-9".
- ◇ Set "bulk modulus liquid" to "2.15e4".
- ◇ Set "dynamic viscosity liquid" to "1.002e-6".
- ◇ Select "solid material model" as "Linear Elasticity" from the drop down menu.
- ◇ Set "effective Poisson ratio" to "0.3".
- ◇ Set "effective Young's modulus" to "1000".

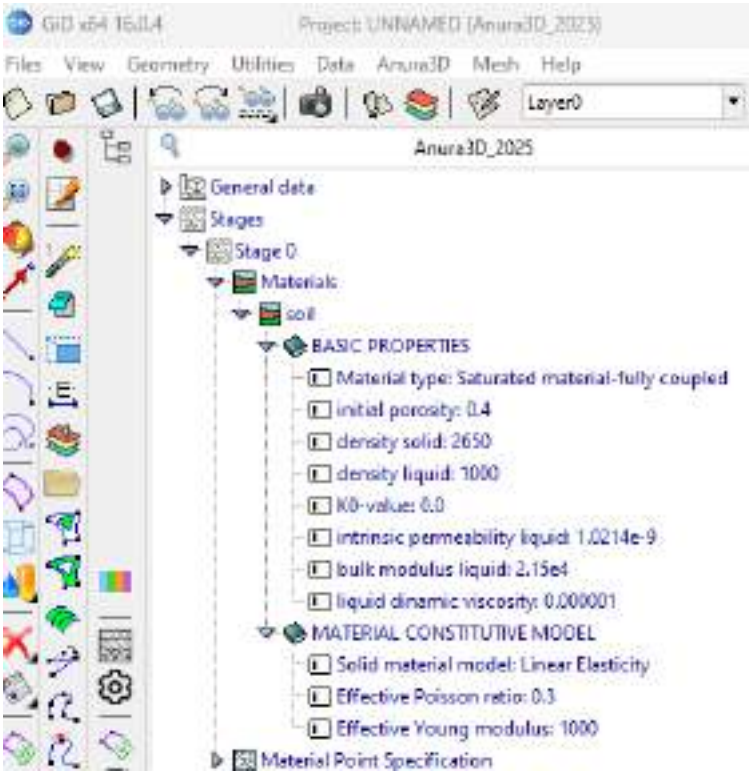


Figure 3.24: *Defining material properties for oedometer problem in GiD.*

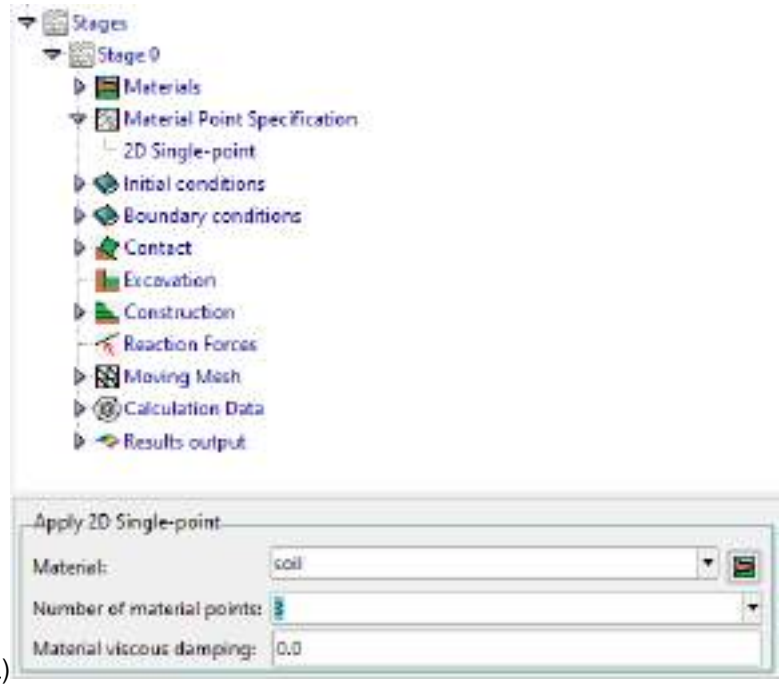
3.2.6 Specifying material points in 2D and 3D

The material points will be distributed throughout the domain by assigning the initial number of material points per element and it's material type. The following steps have to be taken to specify the material point distribution. In the same procedure the user assigns the material type and, if needed, the value of damping to material points.

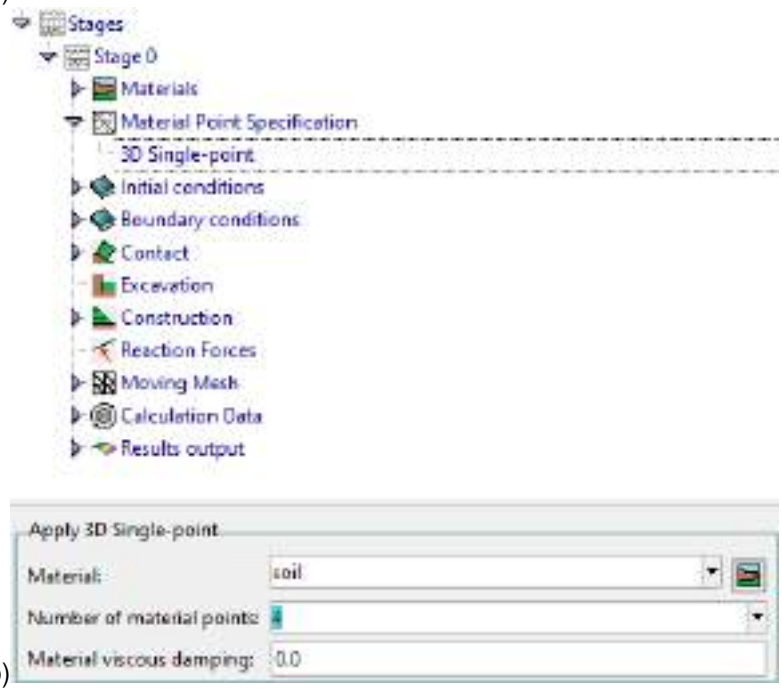
(1) Choose <Material Point Specification> from the data-tree in case of 2D or 3D simulation.

(2) The "Material Point Specification" dialogue box appears below the data-tree. The material model, number of material points and material viscous damping have to be specified as is done in Figure 3.25.

- ◇ For the 2D case, select the Material name at the corresponding top window. Then, enter "3" in the "Number of Material points" field. Note that for triangular elements 1 or 3 material points can be placed initially in an element. No damping is considered in this case. Click on the <Select> button in order to select the surfaces to which the material points should be assigned to.
- ◇ For the 3D case, select the Material name at the corresponding top window. Then, enter "4" in the "Number of Material points" field. Note that for tetrahedral elements 1, 4, 7, 8, 10, 13 or 20 material points can be placed initially in an element. Click on the <Select> button in order to select the volumes to which the material points should be assigned to.



(a)



(b)

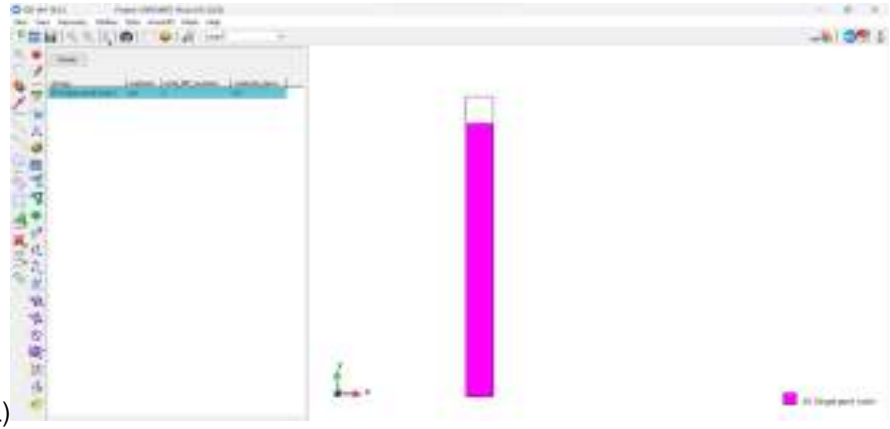
Figure 3.25: *Assigning material points to geometrical entities in GiD: (a) surfaces for 2D case and (b) volumes for 3D case.*

(3) Select the geometrical entity (surface in 2D, volume in 3D). Selected geometrical entities appear highlighted in red.

(4) Press <ESC> OR click on <End> to leave the selection of geometry entities. A group collecting multiple surfaces or volumes is created.

(5) Check if the material points are assigned correctly. Right click on "Material Point Specification" dialogue box, then click on <Draw> and select <Draw Values>.

(6) The material point specification of the oedometer problem should look like as shown in Figure 3.26.



(a)

(b)

Figure 3.26: *Checking assigned material points per element for oedometer problem: (a) 2D case and (b) 3D case.*

(7) Click on <Finish> to quit the material points drawing.

3.2.7 Defining boundary conditions in 2D and 3D

Applying fixities in 2D and 3D

The displacement at the boundaries will be fixed in normal direction and free in the tangential directions for both the solid phase and the liquid phase. Note that by defining fixities for the liquid phase at the bottom boundary, this boundary will be impervious and the drainage length will hence be 1m.

(1) Choose <Boundary conditions> <Fixities> from the data-tree.

(2) The "Fixities" dialogue box appears. A two-phase problem will be considered in this example. The conditions for the solid and water phase have to be applied separately.

- ◇ For the 2D case, fixities have to be applied on lines. Therefore, select the line icon at the bottom of the window as shown in Figure 3.28.
- ◇ For the 3D case, fixities have to be applied on surfaces. Therefore, select the surface icon at the bottom of the window as shown in Figure 3.29(b).

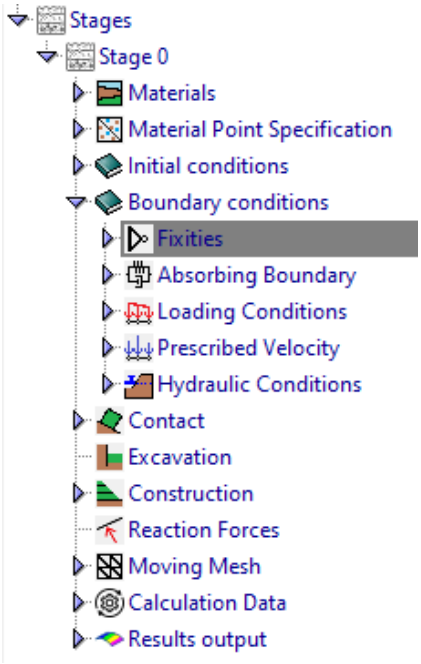


Figure 3.27: *Open dialogue box to define fixities in GiD.*

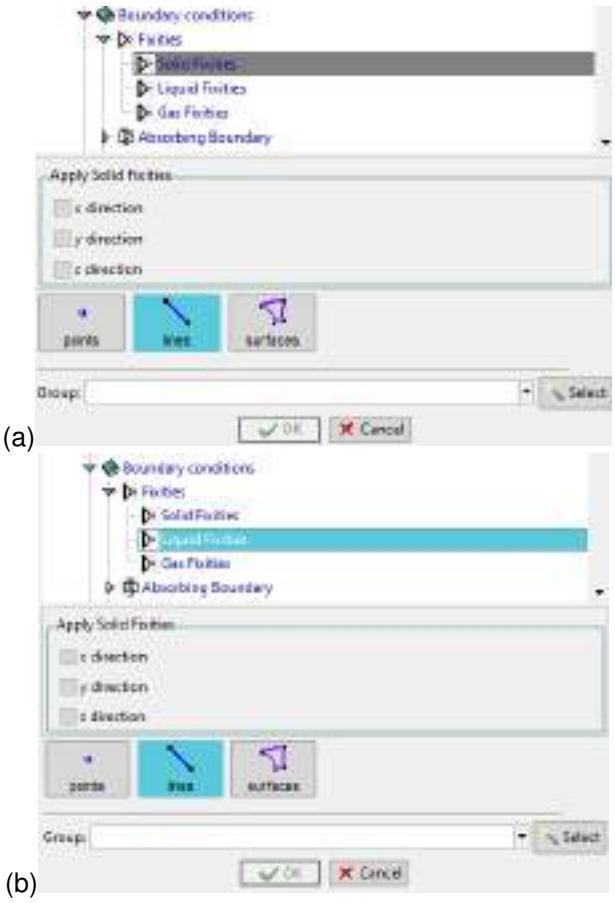


Figure 3.28: *Selecting line fixities and choosing (a) solid and (b) liquid fixities*

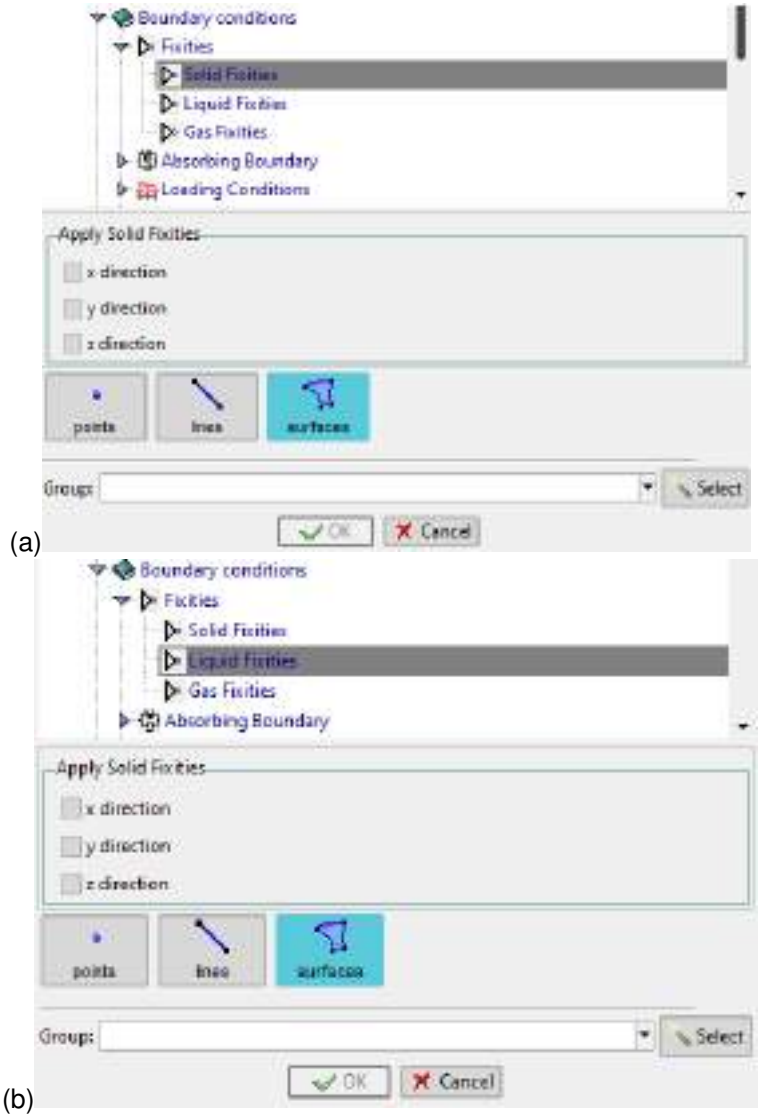


Figure 3.29: *Selecting surface fixities and choosing (a) solid and (b) liquid fixities*

(3) For both solid and liquid fixities, tick the checkbox "x-direction", and click on the <Select> button to select the geometrical entities that should be fixed in x-direction.

- ◇ 2D case: Select the two lines on the left side and the two lines on the right side by clicking on them. The selected lines should be highlighted in red.
- ◇ 3D case: Select the two surfaces on the left side and the two surfaces on the right side by clicking on them. The selected surfaces should be highlighted in red.

(4) Press <ESC> OR click on <End> in the "Fixities" dialogue box to leave the selection of geometrical entities.

(5) Click on <OK>.

(6) Now constraints in y-directions will be assigned. Double-Click on <Solid Fixities> or <Liquid Fixities> to assign a new fixity.

(7) Tick the checkbox "y-direction" and click on the <Select> button to select the geometrical entities that should be fixed in y-direction.

- ◇ 2D case: Select the line at the very top of the geometry and the line at the very bottom of the geometry by clicking with the cursor on them. The selected lines should be

highlighted in red.

- ◇ 3D case: Select the surface at the very top of the geometry and the surface at the very bottom of the geometry by clicking with the cursor on them. The selected surfaces should be highlighted in red.

(8) Press <ESC> OR click on <End> in the "Fixities" dialogue box to leave the selection of geometrical entities.

(9) Click on <OK>.

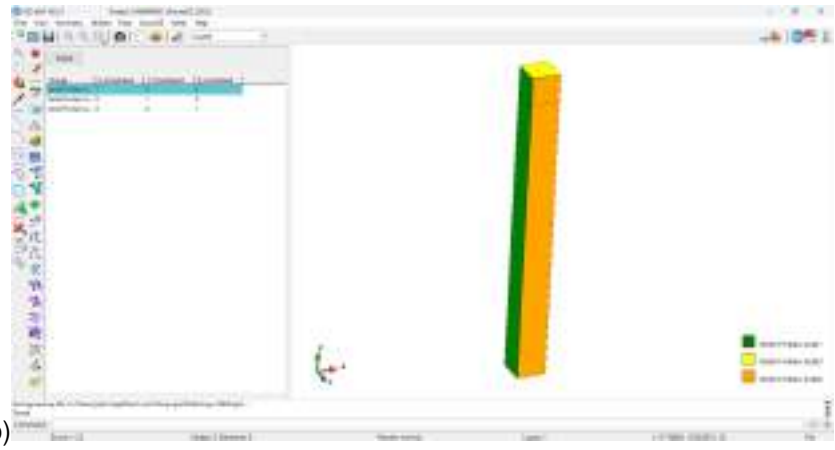
(10) For the 3D case only, we will assign constraints in z-directions. Double-Click on <Solid Fixities> or <Liquid Fixities> to assign a new fixity, then:

- ◇ Tick the check box "z-direction"
- ◇ Click on the <Select> button in order to select the surfaces that should be fixed in z-direction.
- ◇ Select the two surfaces on the front side and the two surfaces on the back side by clicking with the cursor on them. After selection the surfaces should turn red.
- ◇ Press <ESC> OR click on <End> in the "Fixities" dialogue box to leave the selection of surfaces.
- ◇ Click on <OK>.

(11) Check if the fixities are assigned correctly. Right click on the "Solid Fixities" or "Liquid Fixities" dialogue box click on <Draw> and select <Draw Value>.

(12) For example, the assigned solid fixities of the oedometer problem should look like Figure 3.30.

(a)



(b)

Figure 3.30: *Checking assigned fixities for oedometer problem: (a) 2D case and (b) 3D case.*

(13) Click on <Finish>.

Applying loading conditions in 2D and 3D

A vertical load of q = 400 kPa will be applied at the top of the soil column.

(1) Choose <Loading Conditions> from the data-tree.
(2) The "Loading Conditions" dialogue box appears.
(3) The load will be applied on the soil. Therefore, double click on the "Solid Traction" option from the menu as shown in Figure 3.31 for the 2D case (the load has to be applied on a line) and for the 3D case (the load has to be applied on a surface).

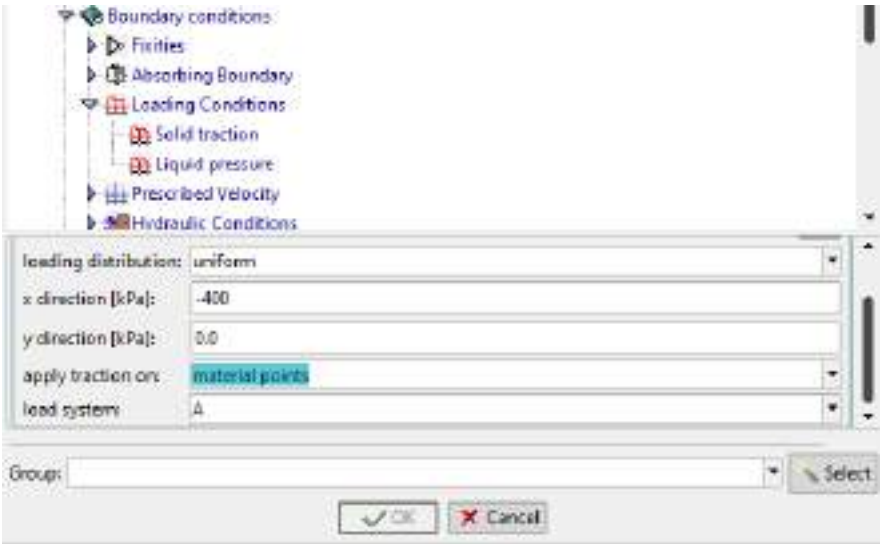


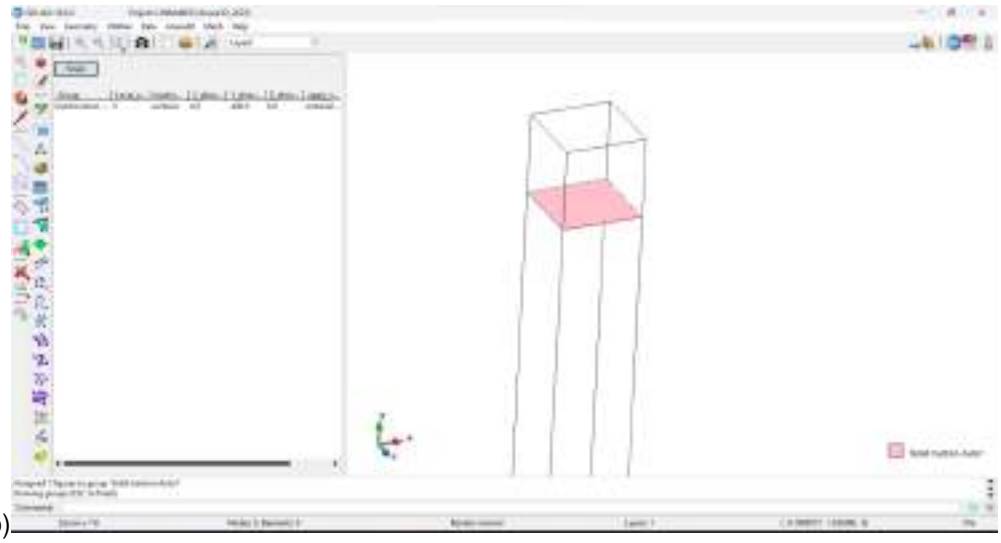
Figure 3.31: *Loading conditions dialogue box in GiD.*

(4) Enter the value of the load. A downward vertical load of 400 kPa will be applied. Therefore, enter "-400.0" in the field "y-direction [kPa]" as shown in Figure 3.31.
(5) As a large deformation problem is considered, the load has to move together with the material. Therefore, select "material points" from the drop down menu "apply traction on" as shown in Figure 3.31.
(6) Click on the <Select> button in order to select the geometrical entities that the load should be assigned to. In 2D only lines can be selected. In 3D only, surfaces can be selected. Selected entities appear highlighted in red.

- ◇ 2D case: Select the horizontal line between the two surface.
- ◇ 3D case: Select the horizontal surface between the two volumes.

(7) Within the Anura3D 2025 problemtype it is possible to assign a load to either Load System A or Load System B. This example only uses one load system and as such, the load is applied to Load System A.
(8) Press <ESC> OR click on <END> in the "Loading Conditions" dialogue box to leave the selection of geometrical entities.
(9) Press <Ok>
(10) Check if the load is assigned correctly. In the "Loading Conditions" dialogue box click on <Draw> and select <Draw values>.
(11) The assigned load of the oedometer problem should look like as shown in Figure 3.32.

(a) //



(b)

Figure 3.32: *Checking assigned loads for oedometer problem. a) 2D case; b) 3D case.*

(12) Click on <Finish> to end the "Draw values" mode.

3.2.8 Creating mesh in 2D and 3D

In Anura3D two element types are available as follows.

- ◇ Triangular elements for 2D calculations
- ◇ Tetrahedral elements for 3D calculations

Due to the implementation in the Anura3D programme kernel, 2D triangular elements have to be defined as linear type elements, and 3D tetrahedral elements have to be defined as quadratic type elements in GiD. It is important to ensure that the correct element type is chosen which is described in the following for both the 2D and 3D case.

It is possible to generate meshes in GiD in a very flexible way. If necessary, the mesh can be refined assigning a proper size by selecting <Mesh> <Unstructured> <Assign sizes...>, or a structured/semi-structured mesh can be generated. In this example, a semi-structured mesh is used.

(1) First the element type has to be set.

- ◇ 2D case: In the GiD main menu <Mesh>, choose the option <Quadratic type> and select the option **"Normal"** as shown in Figure 3.33(a). Then in the main menu <Mesh>, choose the option <Element type> <Triangle> (Fig. 3.34a) and select all surfaces.
- ◇ 3D case: In the GiD main menu <Mesh>, choose the option <Quadratic type> and select the option **"Quadratic"** as shown in Figure 3.33(b). Then in the main menu <Mesh>, choose the option <Element type> <Tetrahedra> (Fig. 3.34b) and select all volumes.

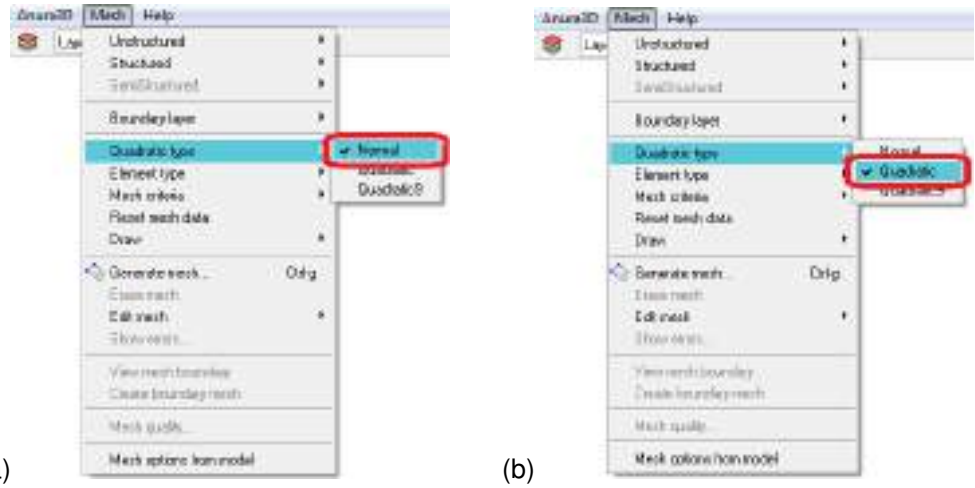


Figure 3.33: *Select element type in GiD: (a) For 2D case as "Normal" and (b) For 3D case as "Quadratic".*

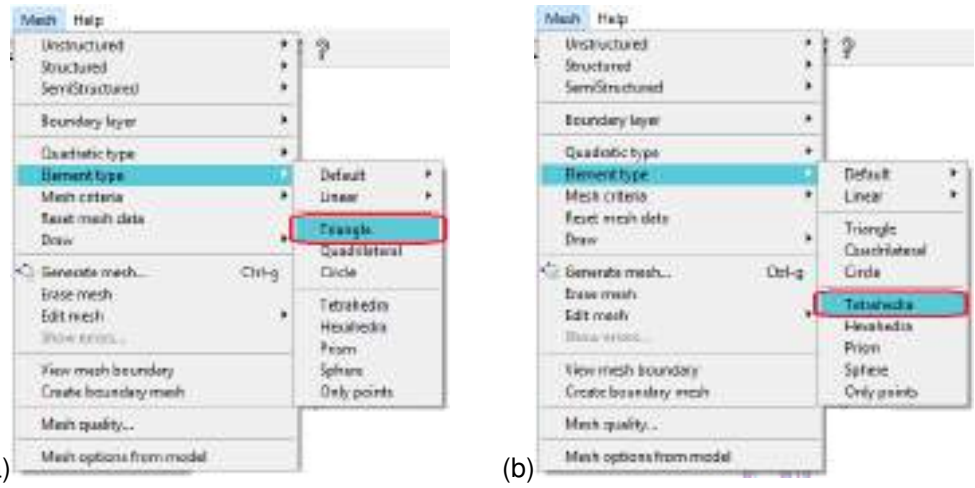


Figure 3.34: *Select element type in GiD: (a) For 2D case use triangles and (b) For 3D case use tetrahedra.*

(2) In order to generate a structured mesh:

- ◇ 2D case: In the GiD main menu <Mesh>, choose the option <Structured> <Surfaces> <Assign size>. Then select all the surfaces and press esc. GiD will ask the mesh size to assign to lines; enter 0.1. Click <Assign> and select all the lines (Fig. 3.35a).
- ◇ 3D case: In the GiD main menu <Mesh>, choose the option <Semi-Structured> <Volumes>. GiD will ask the number of divisions, enter 10, click <Assign>, select the

bottom volume and press Esc. Again GiD will ask the number of divisions, enter 1, click <Assign>, select the top volume and press Esc. Now go in the GiD main menu <Mesh>, choose the option <Semi-Structured> <Set> <Structured direction> and select all vertical lines (Fig. 3.35b).

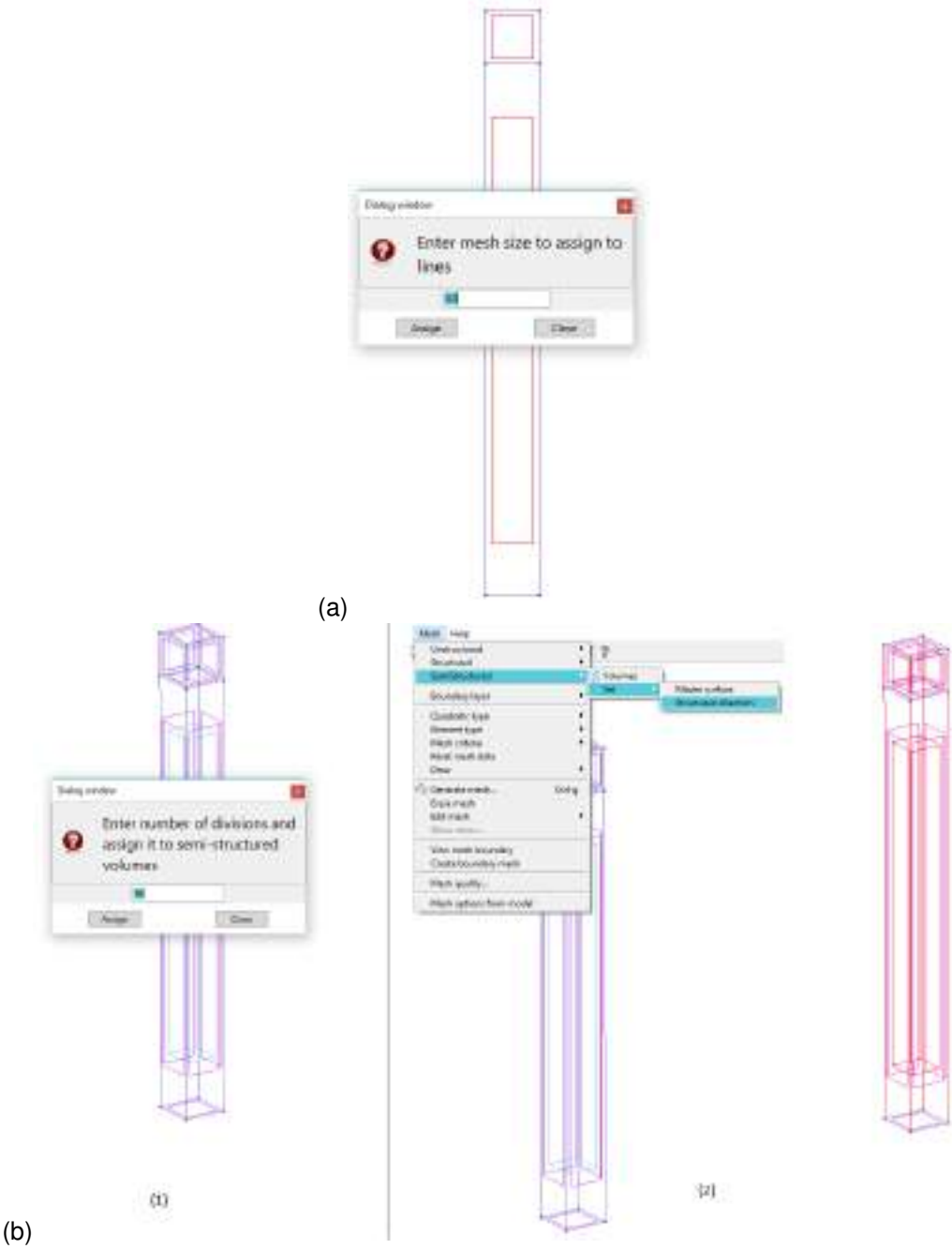


Figure 3.35: *Create structured mesh in GiD: (a) For 2D case and (b) For 3D case.*

(3) Click on the icon <Toggle geometry-mesh view> in the sidebar OR choose <Mesh> <Generate Mesh...> from the menu as shown in Figure 3.36 OR press <CTRL>+<G>.

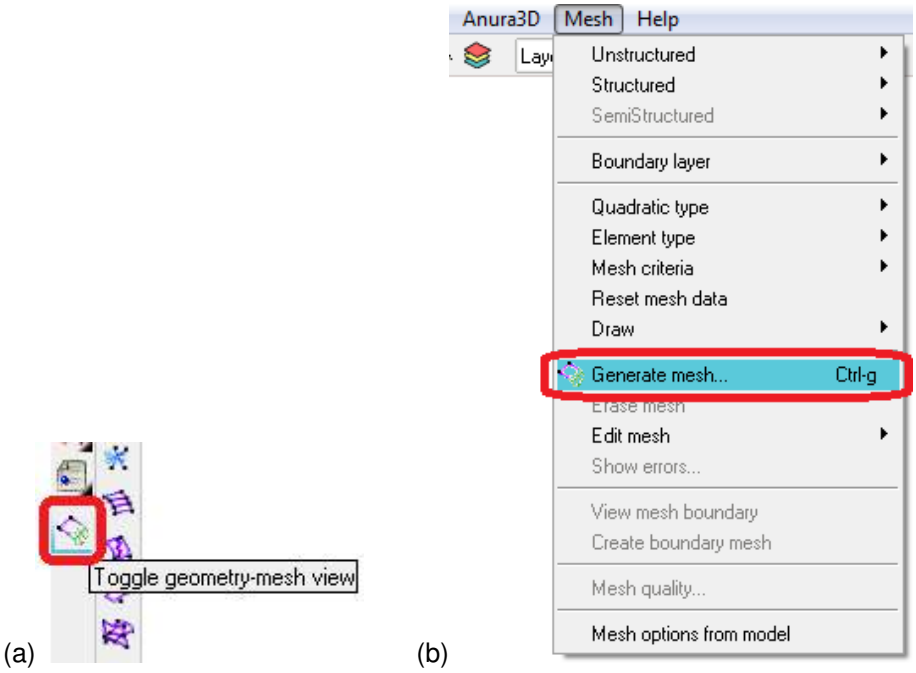


Figure 3.36: *Generating mesh in GiD: (a) sidebar button and (b) drop-down menu.*

(4) The "Mesh generation" dialogue box appears.
Set the element size to "0.1" as shown in Figure 3.37. This corresponds to the minimum side length of the soil column and therefore ensures an optimum mesh size.

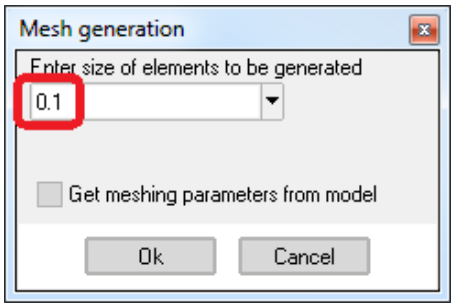
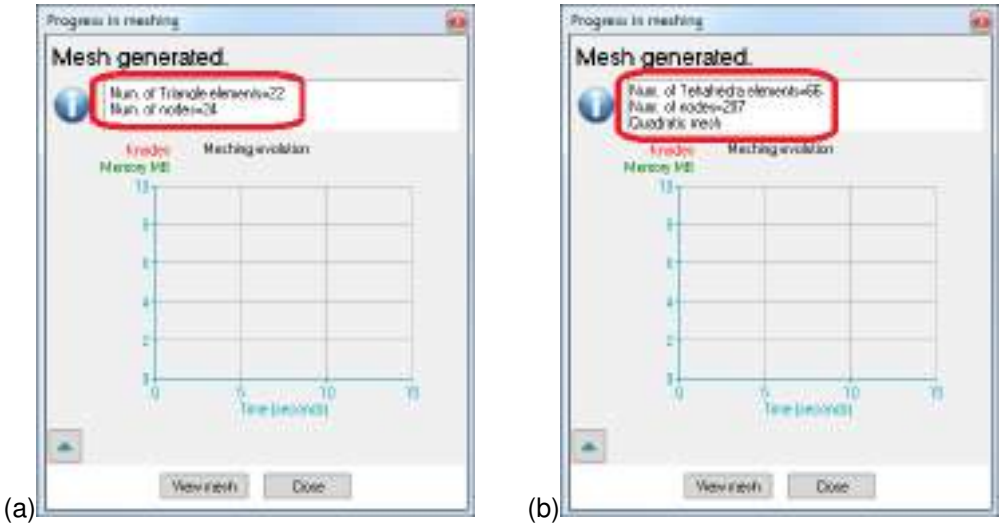


Figure 3.37: *Defining generic element size for meshing in GiD.*

(5) Click the <OK> button. The mesh will be generated and the progress is shown in the "Progress in meshing" dialogue box as shown in Figure 3.38. Note that depending on the meshing preferences the number of generated nodes and elements can change.

- ◇ 2D case: The number of generated linear triangular elements (e.g., 22) can be seen as well as the number of generated nodes (24).
- ◇ 3D case: The number of generated quadratic tetrahedal elements (e.g., 66) can be seen as well as the number of generated nodes (207).

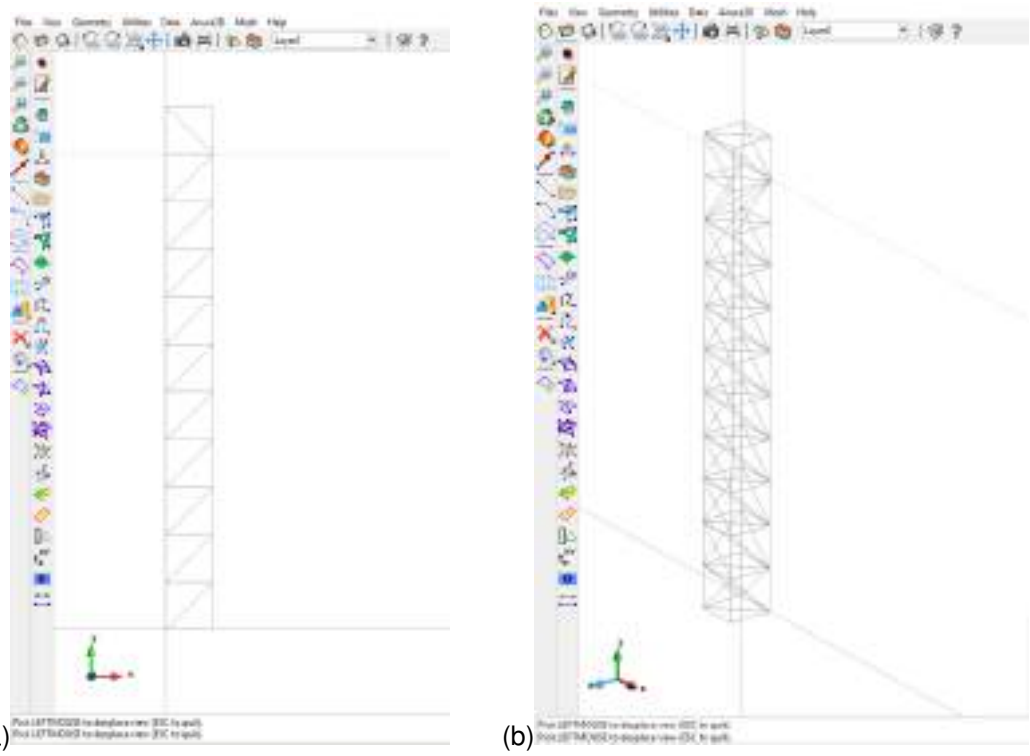


(a) (b)

Figure 3.38: *Information on meshing progress in GiD: (a) 2D case and (b) 3D case.*

(6) Click on the <View mesh> button to quit the meshing process and see the generated mesh.

The mesh with element size of 0.1 for the oedometer problem should look like as shown in Figure 3.39.



(a) (b)

Figure 3.39: *Mesh of the oedometer problem: (a) 22 triangular elements for 2D case and (b) 66 tetrahedral elements for 3D case.*

The quality of the generated mesh can easily be assessed. Choose <Mesh> <Mesh quality...> in the menu. The "Mesh quality" dialogue box appears and mesh properties can be assessed

as shown in Figure 3.40. Several quality criteria can be selected.

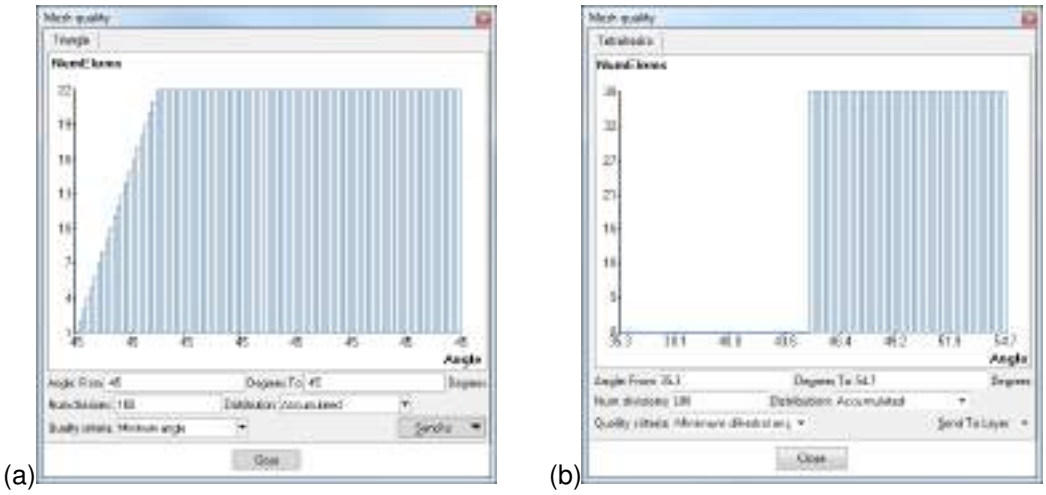


Figure 3.40: *Assessing mesh quality in GiD: (a) 2D case and (b) 3D case.*

3.2.9 Calculation data

To perform a calculation with Anura3D , several calculation parameters need to be defined. These parameters control the calculation process and features that can be enabled or disabled as e.g. contact algorithm, damping, smoothing, etc. The calculation parameters can be defined as described below.

Computation method
The computation method used in this example is the Gauss point integration for fully filled elements and material point integration for partially filled elements. Therefore, the option "MPM – mixed integration" should be selected.

Calculation step data
The compression of the soil column will be analysed for a total time of t = 2.5 s. To create intermediate results, the total time is divided into 50 calculation steps of Δt = 0.05 s each. A Courant number of 0.98 is advised to ensure the stability of the solution.

Gravity load
By default, initial stresses are zero. In this example, gravity is neglected. Therefore, select to not apply a gravity load. The values for gravity data can be kept at their default values, but are not used in the calculation anyway.

External load (solid traction)
The external load is applied suddenly, i.e. in the very first time step, on top of the soil column and is maintained constant throughout the calculation. Therefore, set the solid traction loading type as "apply solid traction – stepwise" and set the solid traction multipliers as 1.0 (initial and final value). As Load System A was selected in Section 3.2.7, be sure to select Solid Traction A, and apply the multipliers to this load system.

Default values
All other calculation options are not used in this example and the provided default values can be kept, i.e. regarding contact algorithm, smoothing algorithms, quasi-static convergence.

The following steps have to be taken in order to define the calculation parameters.

(1) Choose <Calculation Data> from the data-tree.
(2) Enter the values for the calculation parameters as given below and shown in Figure 3.41.

- ◇ Select "COMPUTATION METHOD" as "MPM – mixed integration".
- ◇ Set "number of calculation steps [-]" to "50".
- ◇ Set "time per calculation step [s]" to "0.05".
- ◇ Set "Courant number" to "0.98".
- ◇ Select "SOLID TRACTION A" as "apply solid traction – stepwise".
- ◇ Set "solid traction A multiplier initial [-]" to "1.0".
- ◇ Set "solid traction A multiplier final [-]" to "1.0".

All other calculation parameters can be kept at their default values.

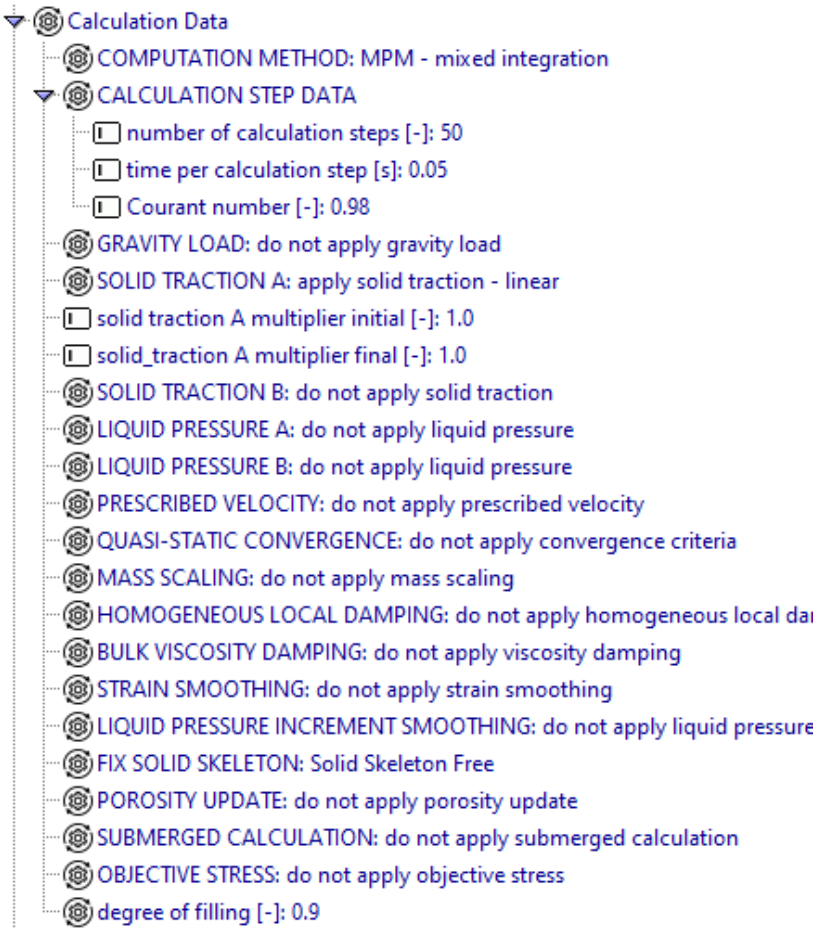


Figure 3.41: *Dialogue box to define calculation data in GiD.*

3.2.10 Results output

ParaView and GiD output

In the results output section of the data tree, the desired result variables to plot in ParaView and/or GiD, as well as the frequency of output writing, can be defined. By default all variables are plotted and the results are written at each calculation step (with the "Calculation steps between output" set to "1"). In this example, keep this value as is, and since no gas phase is present, turn off all gas-related variable by selecting "0" in the <ParaView and GiD Output → Output variables> data tree section. In Figure 3.42 gas scalar variables are deselected. The same can be applied to vector and tensor gas variables. Writing the output is one of the most computationally demanding tasks in Anura3D, so deactivating the output of unnecessary variables is always recommended.

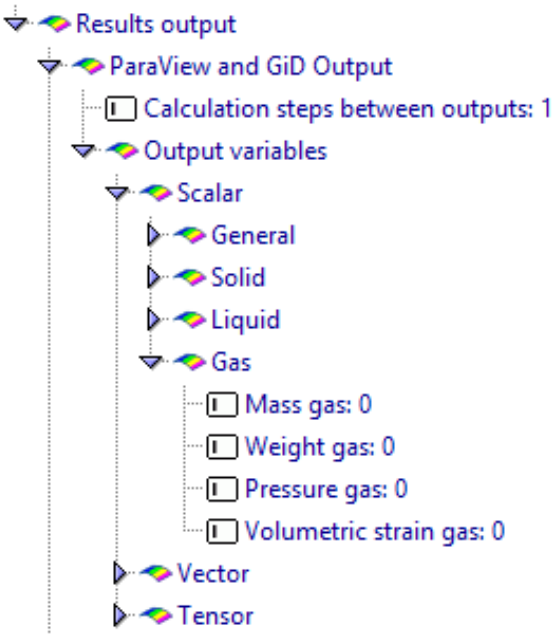


Figure 3.42: *Data tree of ParaView and GiD output options.*

Material point output

Anura3D allows the extraction of additional outputs for selected material points, which are saved in separate .PAR files, one for each material point. The output are written at specified time intervals. In this example, three material points are selected for output, located near the top, center, and bottom of the column.

In the GiD interface, users can select the elements of the computational mesh that contain, in the initial configuration, the material points of interest through the option "Select elements for MPs output" (see Figure 3.43). Notice that, in this example, three material points are initially distributed per element in the 2D case, and four in the 3D case. To output data for only one material point per selected element, set the "Number of output MPs per element" option to "1." Additionally, keep the "Time between output [s]" value to "1". Note that the selection of material Point Output can be done only in the first stage of the simulation. This option is disabled in the following stages.

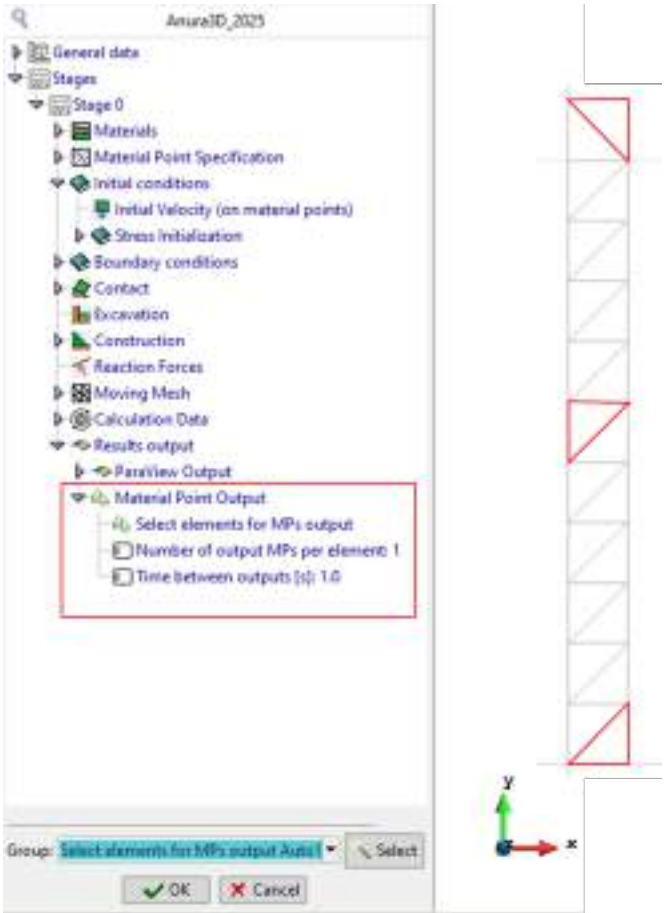


Figure 3.43: *Material point for output selection.*

It is very important to notice that elements for material point output are selected after the mesh generation. If mesh is generated again, they have to be reassigned.

3.3 Performing calculation in Anura3D

To start the calculation go to <Anura3D> and click on <Calculate>. The Anura3D calculation will start automatically

When calculation starts, a window automatically opens. It is called "view process info" window, and it gives information about which load step and stage is being computed. This window can be closed without stopping the calculation process. To reopen it, click on <Anura3D> <View process info> (Figure. 3.44).

Figure 3.44: *Open "View process info" window.*

GiD can be closed without stopping the computation. If there is a process running and GiD is closed, the message "There are 1 processes running. Terminate them?" appears. Click on "no" to continue the calculation.

Input files required for the Anura3D calculation are generated and can be found in a new folder created in the directory specified earlier by the user (Section 3.2.1, step 4), i.e. `oedometer.A3D`:

D:\..\oedometer.A3D

This folder should contain the following files:

- `D:\..\oedometer.A3D\oedometer.GOM_stage0`
- `D:\..\oedometer.A3D\oedometer.OPD_stage0`
- `D:\..\oedometer.A3D\oedometer.CPS_001`
- `D:\..\oedometer.A3D\Anura3D_2025.exe`

3.4 Visualisation of results in ParaView

The recommended way to visualise Anura3D results is by using the ParaView software (see Section 2.5 for installation instructions). ParaView is able to read output files with VTK extension. Information of the mesh and material points is stored in these files at the end of each calculation step as explained previously in Section 2.1.3.

The results for the oedometer problem are presented in this section. First, the VTK files will be opened with ParaView and the evolution of liquid pressures and solid displacements across the soil column will be presented. Finally, the results of specific material points are shown in Section 3.6.

3.4.1 Opening ParaView

(1) Start the ParaView post-processor (version 5.7.0) by clicking on the Windows desktop icon as shown in Figure 3.45.

Figure 3.45: *Start the ParaView post-processor (version 5.7.0).*

3.4.2 Loading the VTK output files

(1) Open the output files: Click the icon <Open> OR select <File> and <Open...> from the menu as shown in Figure 3.46 OR press <Ctrl>+<O>.

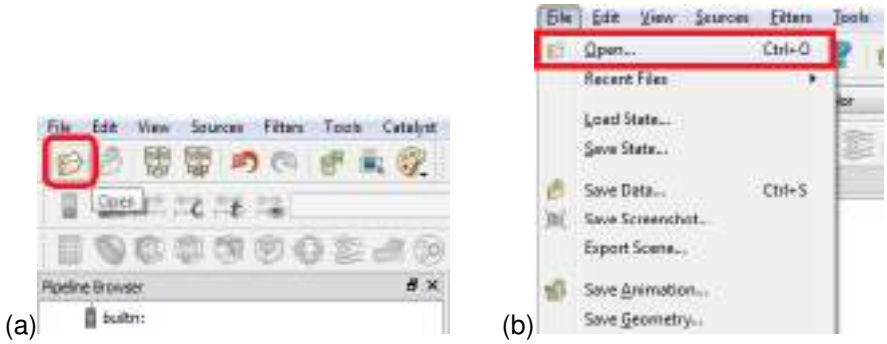


Figure 3.46: *Open the VTK output files in ParaView: (a) sidebar button or (b) drop-down menu.*

(2) Switch to the directory of your project in the "Open File" dialogue box.

(3) Select all VTK files that appear in your project directory in the "Open File" dialogue box and press <OK> as shown in Figure 3.47.

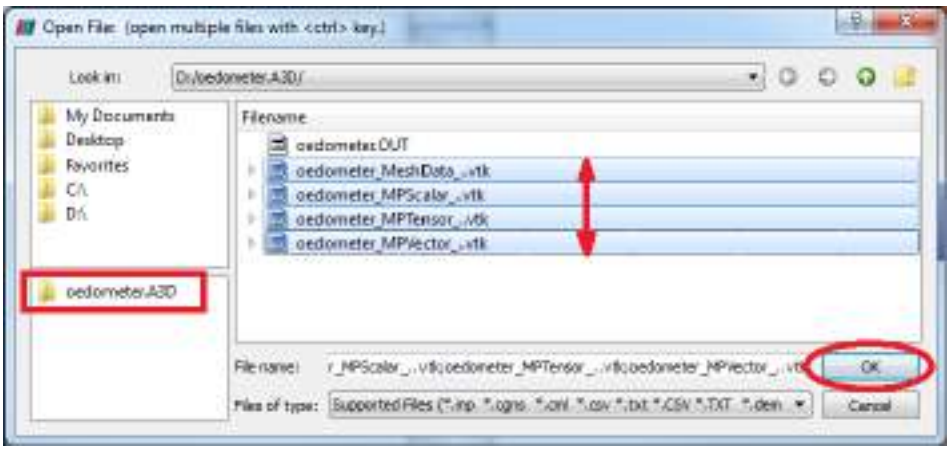


Figure 3.47: *Select the VTK otput files that appear in the project directory.*

(4) Four file directories appear in the "Pipeline Browser" window with file names as indicated in Figure 3.48.

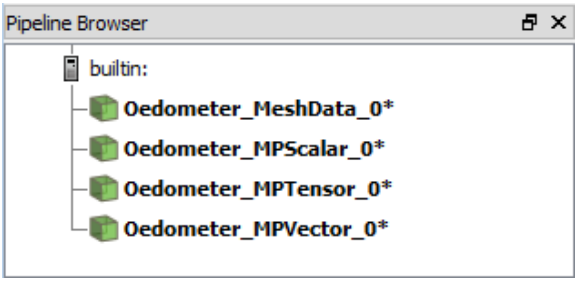


Figure 3.48: *Output VTK files in the "Pipeline Browser" window in ParaView.*

(5) In the "Properties" window click <Apply> to load all data from the VTK output files into ParaView (Figure 3.49a). Once the four file directories are loaded, note that a black symbol of an "eye" appears in front of the name of each directory, as indicated in Figure 3.49b. This means that four type of results are displayed in the "Layout #1" window.

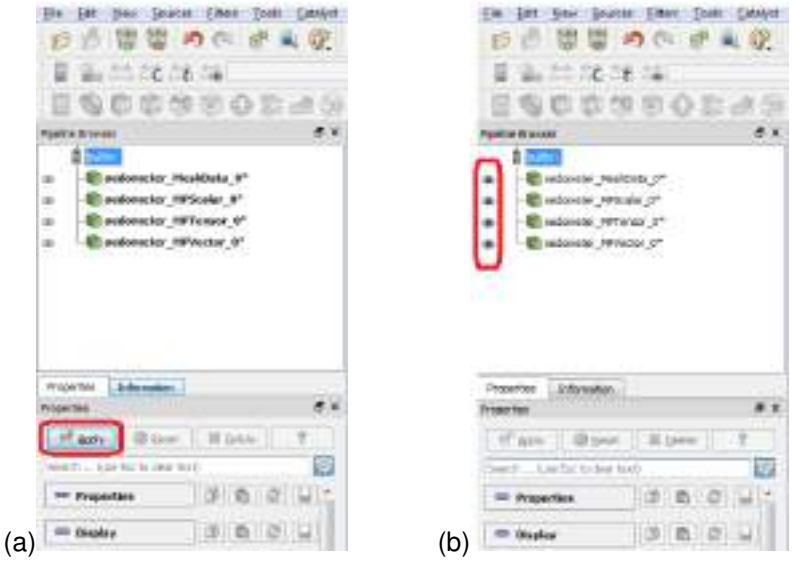


Figure 3.49: *Loading output VTK files in ParaView: (a) sidebar button or (b) drop-down menu*

3.4.3 Showing the mesh in 2D and 3D

The mesh data is stored in `oedometer_MeshData_0`.

(1) Click on the black "eye" symbols that appear in front of the last three files. The "eye" symbols change colour to grey as shown in Figure 3.50 which means that all data from material points is not displayed.

Figure 3.50: *Deactivate data in ParaView.*

(2) In order to visualise the mesh click on `oedometer_MeshData_0` in the "Pipeline Browser" window. Select "active_elements" in the first drop down menu and select "Wireframe" in the third drop down menu as shown in Figure 3.51.

Figure 3.51: *Visualisation of the mesh in ParaView.*

(3) Use the mouse wheel to zoom in and out the view of the results. Press the left mouse button to rotate the view if necessary. The buttons indicated in Figure 3.52 can also be used to change the point of view of the results.

Figure 3.52: *Buttons to change the view point of the results in ParaView.*

(4) The general view of the mesh should look like as shown in Figure 3.53.

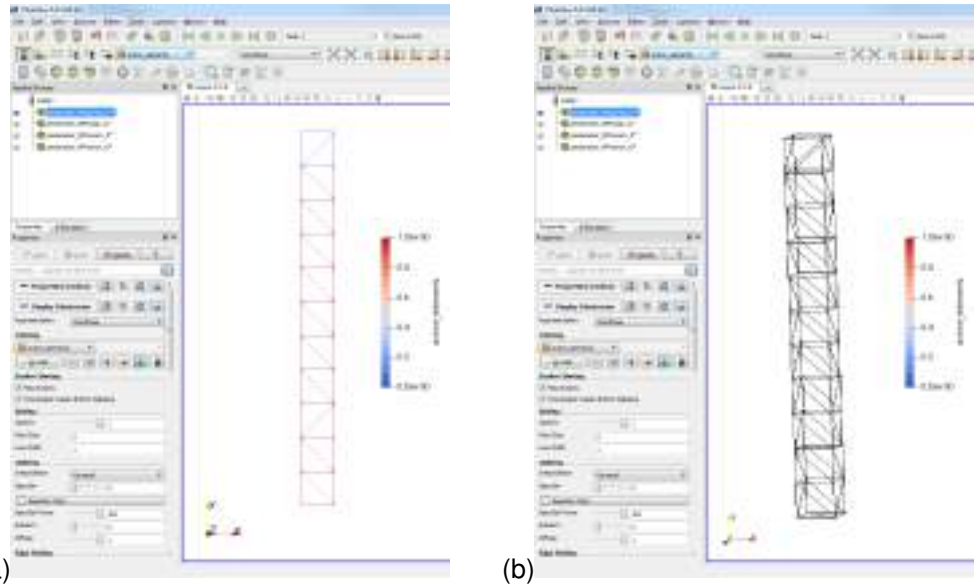


Figure 3.53: *View of the mesh, active and empty elements in ParaView for the oedometer problem: (a) 2D case and (b) 3D case.*

3.4.4 Change view settings to parallel projection (for 3D case only)

For a better visualisation of the three-dimensional model, it is advised to use the parallel projection method. To change the view settings follow the steps below. For the 2D case continue with Section 3.4.5.

(1) In the "Properties" window click on the "Toggle advanced properties" icon as shown in Figure 3.54.

Figure 3.54: *Select advanced properties to change view settings.*

(2) Scroll to the bottom of the "Properties" window.

(3) Select the option "Camera Parallel Projection" to change the view settings as shown in Figure 3.55.

Figure 3.55: *Select "Camera Parallel Projection".*

(4) Save the new settings as default view settings by clicking on the "Save current view settings values as default" icon as shown in Figure 3.55.

(5) Switch off the advanced view settings by clicking on the "Toggle advanced properties" again as shown in Figure 3.54.

This is a one-time action. After restarting ParaView, the view is by default in parallel projection.

3.4.5 Visualisation of liquid pressure in 2D and 3D

The liquid pressure is a scalar state variable stored in `oedometer_MPScalar_0`.

(1) Click on the "eye" symbol just next to `oedometer_MPScalar_0` in the "Pipeline Browser" window to display the data from this file in the window. All scalar data allocated at the material points is available in `oedometer_MPScalar_0` and the possible options are shown in the first drop down menu. Select "pressure_liquid" as indicated in Figure 3.56 to visualise the liquid pressure at the material points.

Figure 3.56: *Select "pressure_liquid" in the drop down menu.*

(2) Increase the size of the material points to 10 in the "Properties" window as indicated in Figure 3.57. Note that occasionally the "Toggle advanced properties" icon has to be clicked first to be able to see the advanced settings.

Figure 3.57: *Increase the point size of the material points.*

(3) Select <Show/hide colour legend> in the "Properties" window as indicated in Figure 3.58 in order to show the colour scale of the liquid pressure on the screen.

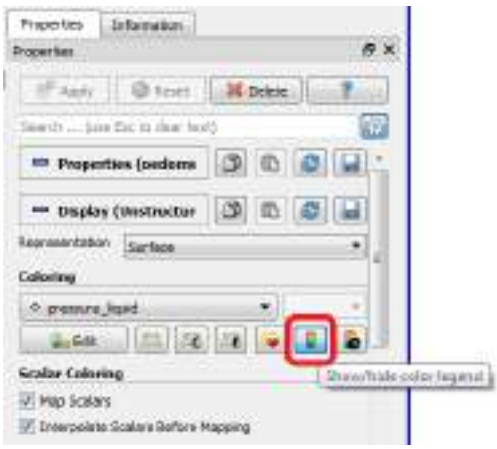


Figure 3.58: *Show the colour scale in the "Layout #1" window.*

(4) The scale can be adjusted according to the data range as indicated in Figure 3.59(a). It also can be customised manually as indicated in Figure 3.59(b).

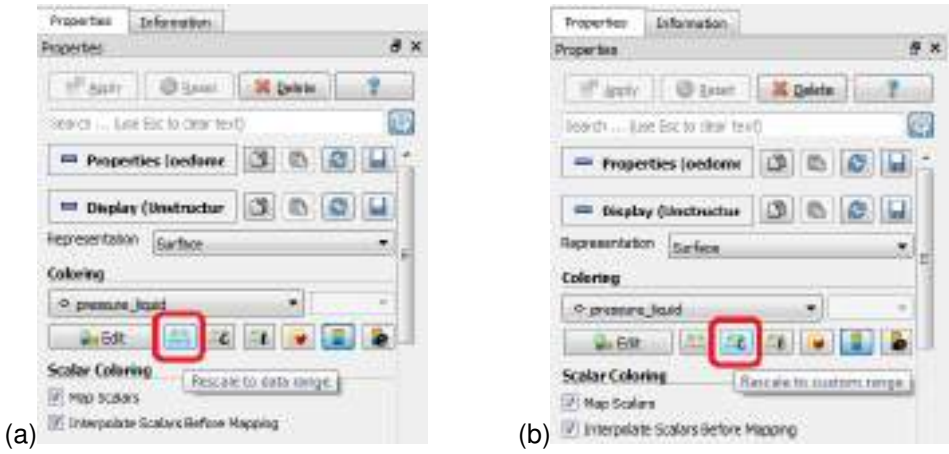


Figure 3.59: *Options to adjust the data scale: (a) rescale to data range (automatic rescaling) and (b) rescale to custom range.*

(5) For this case, the second option is preferable. In the "Set Range" window, type "-400" as minimum value and "0" as maximum value as shown in Figure 3.60. Units are in kPa.

Figure 3.60: *Adjust scale to a custom range.*

(6) Use the commands from the menu indicated in Figure 3.61 to see the animated evolution of liquid pressure. Here the first 50 steps are chosen.

Figure 3.61: *Commands to view the evolution of the results as an animation in ParaView.*

(7) In the "Animation View" menu, select the "Real Time" Mode as indicated in Figure 3.62. Here, the duration of the movie can be specified (in seconds).

Figure 3.62: *Commands to view the evolution of the results as an animation in ParaView.*

(8) The evolution of liquid pressure in the soil column is presented in in Figure 3.63 for the 2D case and in Figure 3.64 for the 3D case. Initially, the liquid pressure is zero, and shortly after increases suddenly to –400 kPa as a reaction of applying the external load on the top of the soil column. Later on, the pressure dissipates more rapidly at the top of the soil column as the top boundary is permeable, whilst the dissipation in deeper material points is slower as the bottom boundary is impervious. Finally, after 2.5 seconds of calculation, the liquid pressure along the soil column is essentially fully dissipated.

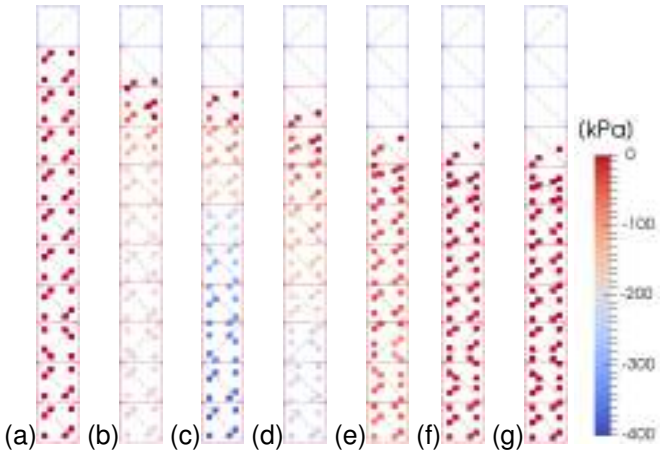


Figure 3.63: *Evolution of liquid pressure for the 2D case: (a)* t = 0 *s, (b)* 0.05 *s, (c)* 0.1 *s, (d)* 0.2 *s, (e)* 0.5 *s, (f)* 1 *s and (g)* 2.5 *s.*

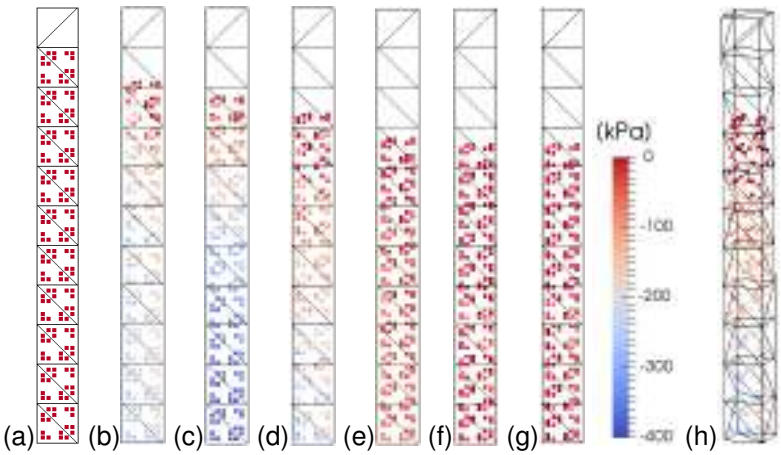


Figure 3.64: *Evolution of liquid pressure for the 3D case. (a)* t = 0 *s, (b)* 0.05 *s, (c)* 0.1 *s, (d)* 0.2 *s, (e)* 0.5 *s, (f)* 1 *s, (g)* 2.5 *s and (h)* 0.2 *s (3D view).*

3.4.6 Visualisation of soil displacement in 2D and 3D

The soil displacement is a vector variable stored in `oedometer_MPVector_0`.

(1) Click on the "eye" symbol next to `oedometer_MPScalar_0` in the "Pipeline Browser" window to hide the `MPScalar` data.

(2) Click on the "eye" symbol next to `oedometer_MPVector_0` in the "Pipeline Browser" window to display the `MPVector` data. All vector data of the material points available is shown in the first drop down menu. Select "displacement_solid" to visualize the displacements of the material points as indicated in Figure 3.65.

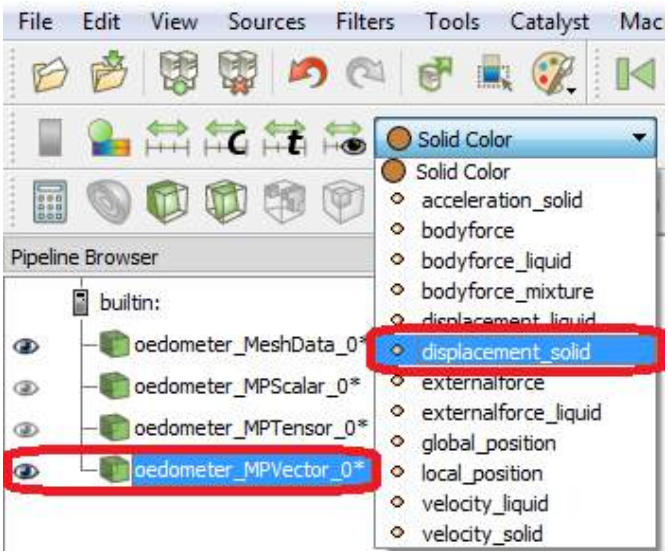


Figure 3.65: *Select "displacement_solid" in the drop down menu.*

(3) Customise the point size and the colour scale as described previously for the liquid pressure to see the evolution of soil displacements.

(4) The evolution of solid displacements is presented for the 2D case in Figure 3.66 and for the 3D case in Figure 3.67. An external load of 400 kPa is applied on top of the soil column and a consolidation process appears due to dissipation of liquid pressures. The final displacement of the top of the soil column is 0.24m. It is important to highlight that this is a large deformation problem.

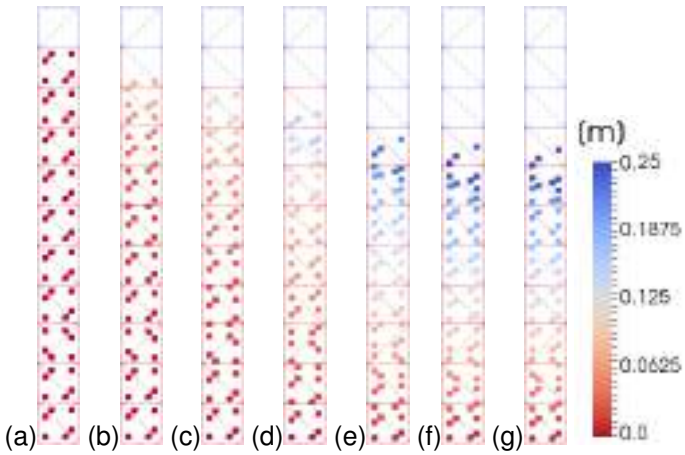


Figure 3.66: *Evolution of solid displacement for the 2D case: (a)* t = 0 *s, (b)* 0.05 *s, (c)* 0.1 *s, (d)* 0.2 *s, (e)* 0.5 *s, (f)* 1 *s and (g)* 2.5 *s.*

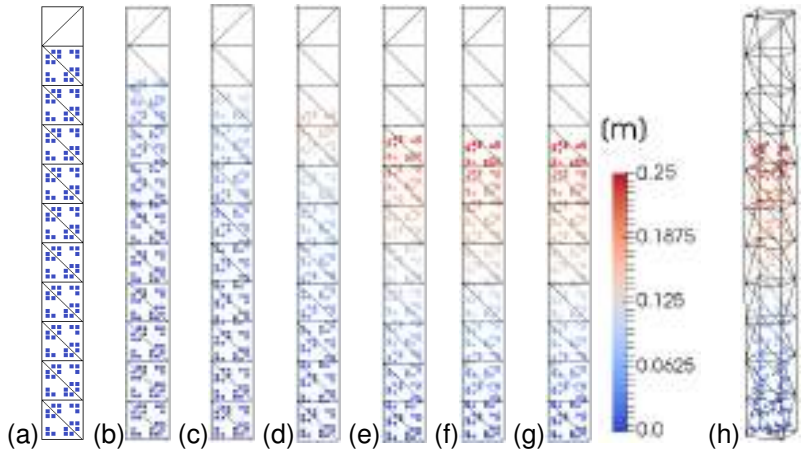


Figure 3.67: *Evolution of solid displacement for the 3D case: (a)* t = 0 *s, (b)* 0.05 *s, (c)* 0.1 *s, (d)* 0.2 *s, (e)* 0.5 *s, (f)* 1 *s, (g)* 2.5 *s and (h)* 2.5 *s (3D view).*

3.4.7 Determining the material point ID

(1) Make sure that the mesh and scalar properties are active. Click on the "eye" symbol next to `oedometer_MeshData_0` and `oedometer_MPScalar_0`. From the scalar data, select the desired variable to be visualised – e.g. "material_id" as indicated in Figure 3.68. Note that the property "material_id" is chosen for a convenient visualisation in uniform colour, any other property can be chosen as well.

Figure 3.68: *Select "material_id" in the drop down menu.*

(2) In the "Layout #1" window, click on the "Split Horizontal" icon as indicated in Figure 3.69 to add an additional output window in which the material point data will be shown.

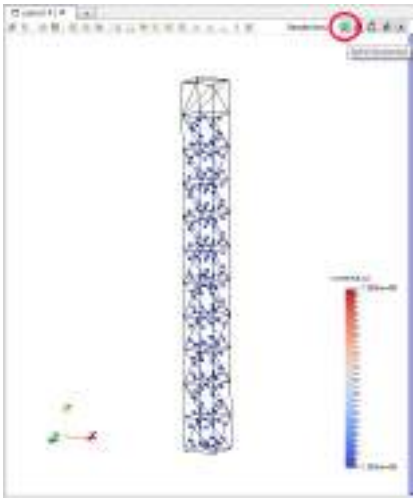


Figure 3.69: *Creating additional output window by splitting view.*

(3) In the new window, click on "SpreadSheet View" button as indicated in Figure 3.70 to show the spreadsheet.

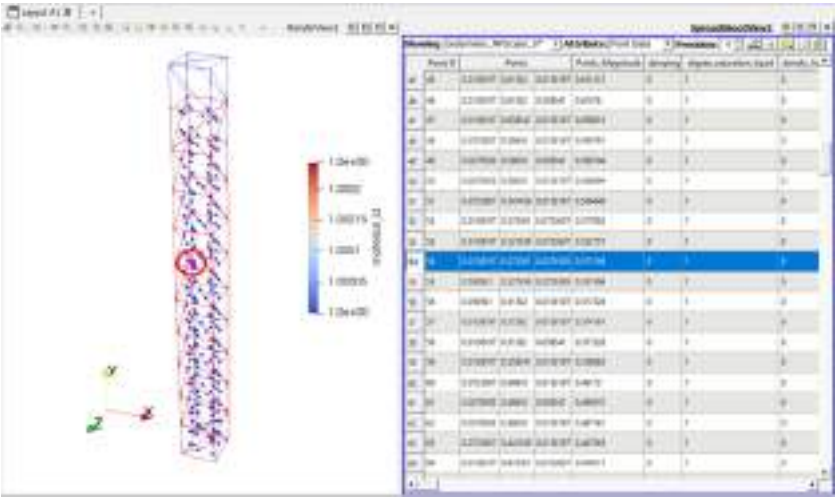


Figure 3.70: *Choose "SpreadSheet View" in the new output window.*

(4) In the spreadsheet click, on a material point number and the respective point will be highlight in the geometry as indicated in Figure 3.71.

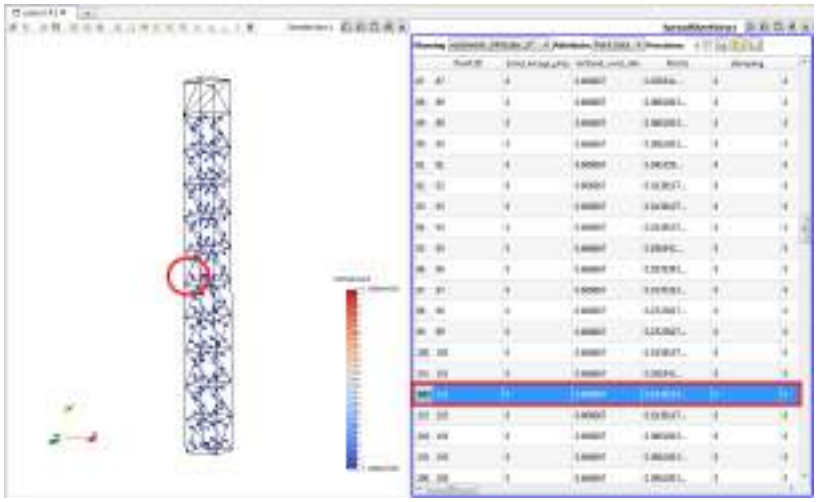


Figure 3.71: *Highlight specific material point in geometry.*

(5) To assess the ID of a particular material point from the geometry click on the "Select Points On" icon in the "Layout #1" window as indicated in Figure 3.72(a). Then click on the material point, which should be assessed as shown in Figure 3.72(b). Repeat this action for three points at the top, middle and bottom of the soil column. Note: Keep the <Ctrl>-key pressed to select multiple points.

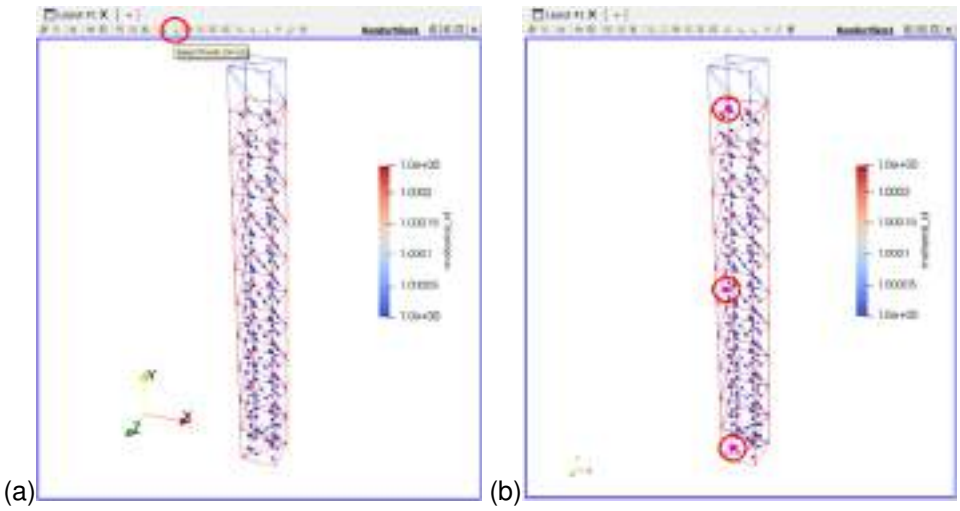


Figure 3.72: *Select specific material points in geometry: (a) sidebar button or (b) drop-down menu.*

(6) In the "SpreadSheet View" window, click on the "Show only selected elements" icons as shown in Figure 3.73. The IDs of the three selected points can be seen as well as all other information stored in the selected points.

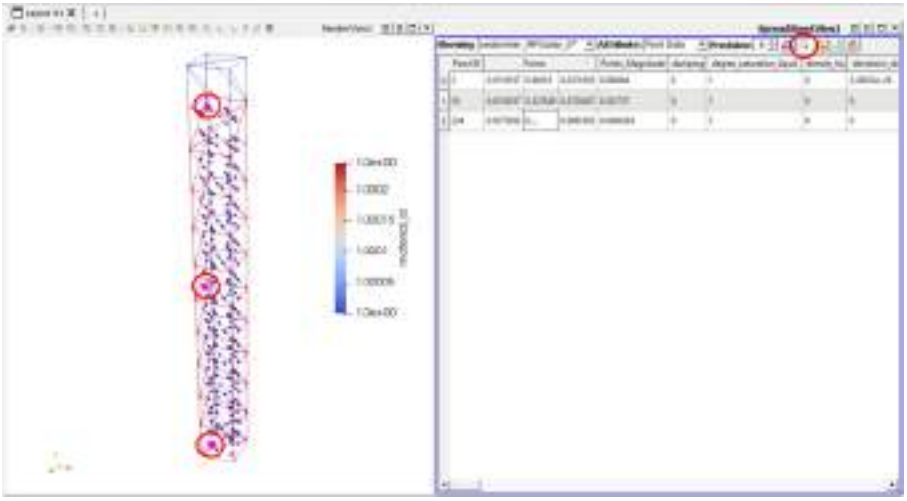


Figure 3.73: *Show selected material points in spreadsheet.*

3.4.8 Plotting time graphs for material points

In ParaView, several options for plotting time graphs are available. The process is explained for the 3D case but can equivalently be adopted for the 2D case. Three alternatives are explained below:

- ◇ Plot point over time;
- ◇ Plot two variables at same time;
- ◇ Plot multiple points.

Plot point over time

As an example, the liquid pressure in one material point at the bottom of the soil column will be plotted over time. The following steps should be taken.

(1) Make sure that the VTK files containing the data are loaded into the "Pipeline Browser". Make sure that you have clicked on the <Apply> button in the "Properties" window. Make sure that the viewing of data is turned on (active "eye"-symbol) as shown in Figure 3.74.

Figure 3.74: *Turn on the "pressure_liquid" data for the oedometer problem in ParaView.*

(2) First, select a point at the bottom of the geometry for which the data should be plotted. See Section 3.4.7 on how to select a specific material point. Click on "Select Points On"

icon and, then, click on a point at the bottom of the soil column.

(3) Choose <Filters> <Data Analysis> <Plot Selection Over Time> from the menu as shown in Figure 3.75.

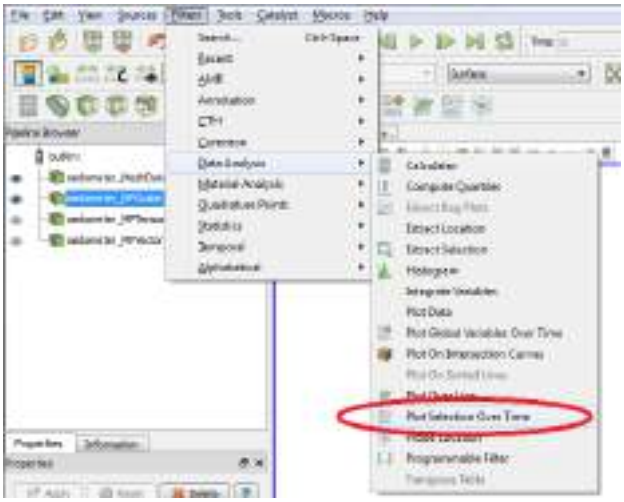


Figure 3.75: *Choose "Plot Selection over Time" in ParaView.*

(4) Click on the "Apply" button in the "Properties" window as shown in Figure 3.76 to activate the graphs window, which will then appear right next to the existing plot. Note that the material point ID can be retrieved in the selection window; that is "234" for the current case.

Figure 3.76: *Apply the graph plotting for the selected point.*

(5) In the "Properties" window, scroll down to the "Display" tab. Unselect everything except "pressure_liquid" as shown in Figure 3.77.

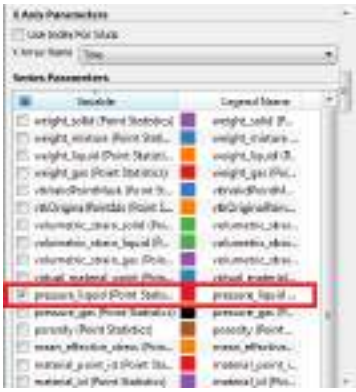


Figure 3.77: *Choose plotting of "pressure_liquid" for the selected point.*

(6) A plot of the liquid pressure at the selected point is created as shown in Figure 3.78. On the horizontal axis the time step number is shown. On the vertical axis the liquid pressure in [kPa] is shown.

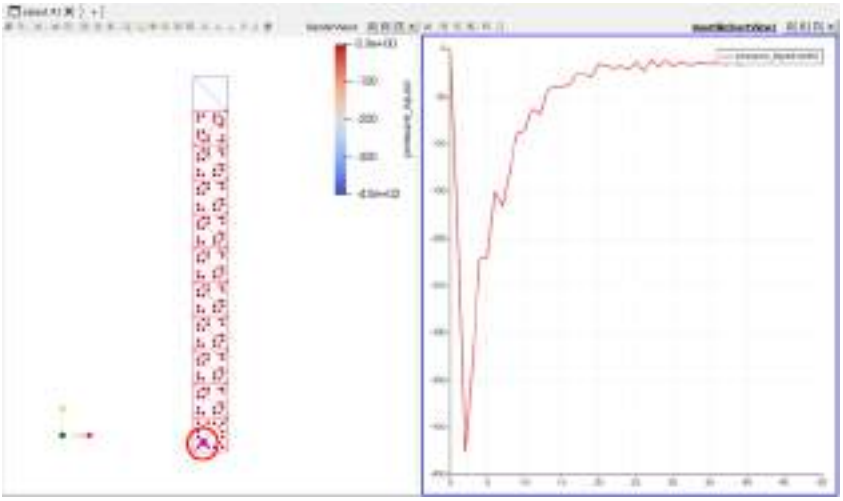


Figure 3.78: *Plot of liquid pressure over time for the oedometer example (3D case).*

The layout and appearance of the chart can be adapted according to the needs of the user by changing the respective parameters in the "Properties" window. Press the play button to visualise the progress in the time graph.

Plot two variables at same time

As an example, the liquid pressure and the solid volumetric strain in one material point at the bottom of the soil column will be plotted over time. The following steps should be taken.

(1) Follow the steps for "Plot Selection Over Time", as described above.
(2) In the "Properties" window, turn on the additional variable "volumetric_strain_solid".
(3) Select the line that says "volumetric_strain_solid".
(4) In the "Properties" window scroll further down. At the bottom of the "Display" tab, change "Chart Axes" to "BottomRight".
(5) A plot of the liquid pressure and the solid volumetric strain at the selected point is created as shown in Figure 3.79.

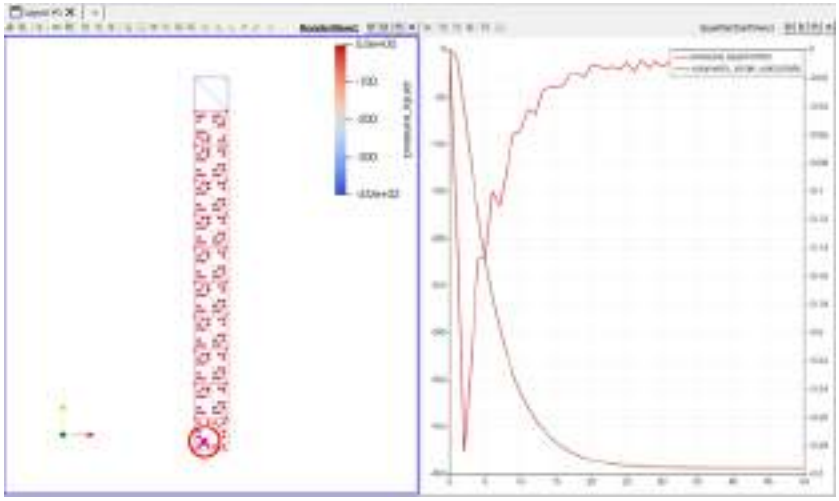


Figure 3.79: *Plot of liquid pressure and solid volumetric strain over time for the oedometer example (3D case).*

Plot multiple points

As an example the liquid pressure for three points at the top, middle and bottom of the soil column will be plotted over time. The following steps have to be taken.

(1) Make sure that the VTK files containing the data are loaded into the "Pipeline Browser", that you have clicked on the <Apply> button in the "Properties" window, and that the viewing of data is turned on (i.e. active "eye"-symbol) as shown in Figure 3.74.
(2) To plot numerous points over time select the points by click on "Select Points On" icon. Then, select the points that should be plotted; these are the top, middle and bottom of the soil column. Keep the <Ctrl> key pressed while selecting the additional point.
(3) Choose <Filters> <Data Analysis> <Plot Selection Over Time> from the menu as shown in Figure 3.75.
(4) Click on the "Apply" button in the "Properties" window to activate the graphs window, which will then appear right next to the existing plot.
(5) In the "Properties" window, unselect the "Only Report Selection Statistics" check box. Click on "Apply" button.
(6) In the "Properties" window, display section, click on "Root" to switch on all three points.
(7) In the "Properties" window, scroll down to the "Display" tab. Unselect everything except "pressure_liquid" for all three points.
(8) A plot of the liquid pressure at the selected points is created as shown in Figure 3.80.

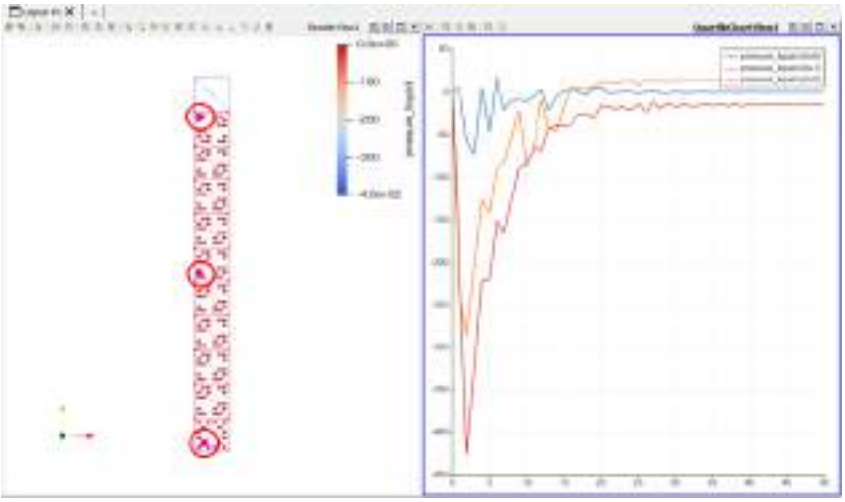


Figure 3.80: *Plot of liquid pressure for multiple points over time for the oedometer example (3D case).*

Multiple variables can be plotted for multiple points in the same way as described above. The layout and appearance of the chart can be adapted according to the needs of the user by changing the respective parameters in the "Properties" window.

3.5 Visualization of results in GiD

An alternative to visualizing results through ParaView is to use GiD. This GiD post-processor is automatically installed when you install the GiD pre-processor as described in Section 2.4. GiD is able to read output files in ASCII and Binary format. It is always recommended to work with Binary files.

Reading Anura3D results with GiD has some limitations. Only the contour fields of all variables (scalar, vector, and tensor) can be displayed. For vector variables, the magnitude and direction of the vectors are available. All the other options in the GiD post-process are not supported for Anura3D.

Some results of the oedometer problem are presented in this section. First, Binary output files will be opened with GiD and the evolution of liquid pressure across the soil column will be presented.

3.5.1 Opening GiD post-processor

Once the calculation is finished, click on the pre/post-processor switching icon located in the upper-right corner to access the post-processor interface (Figure 3.81). All results are loaded automatically.

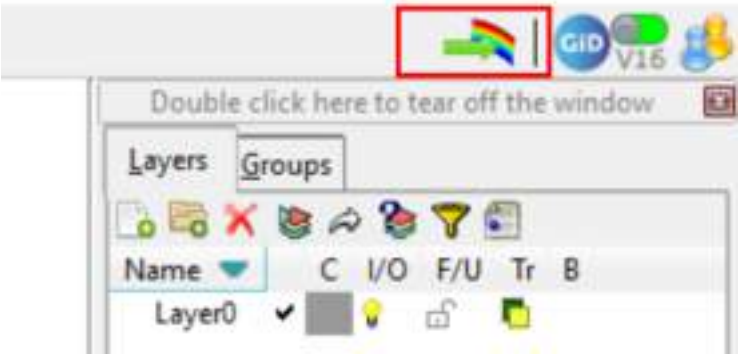


Figure 3.81: *Button to switch from pre- to post-processor.*

An alternative approach to this procedure is described in the following section.

3.5.2 Loading results in GiD post-processor from Binary files

To load results from a different case that the one currently opened in the pre-processor, click on the switching icon in the upper-right corner to access the post-processor interface and follow these steps:

(1) Click the icon <Open> (Figure 3.82)

(a)

Figure 3.82: *Open results in GiD post-processor with sidebar button.*

(2) In the “Postprocess read several files” dialogue box, navigate to the directory of your project and select the “oedometer.POST.lst” file. The “.POST.lst” file opens the results for all the load steps. The option “Read every X steps” is useful if results of a lower number of load steps are needed (Figure 3.83).

(3) Press <Open> as shown in Figure 3.83.

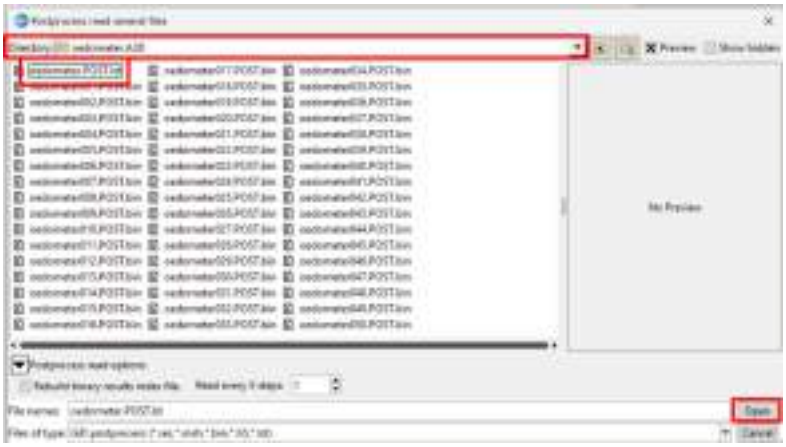


Figure 3.83: *"Postprocess read several files" window for Binary format results.*

3.5.3 Mesh and MPs visualization

The visualization of mesh and MPs is activated/deactivated in the "Style" window, which automatically appears in the right hand-side. To visualize the mesh contour and correctly visualize material points, set the option in the style column as indicated in figure (Figure 3.84).

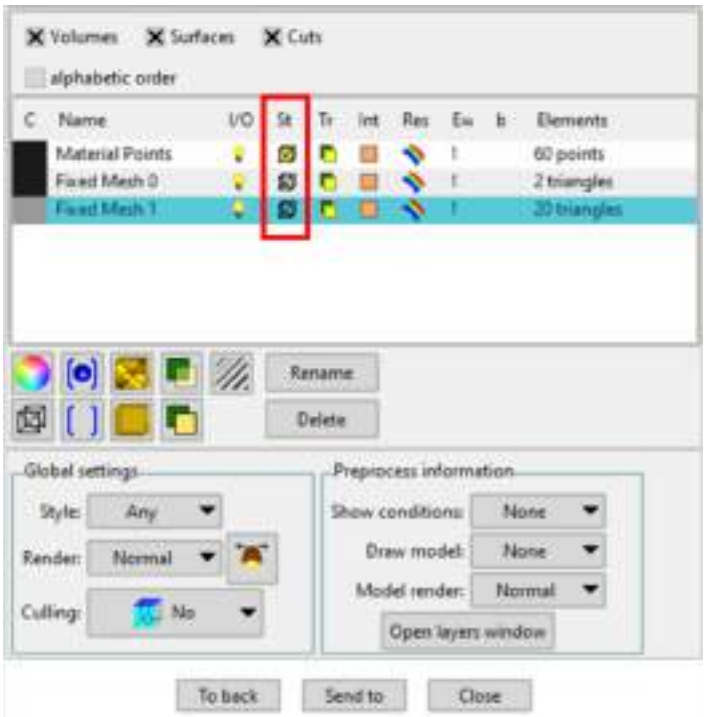


Figure 3.84: *"Style" window with the configuration to see MPs and mesh as contour.*

3.5.4 Change view settings

The view settings of the GiD post-process works exactly as the GiD pre-process (Figure 3.19).

3.5.5 Visualisation of liquid pressure in 2D and 3D

Go to Default Analysis/Step in the results sidebar (Figure 3.85a) and select load step 1 of the scalar results.

Click on the Contour fill icon in the results sidebar (Figure 3.85b) and select <Pressure> and <Liquid>.

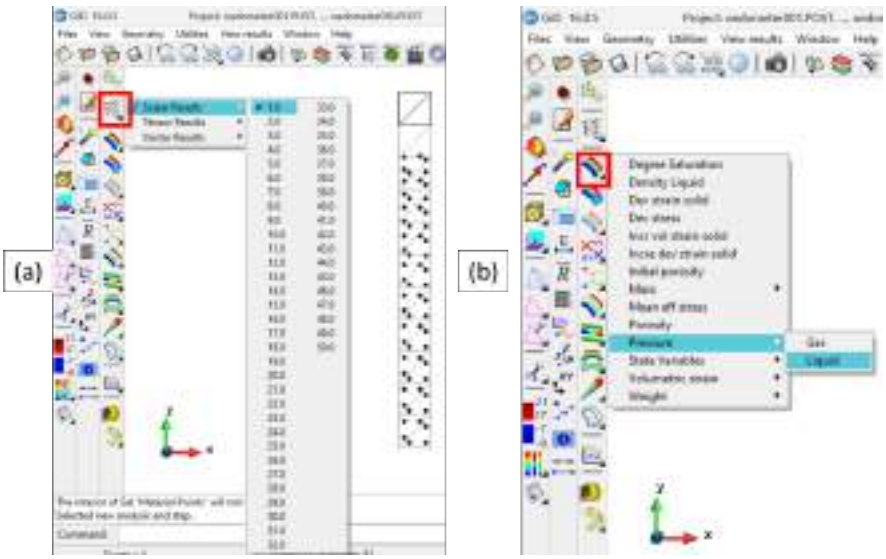


Figure 3.85: *Select result type, load step and variable to visualize*

Scale can be adjusted according to the data range. In this case the maximum is set to 0kPa and the minimum to -400kPa (Figure 3.86).

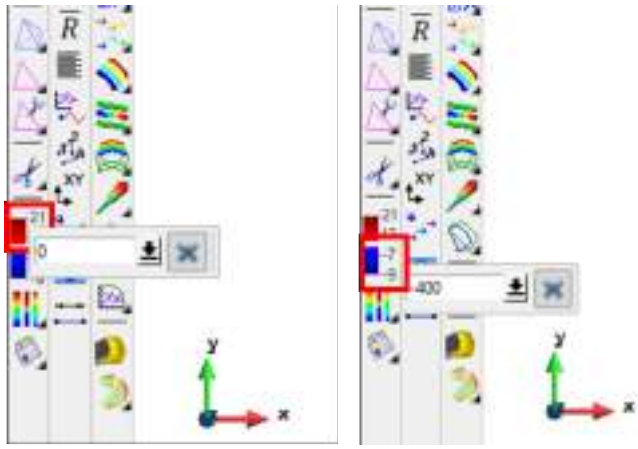


Figure 3.86: *Variable range adjustment.*

Open the animate window icon and select “Total time” in Duration and move through load steps in the play section (Figure 3.87).

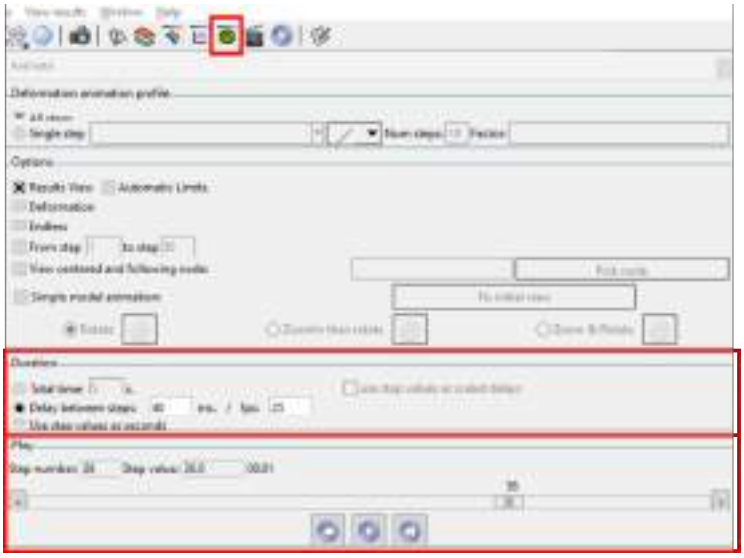


Figure 3.87: *Animation window*

The evolution of liquid pressure in the soil column is shown in Figure 3.88 for the 2D case and in Figure 3.89 for the 3D case.

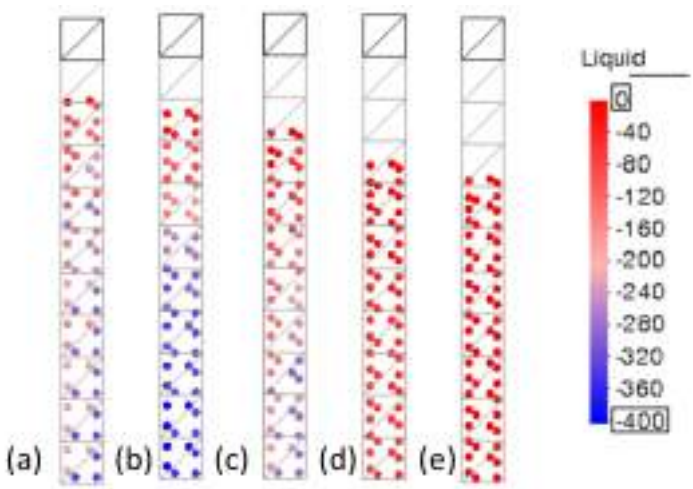


Figure 3.88: *Evolution of liquid pressure for the 2D case: (a) 0.05 s, (b) 0.1 s,(c) 0.2 s, (d) 0.5 s and (e) 2.5 s.*

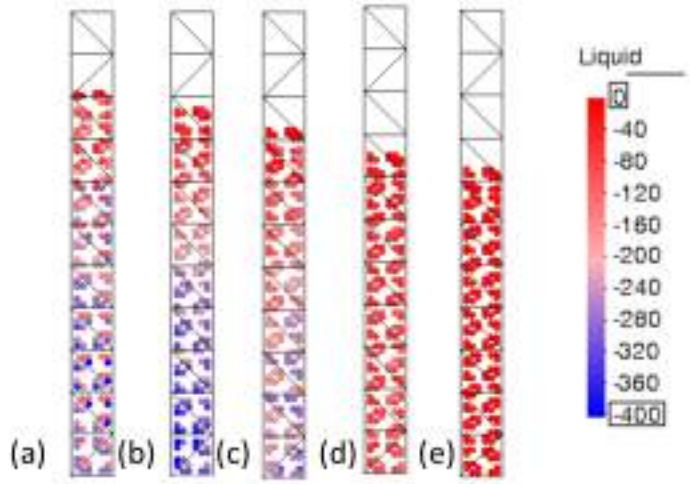


Figure 3.89: *Evolution of liquid pressure for the 2D case: (a) 0.05 s, (b) 0.1 s,(c) 0.2 s, (d) 0.5 s and (e) 2.5 s.*

3.5.6 Determining the material point ID

The material point ID is not stored in the scalar results as in Paraview. To find the ID of specific material point with GiD post-process click with the right button and go to <Label>, <Select on>, <Nodes> (Figure 3.90). After that, enter <Ctrl+A> command and select the desired MPs.

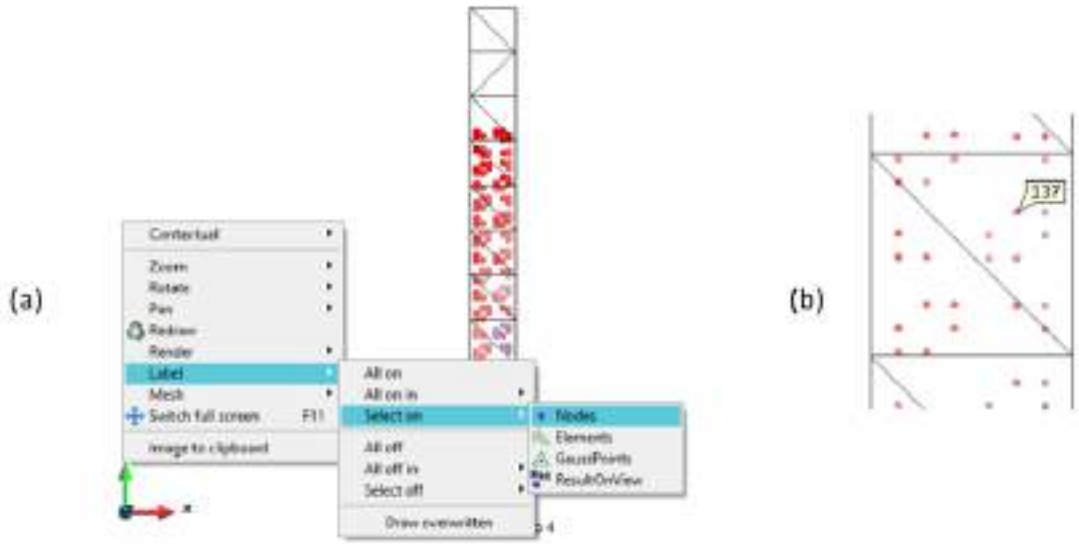


Figure 3.90: *(a) Procedure to determine MP ID (b) Label indicating the MP ID.*

3.5.7 Plotting time graphs for material points

In GiD, several options for plotting time graphs are available. The process is explained for the 2D case but can be equivalently adopted for the 3D case. Two alternatives are described below: plot point over time and plot multyple variables in the same graph.

Plot point over time

Select scalar results and any time step as explained in section Section 3.4.5. Click on the graph icon (Figure 3.91) and all the possible options open. Select plot data over time.

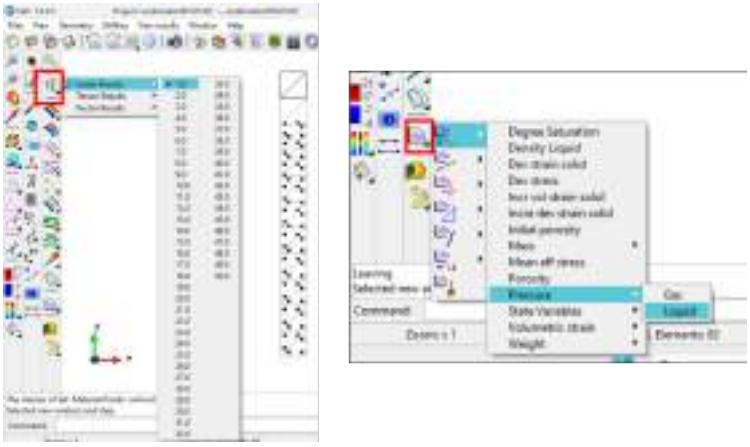


Figure 3.91: *Plot liquid pressure over time procedure.*

Enter <Ctrl+A> command, select one or more material points and press <Esc>. The graph in figure 3.92 displays the liquid pressure over time of a material point at the bottom of the column.

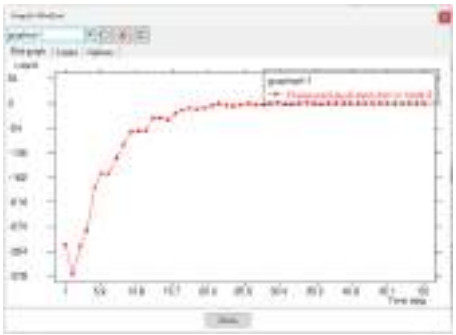


Figure 3.92: *Liquid pressure over time of a MP located at the bottom of the column.*

To export the graph data to an Excel worksheet, go to <Options>, select the desired graph set and click on <show table>. The graph values are displayed and can be copied to a worksheet (Figure 3.93).

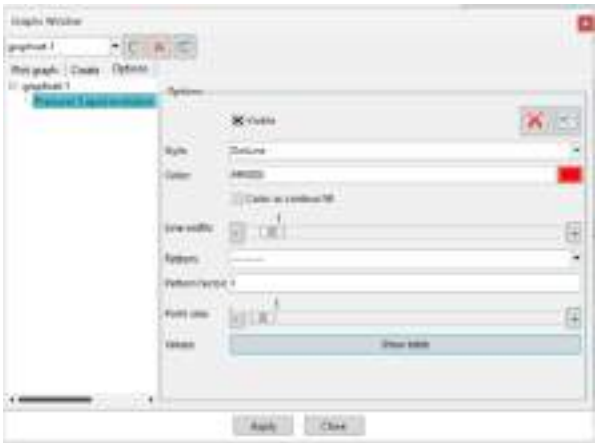


Figure 3.93: *Graph window for showing values in table.*

Plot multiple variables in a single graph

The GiD option to graph variables allows to add new graphs to the selected dataset. In order to graph in a different graph set than the previous one, a new graph set has to be created, as shown in figure 3.94.

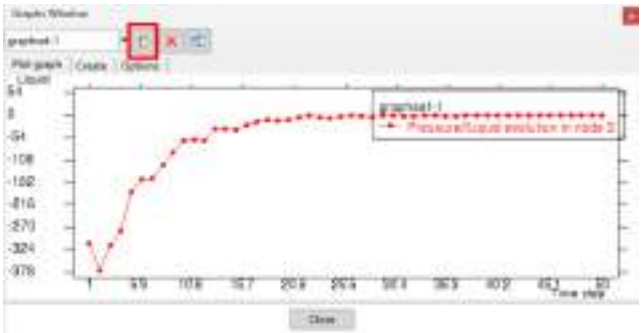


Figure 3.94: *Add a graph set.*

3.6 Visualisation of text output files

The output data for a material point located within each element selected in the "Results output" section of the data tree (as described in Section 3.2.10), and stored in the the file OPD_stage0 under the flag `$$ELEMENTS_FOR_MP_OUTPUT`. This data is written to a set of text files with extension .PAR_X, where "X" represents the ID number of the material point. Each of these files contains information for a single material point, such as the time step, load multipliers, coordinates, displacement, stress, strain and liquid pressure. These files can be opened with any text editor. The process is explained for the 3D case but can be applied similarly to the 2D case. For the 3D case, the following three files will appear in the project directory. Particle ID number can change slightly depending on the selected elements.

- ◇ `oedometer.000.PAR_125`;
- ◇ `oedometer.000.PAR_189`;
- ◇ `oedometer.000.PAR_237`.

In order to process this data any editor can be used. For example, Figure 3.95 has been created using Microsoft Excel. It shows the liquid pressure evolution of material point 54 (approximately in the middle of the soil column). It can be seen that the results have some oscillations but their magnitude decreases as the calculation proceeds. One reason for these oscillations is that Anura3D considers a fully dynamic formulation and all inertial effects play an important role in the results. Another cause for oscillations is due to the effect of material points crossing element boundaries.

Note: Before importing the PAR files in Excel, please make sure that the delimiter is set to a space " ".

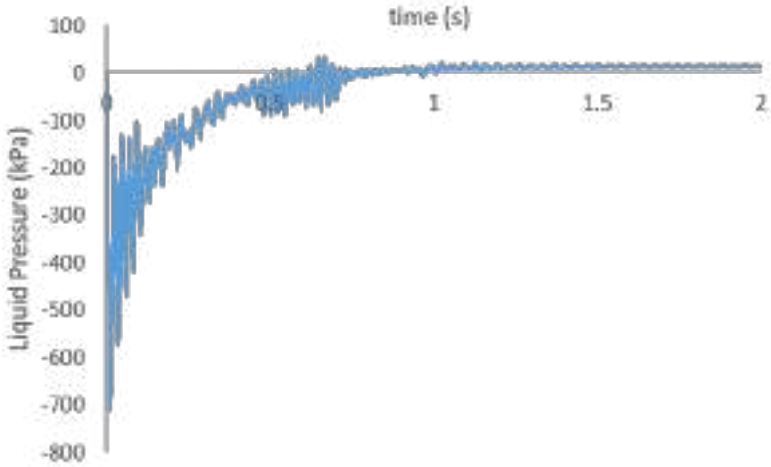


Figure 3.95: *Dissipation of liquid pressure at material point 54 (located at a height of* 0.5276 *m).*

3.7 Variations

The variations proposed for the oedometer problem are listed below.

(1) Examine the influence of homogeneous local damping, and compare the results with the case of using 10% and 75% of local damping.
(2) Solve a small deformation problem by decreasing the value of the external load to 10 kPa, and compare the results with the analytical solution.
(3) Examine the effects of using Bulk Viscosity to damp the highest frequencies of the elements and smooth numerical oscillations.
(4) Examine the effects of using Absorbing Boundaries to reduce the reflections of the propagating waves at the bottom boundary.

The changes can be implemented either by modifying the input data through the GiD preprocessor or by directly editing the input files (i.e., the GOM_stage0 and CPS files). The CPS file, `oedometer.CPS_001`, and the GOM_stage0 file, `oedometer.GOM_stage0`, are located in the project folder `oedometer.A3D`, in the directiry that was previously specified by the user (see Section 3.2.1, step 5):

```
D:\..\oedometer.A3D\oedometer.CPS_001
```

```
D:\..\oedometer.A3D\oedometer.GOM_stage0
```

The CPS and GOM files can be opened and adapted in any text editor.

3.7.1 Variation: Effect of local damping

Engineering problems in which inertia plays an important role require a dynamic formulation. However, some problems involve very slow processes for which a steady-state solution is of interest rather than the dynamic analysis of wave propagation. In order to converge to the quasi-static equilibrium as fast as possible, or to reduce numerical instabilities, Anura3D includes an additional local damping term in the momentum balance equations that introduces

energy dissipation in solid and liquid phases. More information can be found in Chapter 6 of Fern et al. [1] or in Al-Kafaji [2], Yerro [3].

Two additional calculations are proposed in order to investigate the effect of including local damping. To do so, it is not required to repeat all steps from the beginning since only one calculation parameter has to be changed. This variation can be for the 2D or the 3D case. Follow the steps described below.

Calculation with 10% **local damping**
Reopen the previously generated project in GiD and save it with a new appropriate name, e.g. `oedometer_var1_0.1`. Next, adjust the calculation data by modifying the homogeneous local damping value as shown in Figure 3.96. Compute the new case by selecting <Anura3D → Calculate> from the menu. The calculation is performed in the new directory `oedometer_var1_0.1.A3D`Finally, analyze the results.

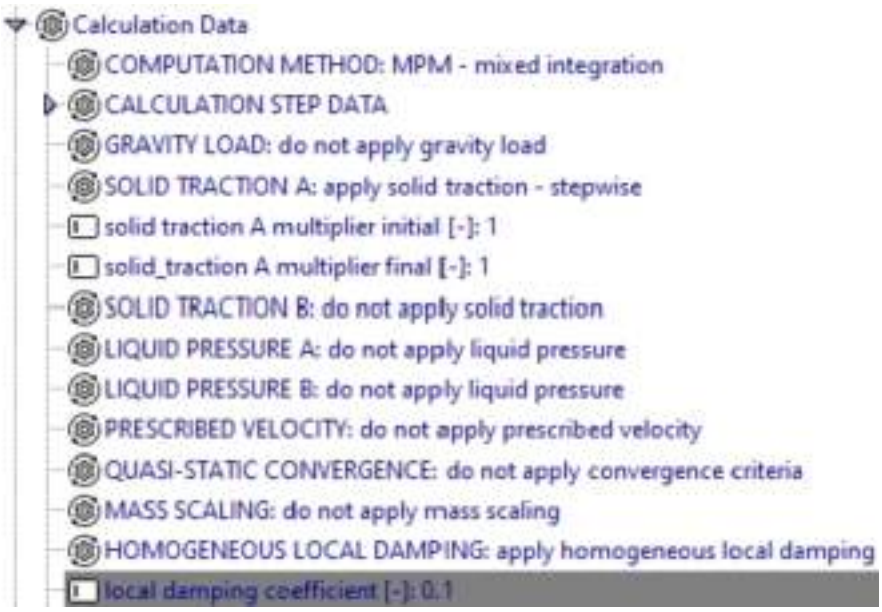


Figure 3.96: *Adjusting homogeneous local damping value in GiD.*

Alternatively:

(1) Create a new folder in your user directory with a new appropriate name, e.g. `oedometer_var1_0.1.A3D`:

```
D:\..\oedometer_var1_0.1.A3D>
```

(2) Copy the CPS_001 file, the GOM_stage0 file, the OPD_stage0 file and the Anura3D executable from the original project directory (`\oedometer.A3D`) to the new one.

(3) The local damping coefficient is the calculation parameter that controls the amount of local damping imposed to the system. It is defined in the CPS file. Open the CPS_001 file with any text editor. Modify the local damping coefficient as indicated below. Note that the first position indicates if the local damping is switched on (1) or off (0), and the second position corresponds to the selected value of the local damping coefficient.

```
$$HOMOGENEOUS_LOCAL_DAMPING
 1 0.1
```

(4) Save the modified CPS file.

(5) Launch the calculation from the command line by following these steps:

- ◇ In Windows start the "Command Line" window. Click on the "Command Prompt" desktop icon as shown in Figure 3.97a OR from the Windows Start Menu select "Run...", type "cmd" and press <ENTER> as shown in Figure 3.97b.

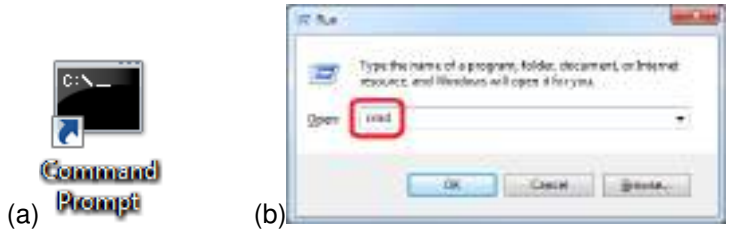


Figure 3.97: *Start "Command Prompt" in Windows.*

- ◇ Change working location in the project directory. At the command prompt, type the name of the disk drive where you stored the project data. For example, "`D:`" and press <ENTER>.

```
C:\...\user\username\>D:
```

- ◇ Change into your previously chosen directory by typing "`cd userdirectory`" and press <ENTER>.

```
D:\>cd userdirectory
```

- ◇ Change into the project directory by typing "`cd oedometer.A3D`" and press <ENTER>.

```
D:\userdirectory\>cd oedometer.A3D
```

- ◇ Run the calculation in the project directory by typing the name of the programme executable "`Anura3D_2025`" followed by the name of the calculation project, "`oedometer`", in this example.

```
D:\userdirectory\oedometer.A3D\>Anura3D_2025 oedometer
```

The calculation starts and output information is generated in the "Command Line" window indicating the progress of the calculation. Any possible input and calculation errors are displayed in this window. The output files are generated and stored in the project directory `D:\..\oedometer.A3D`.

(6) Inspect the results as described in Section 3.4.

Calculation with 75% **local damping**

Follow the steps outlined above, adjusting the local damping coefficient to 0.75.

To proceed, reopen the reference project in GiD. Save it with a new appropriate name, e.g. `oedometer_var1_0.75`, and update the local damping coefficient in the "Calculation Data" menu. Alternatively, create a new folder in your user directory with a new appropriate name, e.g. `oedometer_var1_0.75.A3D`, and manually edit the local damping coefficient directly in the CPS file.

Results

Figure 3.98 shows the effect of including local damping. Note that a high value of local damp-

ing (e.g. 75%) reduces considerably the oscillations but the dissipation of water pressure occurs more slowly since the system is excessively damped.

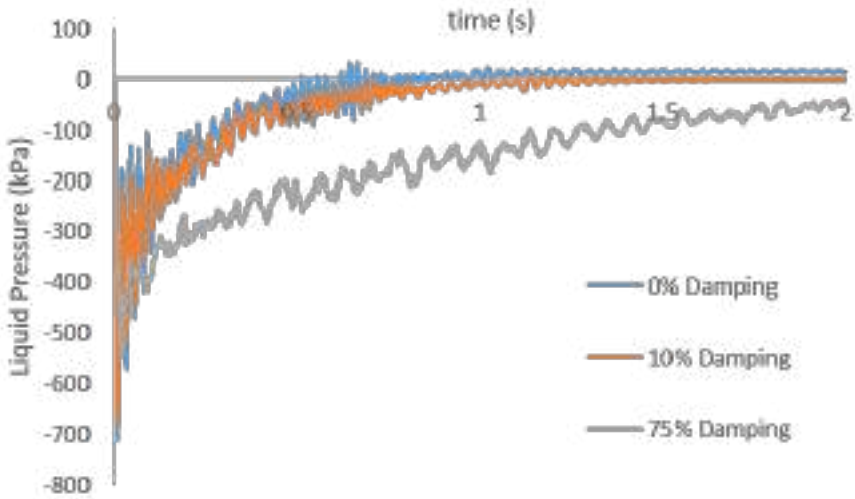


Figure 3.98: *Dissipation of liquid pressure at material point 54 (located at a height of* 0.5276m*) for different local damping coefficients.*

3.7.2 Variation: Small deformation problem

The second variation of the oedometer problem is to decrease the external load that is applied on top of the soil column to 10 kPa in order to solve a small deformation problem to be comparable to analytical solutions. This variation can be for the 2D or the 3D case. Follow the steps described below.

To proceed, reopen the reference project in GiD and save it with a new appropriate name, e.g. `oedometer_var2_10`. Next, decrease the applied load. The assigned value is set to 400 kPa. In order to apply a load of 10 kPa the load multiplier must be set to 0.025. Therefore, update the "solid traction A multiplier initial" and "solid traction A multiplier final" to 0.025 in "Calculation data" as shown in Figure 3.99. Compute the new case by selecting <Anura3D → Calculate> from the menu. The calculation is performed in the new directory `oedometer_var1_0.1.A3D`. Finally, analyze the results.

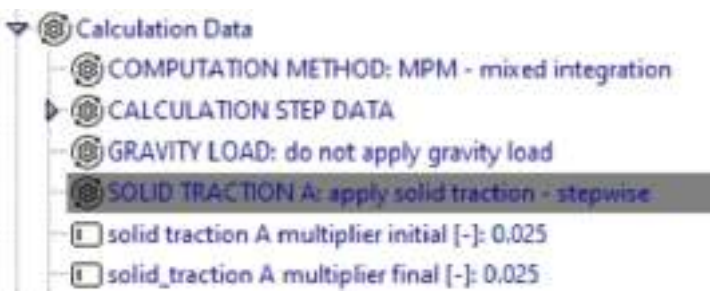


Figure 3.99: *Adjusting applied load in GiD.*

Alternatively:

(1) Create a new folder in your user directory with a new appropriate name, e.g. `oedometer_var2_10.A3D`.

```
D:\..\oedometer_var2_10.A3D>
```

(2) Copy the CPS_001 file, the GOM_stage0 file, the OPD_stage0 file and the Anura3D executable from the original project directory (`\oedometer.A3D`) to the new one.

(3) Open the CPS_001 file with any text editor.

(4) Decrease the applied load by manually editing the load multipliers (initial and final) in the CPS file:

```
$$SOLID_TRACTION
 step 0.025 0.025
```

(5) Save the CPS file.

(6) Repeat the steps as described previously in Sections 3.7.1 to launch the calculation from the "Command Line" window.

(7) Inspect the results as described in Section 3.4.

The evolution of liquid pressures considering an external load of 10 kPa is presented in Figure 3.100.

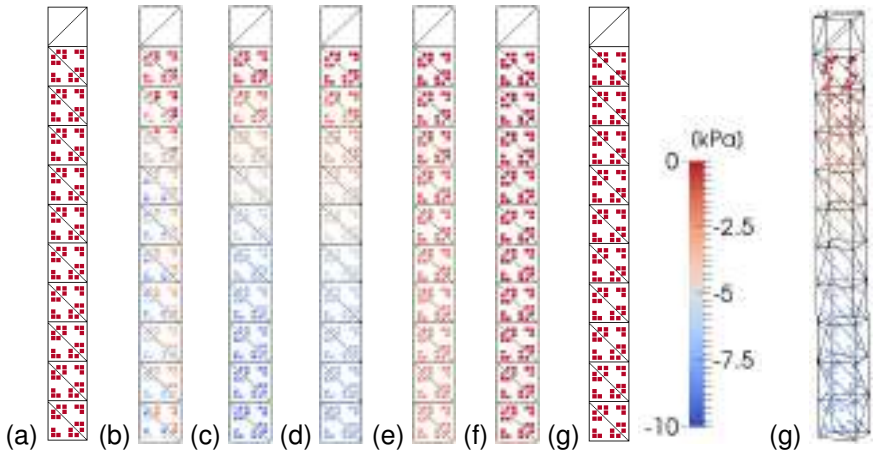


Figure 3.100: *Evolution of liquid pressure for an external load of* 10 *kPa applied on top of the soil column: (a)* t = 0 *s, (b)* 0.05 *s, (c)* 0.1 *s, (d)* 0.2 *s, (e)* 0.5 *s, (f)* 1 *s, (g)* 2.0 *s and (h)* 0.2 *s (3D).*

3.8 Discussion

Some general remarks:

- As in Anura3D a dynamic formulation is used that considers all inertial terms, dynamic effects have to be considered when interpreting the results.
- Including homogeneous local damping in the calculation reduces the energy of the system and the solution is damped. Only small local damping coefficients, up to 5%, are recommended for dynamic problems since higher values lead to energy loss of the system.
- Gauss point integration is recommended especially when large deformations occur. This integration method reduces internal instabilities that appear when material points cross element boundaries.
- In large deformation problems, external loads have to be assembled in the material points in order to move together with the deforming material.

4 Wave propagation

Authors: Alexander Chmelnizkij; Susmitha Ramachandran

This chapter will investigate the case of one-dimensional shock wave propagation through a dry and saturated column. Different damping and boundary conditions will be presented, resulting in different types of wave propagation solutions.

4.1 Problem definition

Consider a similar geometry as shown in Section 3.2.5 with reduced width of the column to a value of 0.001m.

Table 4.1: *Dimensions of the column*

dimension	[m]
height H	1.0
width/depth d	0.001

First, the case of dry material is studied. The properties for the material are given in Table 4.2.

Table 4.2: *Material parameters for the wave propagation problem.*

material	**column**
material type	dry material
initial porosity [–]	0.4
density solid [kg/m^3]	2650
K0-value [–]	0
material model solid	Linear Elasticity
Young modulus [kPa]	10000
Poisson ratio [–]	0.0

4.2 Creating input data

The geometry, boundary conditions, material properties and mesh will be defined using the GiD pre-processor. The calculation options will be defined in the CPS file. Both follows the same procedure as previously described in Section 3.1. Here only the differences with respect to the previous examples will be highlighted.

4.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described previously in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "wave").

4.2.2 Defining geometry

First, a 3D analysis and single-point formulation are chosen for this example. The height of the column is 1 m, and the width and depth are 0.001 m. In the same way, as described in Section 3.2.2, the column's lines, surfaces, and volumes are defined. In the presented case,

one material point per element is used. The analogous case of a two-dimensional geometry is left as an exercise.

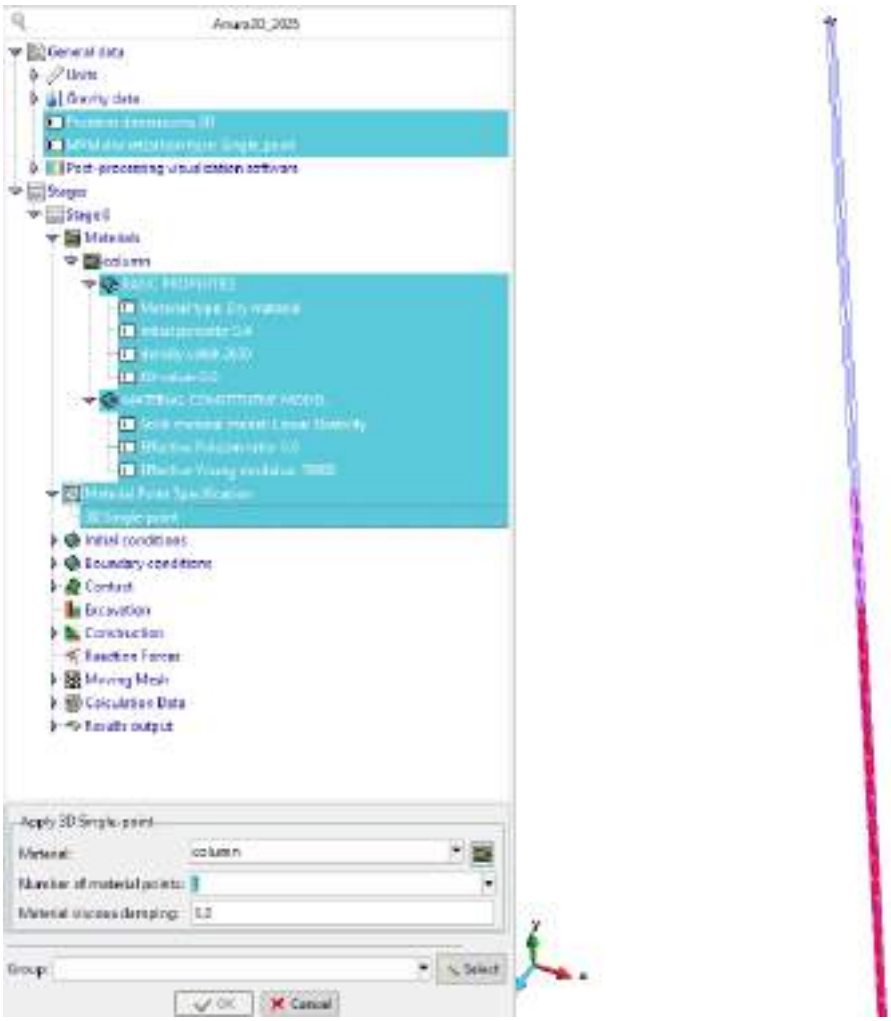


Figure 4.1: *Geometry and material properties/specifications of the column.*

4.2.3 Defining boundary conditions

Define the boundary conditions following the steps as described previously in Section 3.2.7.

Applying fixities

Fixity boundary conditions are applied for the column's sides and bottom. The sides located in the y-z-plane are fixed in the x-direction. The sides located in the x-y-plane are fixed in the z-direction. The bottom surface is fixed in the y-direction.

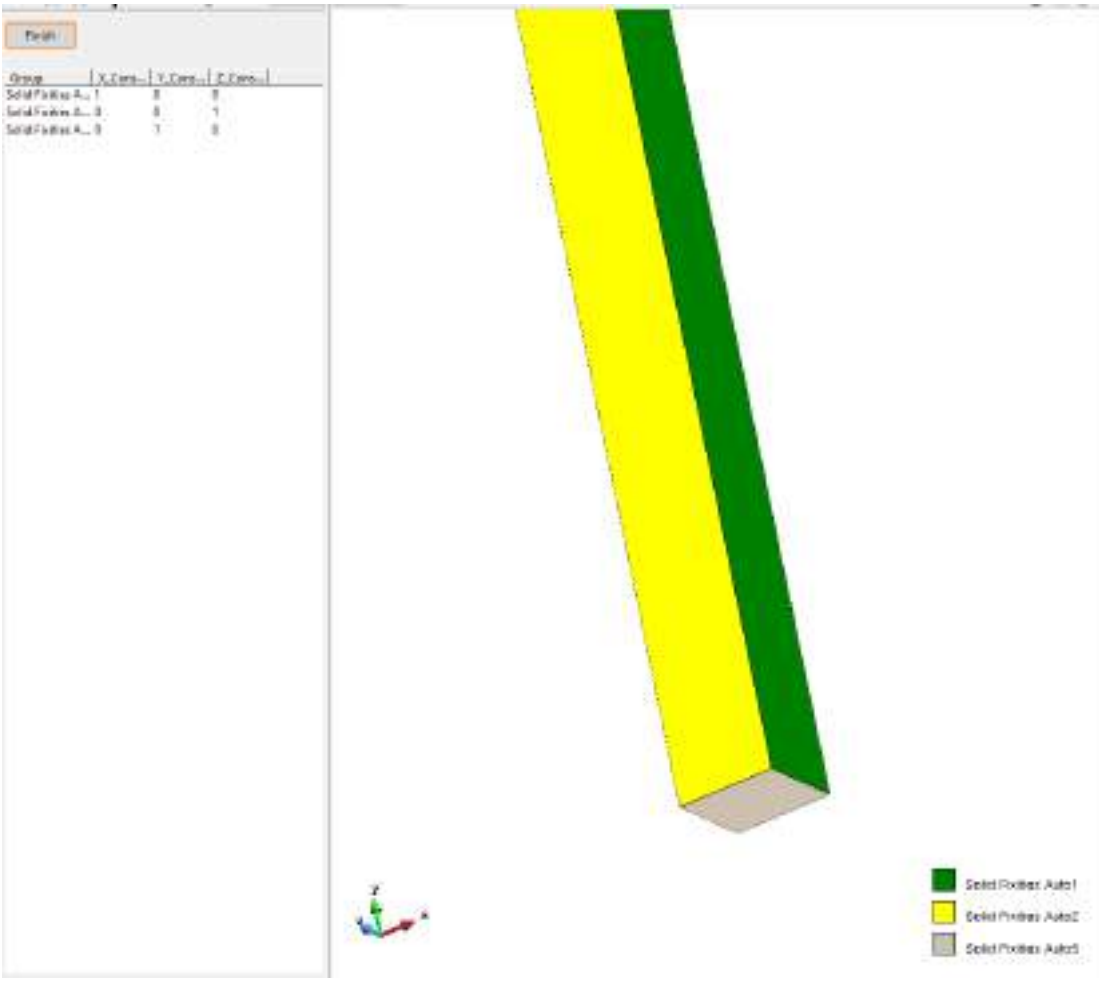


Figure 4.2: *Fixity boundary conditions for the sides and bottom of the column.*

4.2.4 Defining loading conditions

At the top of the column, a magnitude 1 [kPa] traction in the negative y-direction is applied to the surface. The traction is applied uniformly on material points.

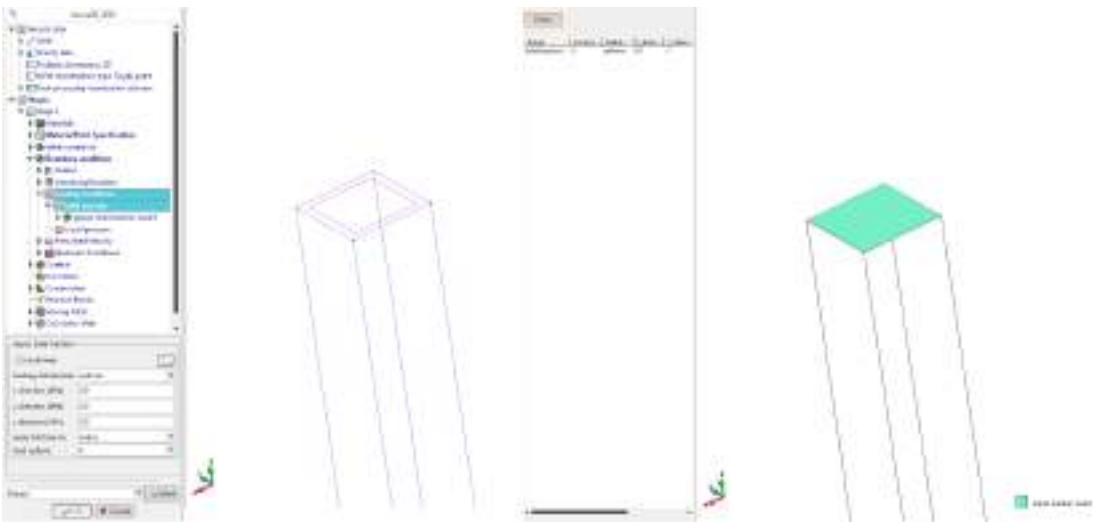
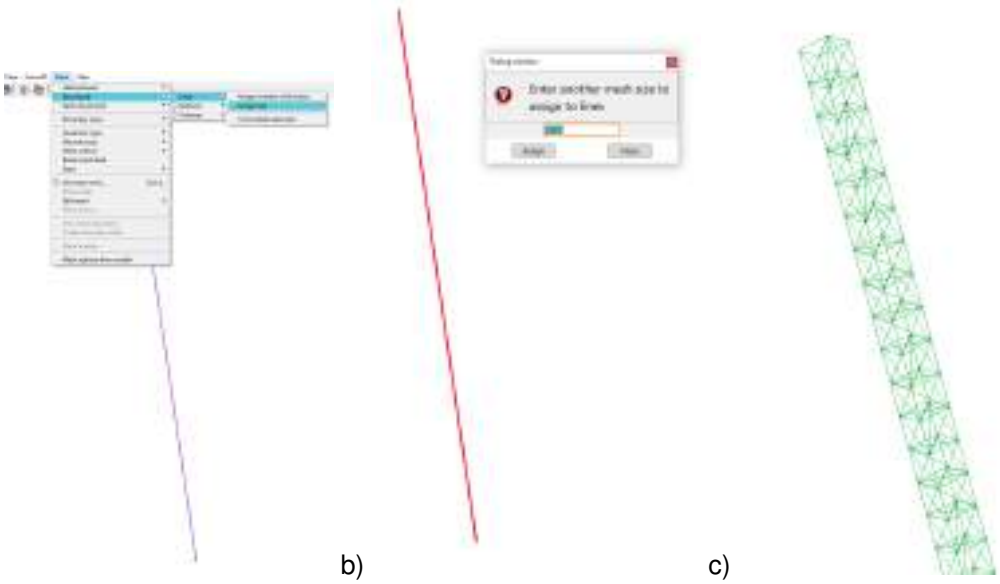


Figure 4.3: *Loading conditions for the top of the column.*

4.2.5 Creating mesh

A tetrahedral volume mesh is defined with an element size of 0.001m.



a) b) c)

Figure 4.4: *Defining mesh properties for the wave propagation problem in GiD. a) Choose strucutred mesh by size for lines b) Select all lines and set the element size to 0.001 m c) Check if the mesh was generated correctly*

4.3 Calculation data

In the calculation data, the number of calculation steps is set to 100 steps with a duration of 0.001 seconds. Material point integration is used as the computational method. The traction is applied stepwise with an initial and final magnitude of 1, corresponding to a suddenly applied traction, kept constant during the simulation. Bulk viscosity is also used with a linear coefficient of 0.06 and a quadratic coefficient of 1.2. The effect of these coefficients on the results will be investigated later.

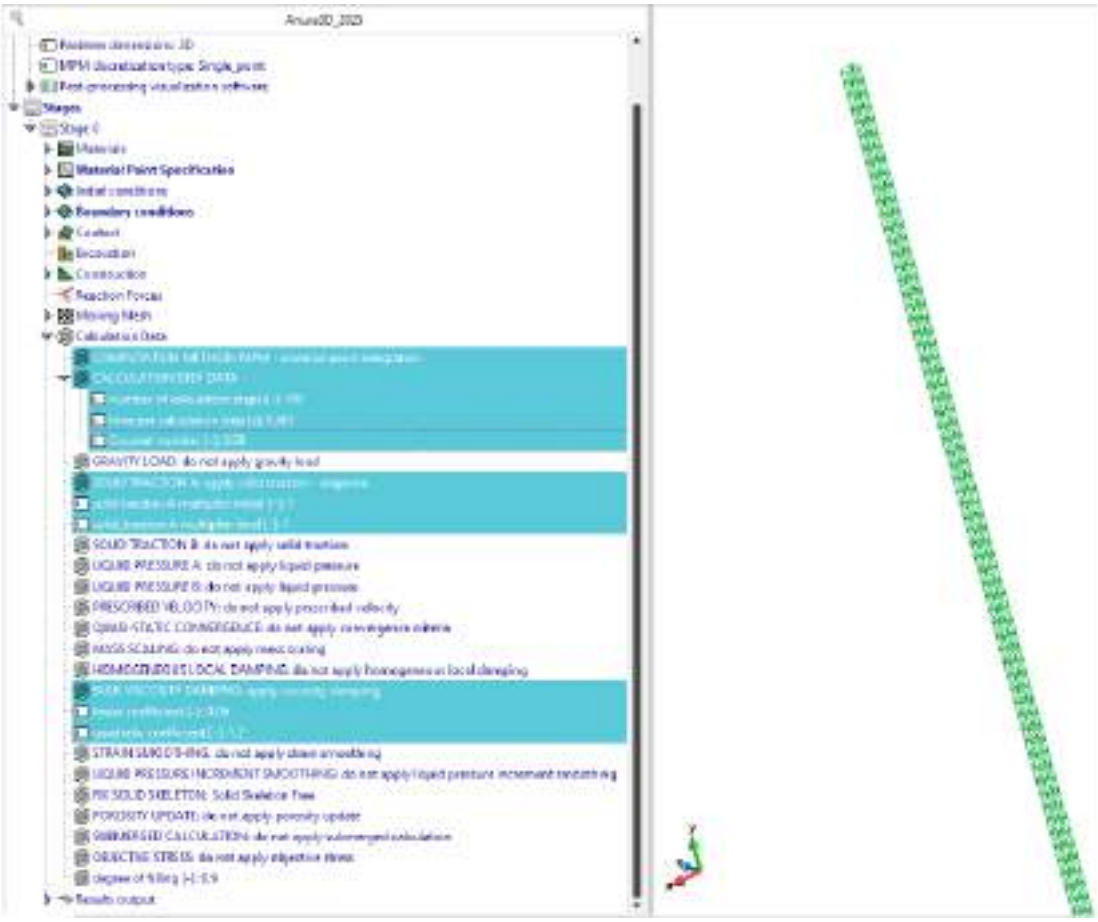


Figure 4.5: *Defining the calculation data.*

4.4 Generating input data and performing calculation

To generate the calculation data and perform the simulations follow the steps described in Section **??**.

4.5 Visualisation of results

The visualization of the results is done in ParaView. Therefore, a material point in the middle of the column is chosen, and its vertical effective stress is considered.

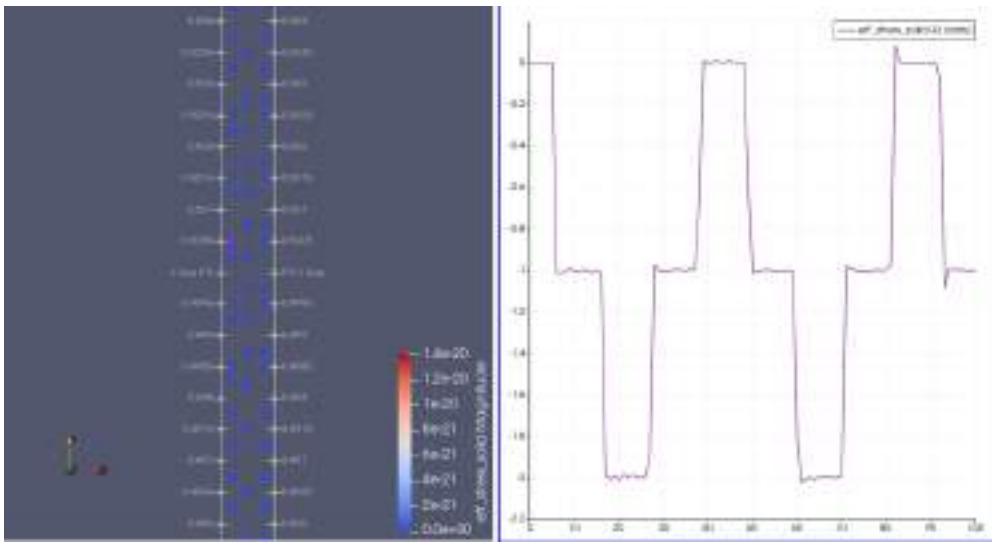


Figure 4.6: *Vertical efficite stresses of a material point in the middle of the column with bulk viscosity.*

As shown in Figure 4.6 the stress wave reaches the material point in the middle of the column after a while and suddenly increases the stress to -1 [kPa]. A bit later, the magnitude is increased to a value of -2 [kPa]. The second increase shows the arrival of the reflected wave from the fixed bottom of the column. As the top of the column is not subjected to fixities, the reflection turns a compressional wave into a tensile one, so the stress drops at the considered material point.

4.6 Variations: Bulk viscosity

The effect of bulk viscosity on the results can be studied by changing flags in the CPS file. To simulate the wave propagation without bulk viscosity, change the following entry of the CPS file,

```
$$BULK_VISCOSITY_DAMPING
1 0.06 1.2
```

to

```
$$BULK_VISCOSITY_DAMPING
0 0.06 1.2
```

The simulation results without bulk viscosity, as shown in Figure 4.7, oscillate stronger than the previous calculation. In particular, the oscillations appear stronger near the stress discontinuities.

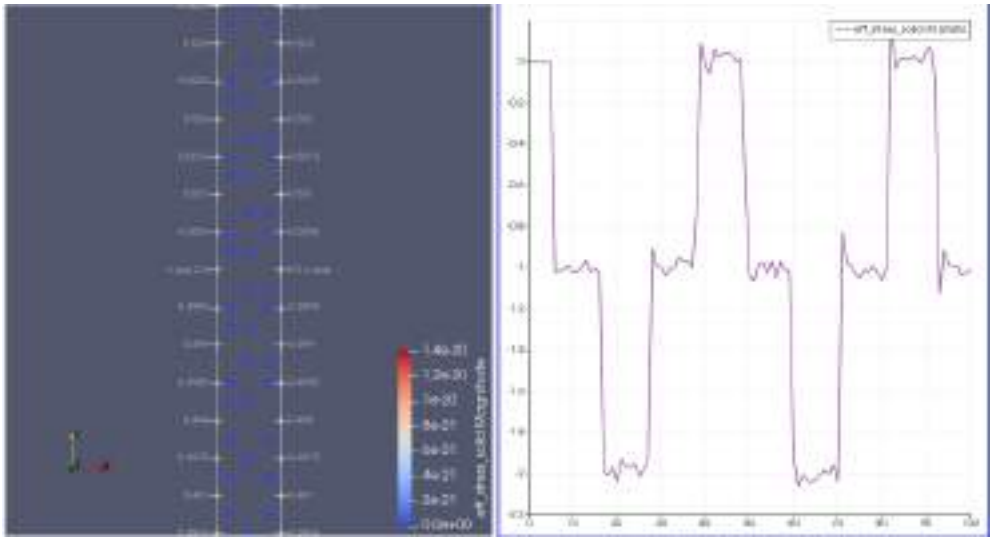


Figure 4.7: *Vertical efficite stresses of a material point in the middle of the column without bulk viscosity.*

A further increase of the bulk viscosity coefficients leads to over-smoothed results, as shown in Figure 4.8.

```
$$BULK_VISCOSITY_DAMPING
0 0.4 2
```

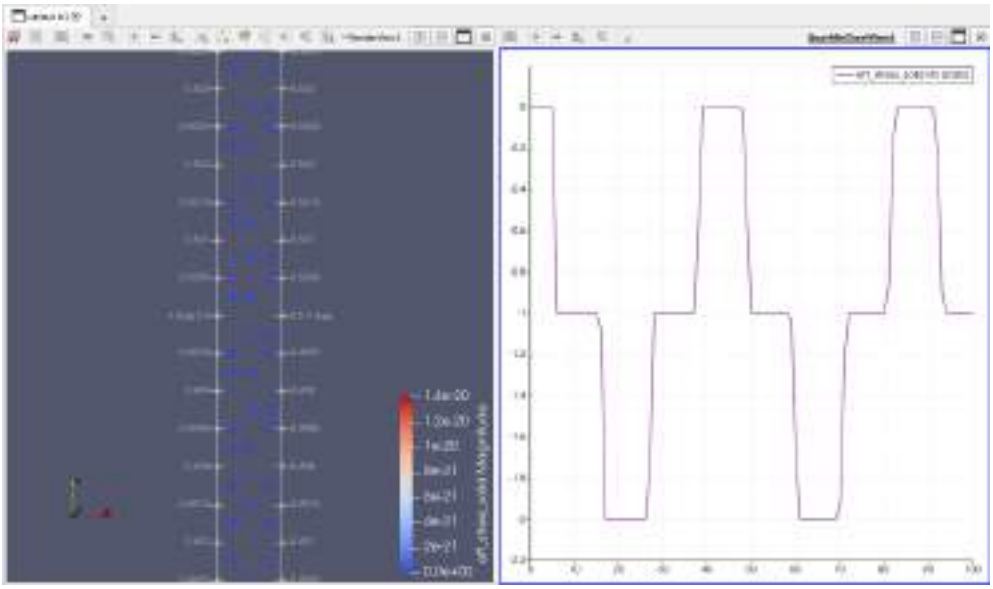


Figure 4.8: *Vertical efficite stresses of a material point in the middle of the column with higher bulk viscosity.*

4.7 Variations: Absorbing Bounadries

To avoid reflection at the bottom of the column instead of fixities, absorbing boundaries can be defined at the bottom. Therefore the assigned fixity needs to be unassigned, as shown in Figure 4.9.

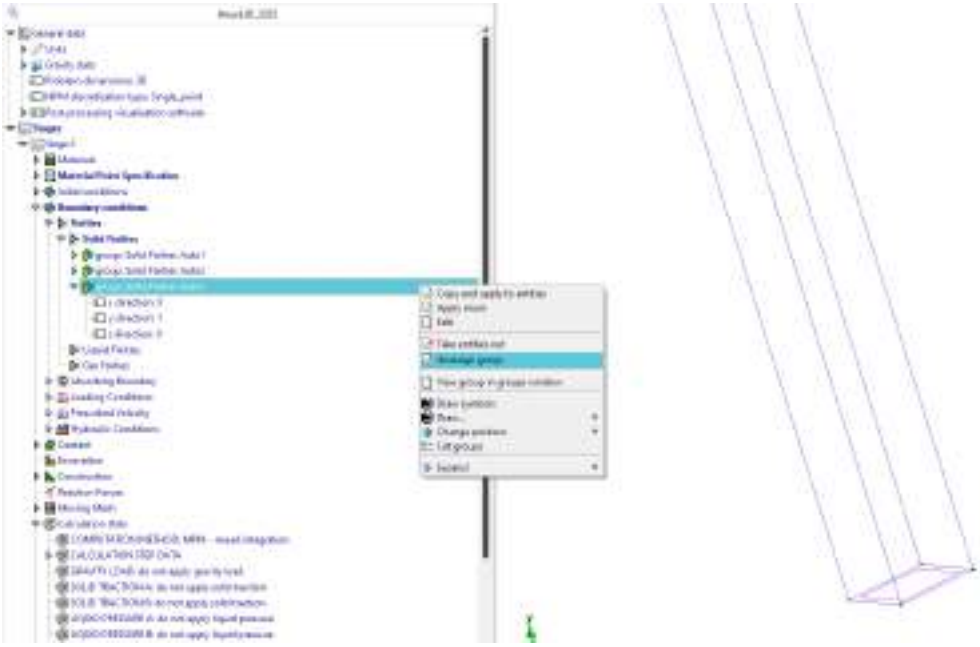


Figure 4.9: *Unassign fixity in y-direction at the bottom of the column.*

The absorbing properties of the boundary need to be defined in every direction. In Figure 4.10 the corresponding coefficients for the springs and dashpots are shown.

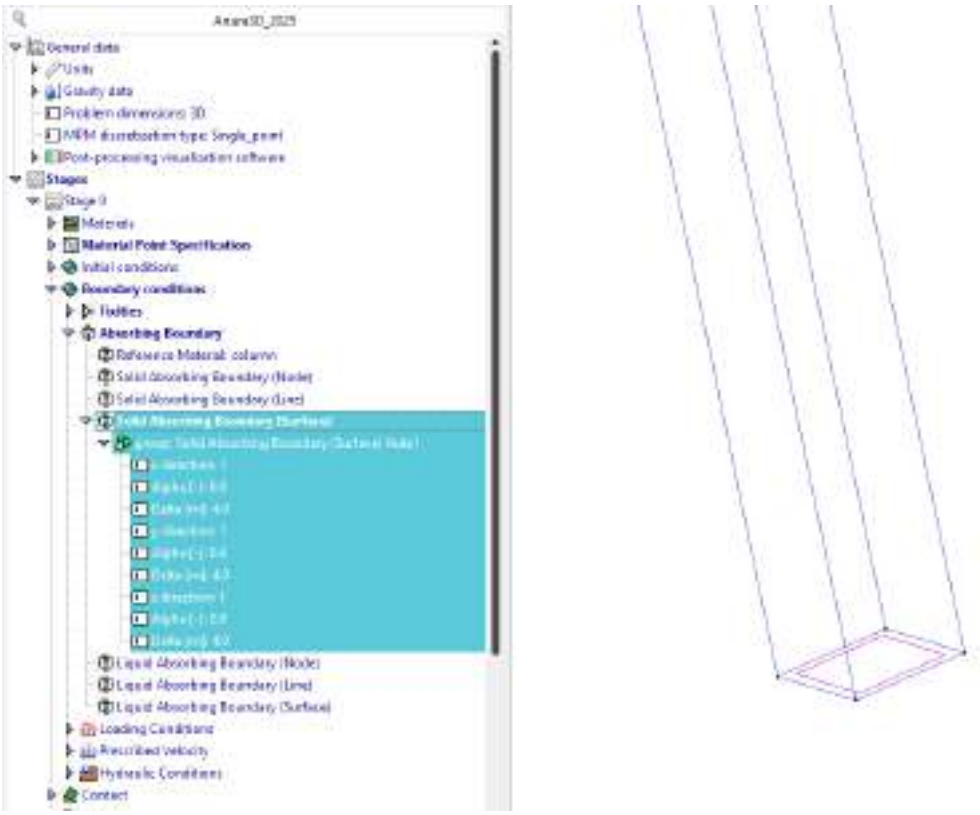


Figure 4.10: *Assign spring and dashpot properties to the bottom surface.*

Run the simualtion for 50 instead of 100 calculation steps. This can be either modified in

"Calculation Data" shown in Figure 4.5 or directly in the CPS file.

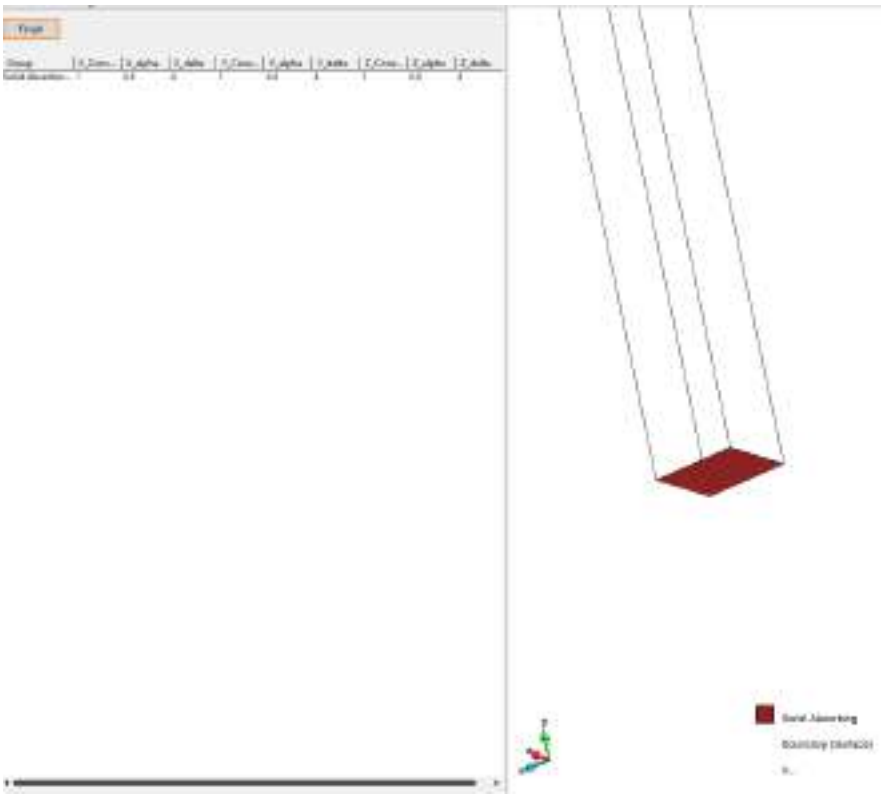


Figure 4.11: *Check if absorbing boundaries were correctly applied.*

The simulation results show that the reflected wave is almost entirely absorbed by the boundary and a constant stress level of approx. -1 [kPa] is reached quite fast.

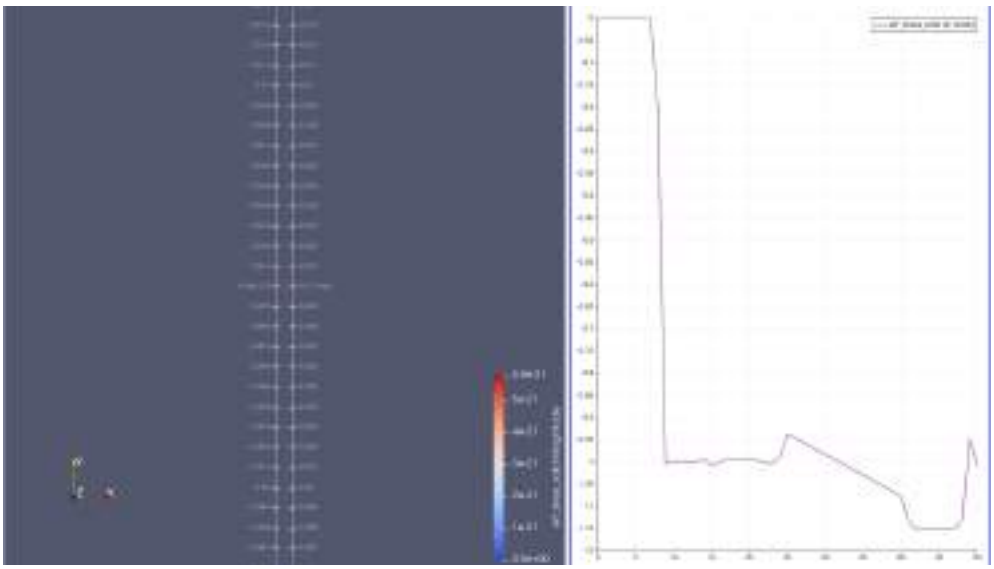


Figure 4.12: *Resulting vertical stresses in the middle of the column using absorbing boundaries.*

4.8 Variations: Coupled wave propagation

The wave propagation in saturated material is considered now. Two-dimensional elements are used now to discretize the column. The three-dimensional case presented in the previous section could be used here analogously. First, the material specifications and parameters are chosen, as shown in Figure 4.13.

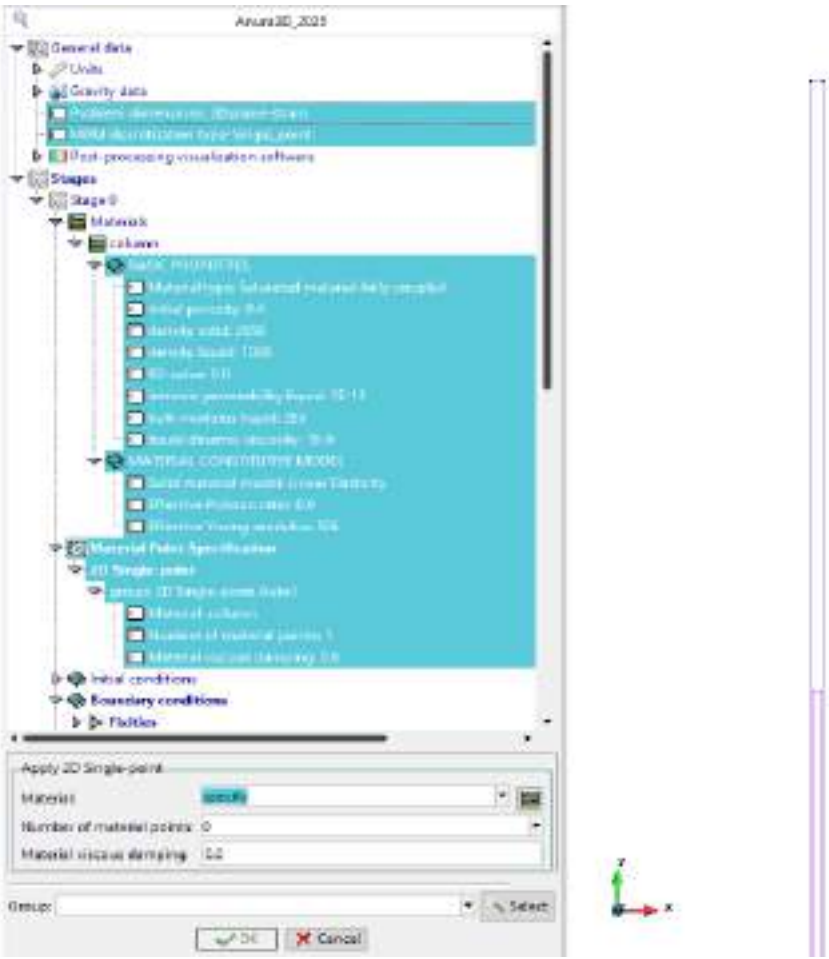


Figure 4.13: *Defining material specifications and properties for the coupled wave propagation.*

As a saturated material is used in this analysis, boundary conditions for the solid and liquid phases are needed. In contrast to the 3D case, the boundary conditions are now applied to lines instead of surfaces.

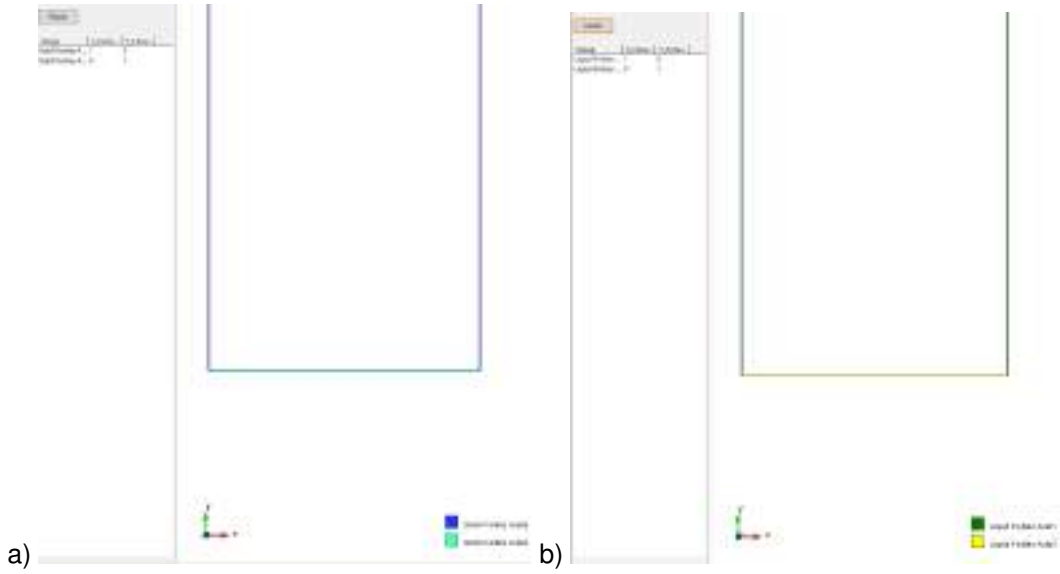


Figure 4.14: *Defining fixity boundary conditions for a) solid and b) liquid phases.*

Analogous to the previous example, a traction of magnitude 1 [kPa] for the solid phase is introduced at the top. The pressure of the liquid phase is zero at the top.

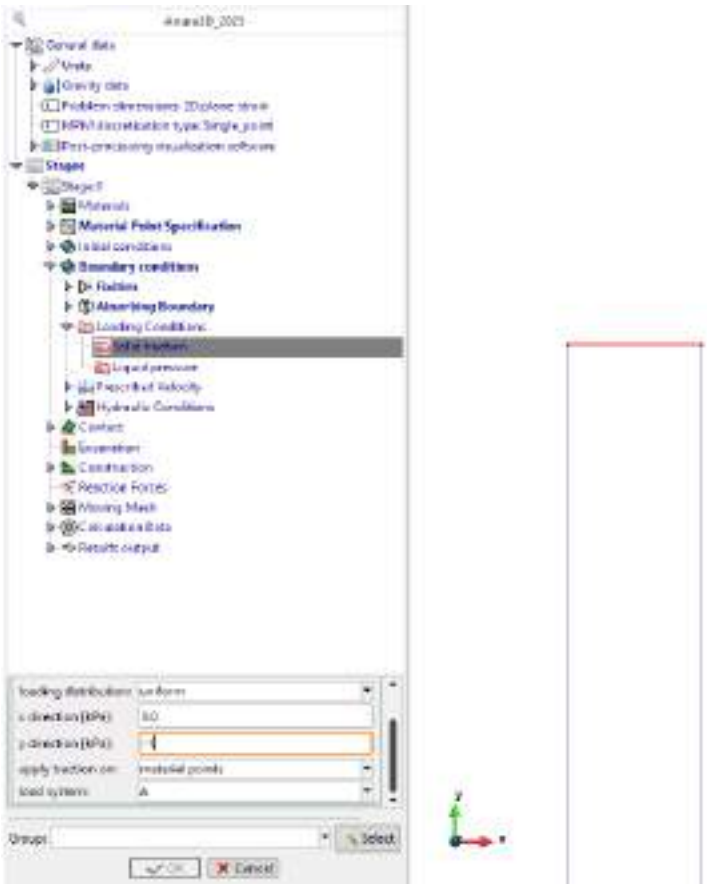


Figure 4.15: *Defining solid traction in negative y-direction.*

4.8.0.0.1 Creating mesh

Triangular elements with an element size of 0.0001m are used to discretize the geometry. The mesh is chosen to be structured along every line. The resulting mesh is shown in Figure 4.16.

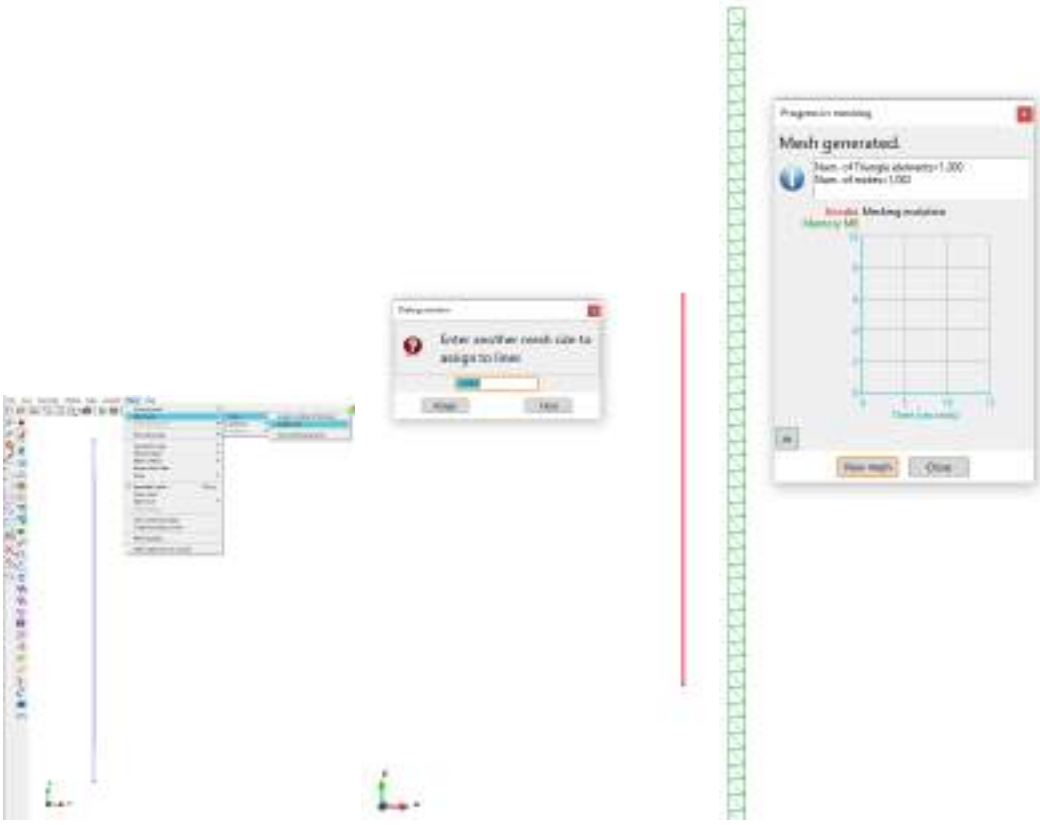


Figure 4.16: *Defining mesh properties for a structured trinagular mesh.*

4.8.0.0.2 Calculation data

The parameters for the calculation are chosen according to Section 4.3 with a slight deviation. The following flags are adjusted for the coupled analysis:

```
$$TIME_PER_LOADSTEP
0.0000002
```

```
$$BULK_VISCOSITY_DAMPING
1 0.42 1.2
```

```
$$OUTPUT_NUMBER_OF_MATERIAL_POINTS
1
$$OUTPUT_MATERIAL_POINTS
566
```

4.8.0.0.3 Visualisation of results

The output for the material point selected with the ID 566 can be found in the calculation folder with the file name ending PAR_566. Extracting the columns with the names SigmaYY and WPressure and plotting them over time gives the following figure.

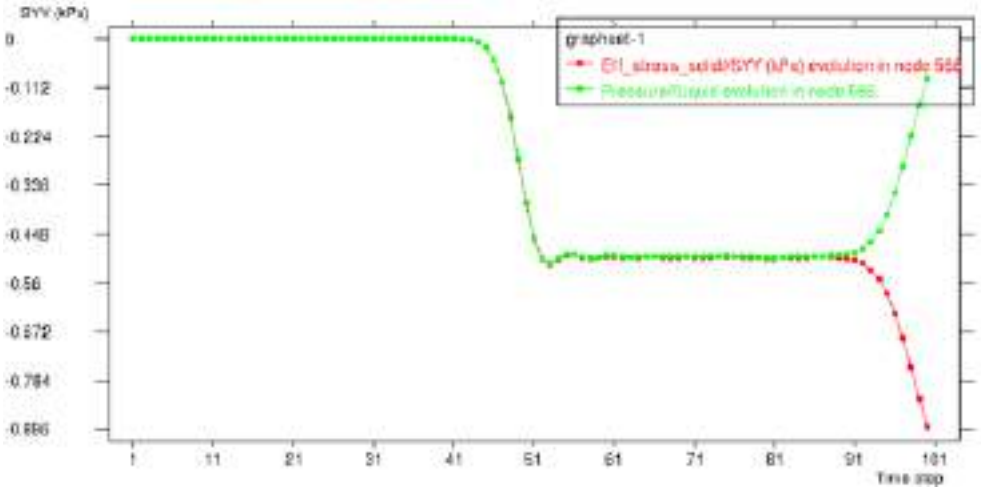


Figure 4.17: *Resulting vertical stresses and pore pressure for material point 566 plotted over time.*

5 Triaxial compression test

Authors: Pietro Marveggio; Alexander Rohe; James Fern; Alba Yerro

The aim of this tutorial is to simulate a triaxial compression test by using the finite element (FE) package of Anura3D . Besides the Material Point formulation, Anura3D includes two FE-based packages: the Standard FEM and the Updated Lagrangian FEM. In the first formulation all the integrations are carried out by considering the initial reference configuration at time t=0 (i.e. original volume and surface area), while in the latter the current configuration at time 't' is used (i.e. the volume and surface are updated accounting for body deformations). In this chapter, for the sake of simplicity, the Standard FEM approach will be used to simulate a simple boundary value problem considering a homogeneous cubic specimen. The following steps are required to complete the simulation.

- ◇ Creation of geometry
- ◇ Definition of material model
- ◇ Loading and boundary conditions definitions
- ◇ Generation of the mesh
- ◇ Execution of calculation
- ◇ Output of results

The simulation is performed considering one of the constitutive models included in Anura3D . The simulations are run in drained conditions using the one-phase single-point formulation (see Chapter 2, Fern et al. [1]). In the second part of the chapter, the procedure required to run a simulation by adopting any constitutive model implemented by the user is described (External soil model).

5.1 Problem definition

The triaxial compression test is carried out as strain-driven for which the top face of a soil cube is compressed with a constant velocity of 0.01 m/s. The gravity is neglected as its contribution towards the effective stress state is very small and, therefore, the dimensions of the cube do not influence the results; a 1-m wide cube is used. Figure 5.1 describes the geometry of the problem. The boundary conditions are such that the bottom surface is constrained in normal direction while the lateral surfaces have a compressive load of 50 kPa in the normal direction. The first example uses the Mohr-Coulomb model with the parameters listed in Table 5.1. The specimen is assumed to be isotropically consolidated and the initial stress can be initialised using the k_0-procedure (Chapter 6, Fern et al. [1]) with K_0 = 1.0.

5.2 Creating input data

The geometry, material properties, fixities, number of material points, contact conditions, initial calculation parameters, and mesh are defined with GiD as previously described in Section 3.2.

5.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as in Section 3.2.1. Save the project in the desired directory on the hard disk with a given name (e.g. "triaxial").

Table 5.1: *Material parameters of the triaxial problem.*

Parameter	Symbol	Unit	Value
Material type: dry material			
Initial porosity	n_0	[-]	0.3
Density solid	ρ_s	[kg/m^3]	2,650
Material model solid: Mohr-Coulomb			
Poisson ratio	ν	[-]	0.25
Young's modulus	E'	[kPa]	10,000
Cohesion	c'	[kPa]	3
Friction angle	φ'	[°]	35
Dilatancy angle	ψ	[°]	15

5.2.2 Defining geometry

Create the geometry as follows.

(1) Define points and lines of one the faces of the model (Figure 5.2) using the point coordinates indicated in Table 5.2.

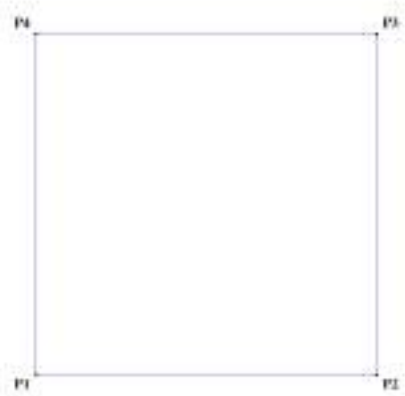


Figure 5.2: *Geometry of the front face of the model. Key points are indicated.*

(2) Generate the surfaces and create the volumes following the explanations in Chapter 3.2.2. Note that the thickness of this model must be z = 1 m. The final geometry should be as

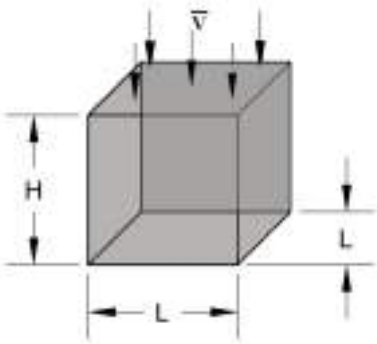


Figure 5.1: *Schematic description of the triaxial compression simulation.*

Table 5.2: *Key point coordinates of front faces of the sliding blocks problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	0	0
P2	1	0
P3	1	1
P4	0	1

shown in Figure 5.3.

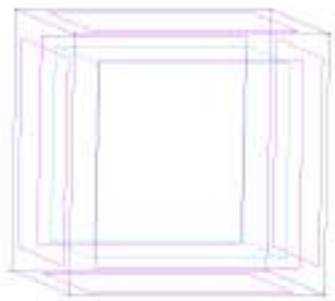


Figure 5.3: *Geometry of the triaxial test problem.*

5.2.3 Defining material properties

Before specifying material properties, ensure that the Anura3D problemtype has been set as described in Section 3.2.1 step (2).

The triaxial model consists of a single material. Follow the instructions detailed in Section 3.2.5 to define the material and assign it to the cubic volume. Material parameters are given in Table 5.1. Check if the assigned materials are correctly assigned as was shown in

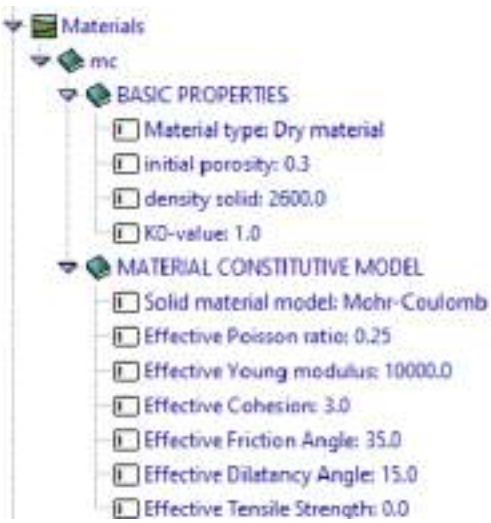


Figure 5.4: *Material dialogue box in GiD.*

Chapter 3.

5.2.4 Specifying material points

Specify the material point distribution following instructions from Section 3.2.6. For this case, choose 1 material points per element for box. As a standard FE approach will be considered, this value simply represents the number of integration points employed in the simulation. Check if the material points are assigned correctly, as was shown in Chapter 3.

5.2.5 Defining initial conditions

As mentioned previously, the stresses are initialized by means of the K_0-procedure. To do this, the "Apply K_0-procedure" option should be set on "Yes" and the initial vertical load has to be set to "-50.0 kPa". This option is found in the Anura3D tree menu under the section "Initial conditions" → "Stress initialization" → "Horizontal layered soil" (Figure 5.5).

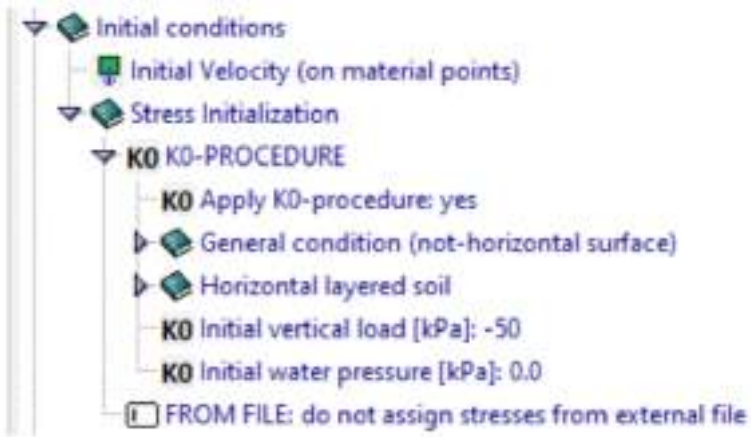


Figure 5.5: *Initial conditions to apply K0-procedure*

5.2.6 Defining boundary conditions

Define the boundary conditions following the steps as described previously in Section 3.2.7.

Applying fixities
As the material is dry, only the solid phase needs to be considered. The solid displacement at the bottom have to be fixed in direction normal to the surface. After the fixity definition, check if the condition is assigned correctly.

Applying loading conditions
On the lateral surface of the box, a uniform distributed load of 50 kPa is applied inwards in normal direction with respect to the surface. As the simulation will run according to the standard FE approach, the load are applied on the nodes.
Before going on with the pre-processing phase, check if the applied load is assigned correctly.

Applying prescribed velocity
The triaxial simulation is strain driven. To do this, a vertical velocity of 0.001 m/s is applied downwards at the top surface of the soil column. The following steps need to be done.

(1) Choose <Prescribed Velocities> from the data-tree.
(2) The "Prescribed velocities" tree appears. The prescribed velocity has to be applied on a surface. As the velocity is applied on nodes, double click on the "Nodal Velocity" icon, as

shown in Figure 5.6.

Figure 5.6: *Prescribed velocity conditions dialogue box in GiD. Assign the velocity to the surface nodes*

(3) Enter the value of the downward vertical velocity of 0.001 m/s will be applied. Therefore, enter "-0.001" in the field "y-direction [m/s]" as shown in Figure 5.6.
(4) Click on the <Select> button in order to select the surface where the load should be assigned to as shown in Figure 5.6.
(5) Select the upper surface of the cube, which should be highlighted in red.
(6) Press <ESC> OR click on <End> to leave the selection of surfaces.
(7) Check if the prescribed velocity is assigned correctly. The prescribed velocity condition applied should be as shown in Figure 5.7.

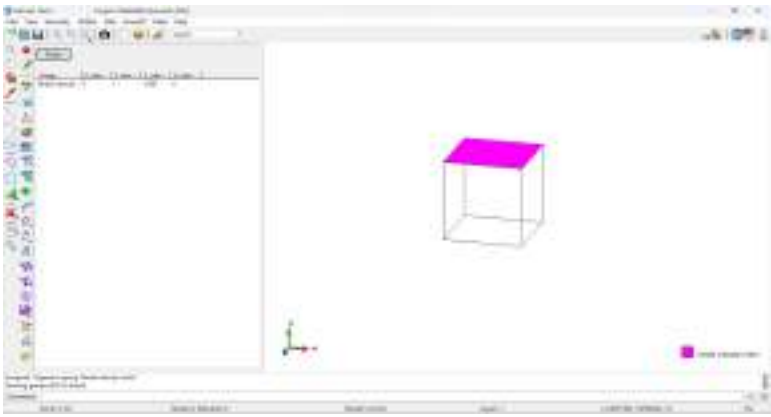


Figure 5.7: *Checking prescribed velocity for the triaxial problem in GiD.*

(8) Click on <Finish> and <Close> to quit the assigning fixities process.

5.2.7 Creating mesh

A structured mesh is used for this example. As the problem considered is sufficiently simple, only a few elements are used to run the simulation. To generate the mesh follow the steps detailed below. (Note that 3D tetrahedral elements have to be defined as quadratic type elements in GiD).

(1) Choose <Mesh> <Generate Mesh> from the menu OR press <CTRL>+<G>.

(2) The "Mesh generation" dialogue box appears. For this example, set the mean element size to "1" as shown in Figure 5.8. This corresponds to the depth z of model.

Figure 5.8: *Enter the size of elements equal to 1 m.*

(3) Click the <OK> button. The mesh will be generated and the progress is shown in the "Progress in meshing" dialogue box as shown in Figure 5.9. The number of generated tetrahedral elements (12) can be seen as well as the number of generated nodes (35).

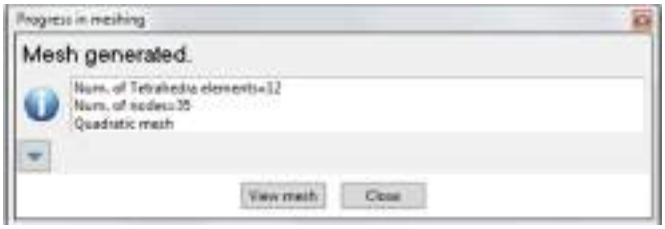


Figure 5.9: *Mesh generated dialogue box.*

(4) Click on the <View mesh> button to quit the meshing process and see the generated mesh. It should look like Figure 5.10

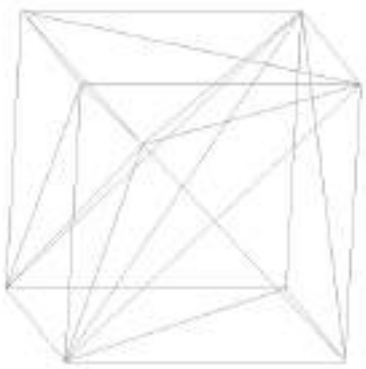


Figure 5.10: *Computational mesh of the triaxial test problem.*

5.2.8 Initialization of calculation data

The calculation of the triaxial problem will be carried out in one stage.

Computation method

The computation method used is the standard finite element one. Therefore the option "Standard FEM" should be selected.

Calculation step data
The compression of the soil column will be analysed for 5 seconds. To create intermediate results, the total time is divided into 50 load steps of 0.1 s each. A Courant number of 0.5 is suggested to ensure the stability of the solution.

Gravity Load
In this example, the gravity is neglected. Therefore, the application of gravity option should be set on "do not apply gravity load" (default value value).

External Load
The external load is applied suddenly on the lateral surface of the sample and is maintained constant throughout the calculation. Therefore, set the solid traction box either to "apply solid traction - linear" or "apply solid traction - stepwise", as both works in this case, since the multipliers for solid traction has to be set to 1.0 (both initial and final value).

Prescribed velocity
As for the external load, also the prescribed velocity is applied suddenly on the top surface of the sample and is maintained constant throughout the calculation. Again the solid traction box can be set either to "apply prescribed velocity - linear" or "apply prescribed velocity - stepwise" and the multipliers to 1.0 (both initial and final value).

Default values
All other calculation options are not used in this example and the provided default values can be kept, i.e. regarding liquid pressure, contact algorithm, smoothing algorithms, quasi-static convergence, etc.

To define the initial calculation parameters, follow the instructions detailed in Section 3.2.9. For this example, enter the values given below;

- Select "COMPUTATION METHOD" as "Standard FEM" from the drop down menu
- Set "number of calculation steps [-]" to "50"
- Set "time per calculation step [s]" as "0.1".
- Set "Courant number" to "0.5"
- Set select "apply solid traction - stepwise" in the solid traction box and set the corresponding multipliers to "1.0 1.0"
- Select "apply prescribed velocity - stepwise" in the prescribed velocity box and set the corresponding multipliers to "1.0 1.0"

All other calculation parameters can be kept at their default values.

5.2.9 Generating input files

After the geometry, material properties, boundary conditions, loading conditions and calculation parameters are defined and the mesh is generated, the input files for the Anura3D analysis (i.e. *GOM* and *CPS* file) have to be created. To generate the *GOM* and *CPS* files, follow the steps as described in Section **??**.

The generated input files `triaxial.GOM` and `triaxial.CPS_001` can be found in the new folder `triaxial.A3D` in the directory, which was specified earlier by the user:

D:\..\triaxial.A3D\triaxial.GOM

D:\..\triaxial.A3D\triaxial.CPS_001

5.3 Performing calculation in Anura3D

The Anura3D calculation can be started following instructions described in Section 3.3.

The calculation starts and output is generated in the "Command Line" window indicating the progress of the calculation. Any possible input and calculation errors are displayed in this window as well. The output files are generated and stored in the project directory `D:\..\tri.A3D`.

5.4 Visualisation of text output files

The VTK files can be opened with ParaView and the evolution of volumetric and deviatoric strains and stresses in one point of the sample can be extracted and analysed. As the behaviour of the material inside the box is intended to be homogeneous, any of the integration points can be used to extract information required for the analysis. In this example, the average values over the sample is used. Following the steps described in Section 3.4.8 without specifying any material point,the evolution of variables over time can be obtained. These are included in the MPMScalar files. These data can then be processed in a spreadsheet, e.g. MS Excel.

The result is a vertical compression stress that is linearly increasing until the yield surface is reached. From that moment on, the vertical stress remains constant and only plastic deformation occurs (see Figure 5.11). The evolution of the volumetric strain is determined by the linear elastic parameters until when the yield surface is reached. From that point on, the volumetric strain starts increasing due to the definition of a non-zero dilatancy angle (see Figure 5.12). Figure 5.13 shows that the stress state evolves until the yield function is reached, from that moment on the deviatoric stress remains constant with deviatoric strain, while the volumetric strain rate increment changes sign according to the dilatancy of the material.

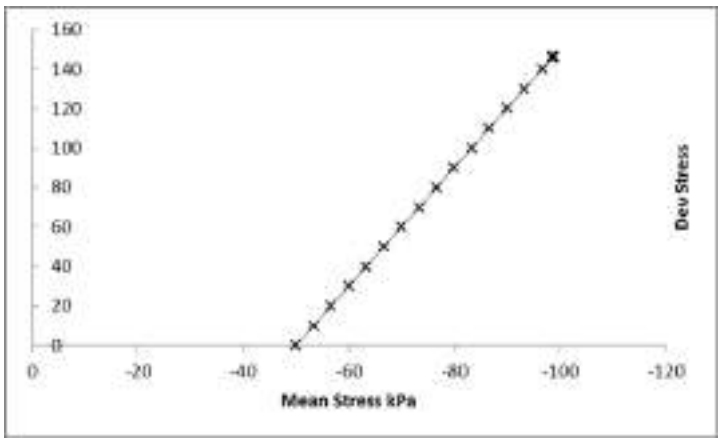


Figure 5.11: *Triaxial test using Mohr-Coulomb: deviatoric stress with isotropic pressure*

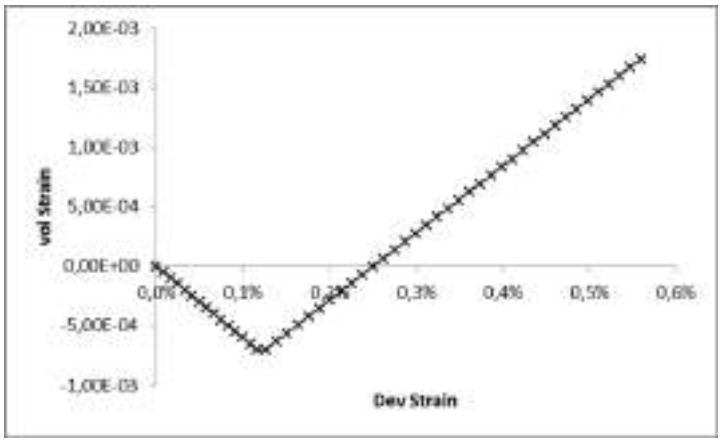


Figure 5.12: *Triaxial test using Mohr-Coulomb: volumetric strain with deviatoric strain*

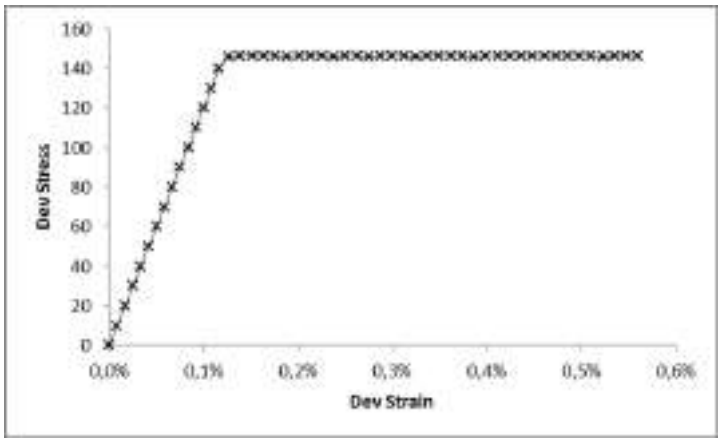


Figure 5.13: *Triaxial test with Mohr-Coulomb: deviatoric stress vs deviatoric strain*

5.5 External Soil Model

Anura3D has a feature to deal with external soil models (ESM), that are provided by the user. This facility allows any user to make use of a wider range of constitutive soil models in Anura3D . The ESM is based on the ABAQUS standard (implicit) for which the stiffness matrix

DDSDDE is returned to Anura3D . The ESM must be included in Anura3D executable. Although the user is free to modify the code, the procedure here suggested consists in creating a specific module, to be included in specific files of the solution. Such module should be programmed in FORTRAN (or another programming language), then compiled together with the main executable. The user supplied material subroutine is required to update the integration point effective stress and state variables, providing as well an estimation of the constrained modulus, that is required to evaluate the maximum time step size to ensure stability of the analysis.
To include an external material, essentially three steps have to be followed:

- Prepare the ESM module
- Include the module into "ExternalSoilModel.for" and "ReadMaterialData.for"
- Define the pointer to the ESM in "ReadMaterialData.for"
- Compile the executable
- Define user supplied material in GiD.

5.5.1 Preparation of the FORTRAN File

The A3D interface for external soil model is intended to work according to the ABAQUS User material (UMAT) file standard. In fact the FORTRAN file will be composed by two main subroutines. One is the actual ESM Anura3D interface, while the other is the UMAT.
External soil model functions are here written using FORTRAN, to be consistent with the ABAQUS standard. In order for the Anura3D solver to recognise the user subroutine, the name of the subroutine as well as the data types of the arguments must exactly match those given below.

A general formulation of the ESM subroutine ("ESM_NONLINEAR" included in the module "ModNoNLinearElasticity") is here shown. Note that the name of the module you choose should be the one included later in specific modules, while the name of the subroutine should correspond to the pointer that will be later defined.

```
module ModNoNLinearElasticity
contains

Subroutine ESM_NONLINEAR(NPT,NOEL,IDSET,STRESS,EUNLOADING,PLASTICMULTIPLIER, &
DSTRAN,NSTATEV,STATEV,NADDVAR,ADDITIONALVAR,CMNAME,NPROPS,PROPS,NUMBEROFPHASES,NTENS)


implicit double precision (a–h, o–z)
integer :: NTENS, NSTATEV, NADDVAR, NPROPS, NPT, NOEL, IDSET, NUMBEROFPHASES
double precision :: EUNLOADING, PLASTICMULTIPLIER
CHARACTER*80 CMNAME
DIMENSION STRESS(NTENS), DSTRAN(NTENS),STATEV(NSTATEV),ADDITIONALVAR(NADDVAR),PROPS(NPROPS)


!–––Local variables required in standard UMAT
integer :: IStep, TimeStep
double precision, dimension(:), allocatable :: ddsddt ! only for fully coupled thermal analysis: variation of stress increment c
double precision, dimension(:), allocatable :: drplde ! only for fully coupled thermal analysis: variation of volumetric heat ge
double precision, dimension(:), allocatable :: stran
double precision, dimension(:), allocatable :: time
double precision, dimension(:), allocatable :: predef
double precision, dimension(:), allocatable :: dpred
double precision, dimension(:), allocatable :: coords
```

```
double precision, dimension(:,:), allocatable :: ddsdde ! Jacobian matrix of the constitutive model (tangent stiffness mat
double precision, dimension(:,:), allocatable :: drot
double precision, dimension(:,:), allocatable :: dfgrd0
double precision, dimension(:,:), allocatable :: dfgrd1
double precision :: sse, spd, scd ! specific elastic strain energy, plastic dissipation, creep dissipation
double precision :: rpl ! only for fully coupled thermal analysis: volumetric heat generation
double precision :: drpldt ! only for fully coupled thermal analysis: variation of volumetric heat generation due to temper
double precision :: pnewdt, dtime, temp, dtemp, celent
double precision :: Value ! auxiliary variable holding any real valued number
double precision :: Porosity, WaterPressure, WaterPressure0, GasPressure, GasPressure0, DegreeSaturation


integer :: ndi, nshr, layer, kspt, kstep, kinc

!---Local variables defned by the user
! e.g. integer :: var_local
!---User can define here additional variables

allocate( ddsddt(ntens), drplde(ntens), stran(ntens), time(2), predef(1), dpred(1), &
coords(3), ddsdde(ntens,ntens), drot(3,3), dfgrd0(3,3), dfgrd1(3,3) )

!Initialization
Eunloading = 0.0
PlasticMultiplier = 0.0

!Rename additional variables
Porosity = AdditionalVar(1)
WaterPressure = AdditionalVar(2)
WaterPressure0 = AdditionalVar(3)
GasPressure = AdditionalVar(4)
GasPressure0 = AdditionalVar(5)
DegreeSaturation = AdditionalVar(6)
time(1) = AdditionalVar(7) !TotalRealTime
time(2) = AdditionalVar(8) !OverallTotalTime
dtime = AdditionalVar(9) !TimeIncrement
IStep = AdditionalVar(10)
TimeStep = AdditionalVar(11) !Note: Very first time and load step: Istep=1 and TimeStep=1
!Call the UMAT
call umat(stress, statev, ddsdde, sse, spd, scd, rpl, ddsddt, drplde, drpldt, stran, dstran, time, dtime, temp, &
dtemp, predef, dpred, cmname, ndi, nshr, ntens, nstatev, props, nprops, coords, drot, pnewdt, celent, dfgrd0, &
dfgrd1, noel, npt, layer, kspt, kstep, kinc)


!---Definition of Eunloading -> required to define the max time step
Eunloading = max(ddsdde(1,1),ddsdde(2,2),ddsdde(3,3))
!---Always define this value to run the simulation

! PlasticMultiplier can be given as an output because plastic points can be plotted as a result




return

end subroutine ESM_NONLINEAR


```

```
SUBROUTINE UMAT(STRESS,STATEV,DDSDDE,SSE,SPD,SCD, &
RPL,DDSDDT,DRPLDE,DRPLDT, &
STRAN,DSTRAN,TIME,DTIME,TEMP,DTEMP,PREDEF,DPRED,CMNAME, &
NDI,NSHR,NTENS,NSTATEV,PROPS,NPROPS,COORDS,DROT,PNEWDT, &
CELENT,DFGRD0,DFGRD1,NOEL,NPT,LAYER,KSPT,KSTEP,KINC)

implicit double precision (a-h, o-z)
! !DEC$ ATTRIBUTES DLLEXPORT, ALIAS:"UMAT" :: UMAT
! INCLUDE 'ABA_PARAM.INC'
integer :: NTENS, NSTATEV, NPROPS
CHARACTER*80 CMNAME
DIMENSION STRESS(NTENS),STATEV(NSTATEV), &
DDSDDE(NTENS,NTENS),DDSDDT(NTENS),DRPLDE(NTENS), &
STRAN(NTENS),DSTRAN(NTENS),TIME(2),PREDEF(1),DPRED(1), &
PROPS(NPROPS),COORDS(3),DROT(3,3),DFGRD0(3,3),DFGRD1(3,3)


! Arguments:
! I/O Type
! PROPS I R() : List with model parameters
! DSTRAN I R() : Strain increment
! DDSDDE O R(,) : Material stiffness matrix
! STRESS I/O R() : stresses
! STATEV I/O R() : state variables

!--- Local variables
dimension dSig(NTENS), Sig(NTENS)

! Contents of PROPS(3)
! 1 : E Young Modulus
! 2 : ENU Poisson's ratio
 ! 3 : PLIM

 E = PROPS(1)
 ENU = PROPS(2)
 PLIM = PROPS(3)
 one = 1.0d0
 two = 2.0d0
 PRESS = (STRESS(1) + STRESS(2) + STRESS(3))/3
 EDEG = E/(1+PRESS/PLIM)
 G = EDEG/two/(one+ENU)
 ! calculate elastic stress increment (DSigE = elastic stiffness D * strain increment DEps)
 FAC = two * G / ( one - two * ENU )
 D1 = FAC * ( one - ENU )
 D2 = FAC * ENU
 DSTRANVOL = DSTRAN(1) + DSTRAN(2) + DSTRAN(3)
 dSig(1) = (D1 - D2) * DSTRAN(1) + D2 * DSTRANVOL
 dSig(2) = (D1 - D2) * DSTRAN(2) + D2 * DSTRANVOL
 dSig(3) = (D1 - D2) * DSTRAN(3) + D2 * DSTRANVOL
 dSig(4) = G * DSTRAN(4)
 if (NTENS == 6) then
 dSig(5) = G * DSTRAN(5)
 dSig(6) = G * DSTRAN(6)
 end if
 ! elastic stress
 Sig = STRESS + dSig

```

```
␣!␣stress␣state␣parameters␣update
␣do␣i␣=␣1,␣NTENS
␣STRESS(i)␣=␣Sig(i)
␣end␣do

␣DDSDDE␣=␣0.0
␣DDSDDE(1:3,1:3)␣=␣D2
␣DDSDDE(1,1)␣=␣D1
␣DDSDDE(2,2)␣=␣D1
␣DDSDDE(3,3)␣=␣D1
␣DDSDDE(4,4)␣=␣G
␣if␣(NTENS␣==␣6)␣then
␣DDSDDE(5,5)␣=␣G
␣DDSDDE(6,6)␣=␣G
␣end␣if

␣return
␣end␣subroutine␣umat

␣end␣module␣ModNoNLinearElasticity
␣␣␣␣␣␣␣
```

The descriptions of the arguments that are used to communicate between the Anura3D solver and the External Soil Model subroutine are listed below:

- *NPT*: global integration/material point number
- *NOEL*: global element number
- *IDSET*: ID of material parameter set
- *STRESS*: Effective stress at previous step (Input). It must be updated in the subroutine (output)
- *EUNLOADING*: Unloading constrained modulus, that is required to evaluate the maximum time step size to ensure stability of the analysis.
- *PLASTICMULTIPLIER*: Value that is stored and can be plotted as output to detect plastic region
- *DSTRAN*: Strain increment at the point/integration point
- *NSTASEV*: Number of state variables
- *STASEV*: State variables
- *NADDVAR*: Number of additional variables
- *ADDITIONALVAR*: additional variables, detailed below
- *CMNAME*: Name of the material constitutive model
- *NPRPOPS*: Number of properties of the material
- *PRPOPS*: Model parameters
- *NUMBEROFPHASES*: Number of phases of the material point
- *NTENS*: Dimension of the stress and strain array

This is the list of additional variables that the code provides to the user:

- *ADDITIONALVAR(1)*: Porosity
- *ADDITIONALVAR(2)*: Water pressure at the current step
- *ADDITIONALVAR(3)*: Water pressure at the previous step
- *ADDITIONALVAR(4)*: Gas pressure at the current step (not available in v2019.1)
- *ADDITIONALVAR(5)*: Gas pressure at the previous step (not available in v2019.1)
- *ADDITIONALVAR(6)*: Degree of saturation (not available in v2019.1)

- *ADDITIONALVAR(7)*: Total time of the current step
- *ADDITIONALVAR(8)*: Total time
- *ADDITIONALVAR(9)*: Time increment
- *ADDITIONALVAR(10)*: Total time step
- *ADDITIONALVAR(11)*: Time step of the current calculation step

After the call of the UMAT and the stress update, it is always important to define a finite value of the unloading oedometric modulus *EUnloading*. For many constitutive models, its definition is not trivial, so its value can be in general set equal to the maximum value among ddsdde(1,1), ddsdde(2,2) and ddsdde(3,3), as it is suggested by the template. If this simplification becomes too inaccurate, in order to ensure stability, the user should set a sufficiently low value of Courant number, that slows the analysis.

Concerning the UMAT structure, as this follows the ABAQUS standard, it will not be described here. An example of non-linear elastic material is in any case included for the sake of completeness. In this example the Young modulus increases with the increasing of the isotropic pressure at the integration point.

The file can be generated by means of a standard text editor and saved as *namefile.f* in order the compiler to recognize the FORTRAN format. Add the file you created into the Visula Studio project

5.5.2 Include the Module into the code

To take advantage of an external material model, to run a simulation with Anura3D , the model previously defined should be included in the model ModExternalSoilModel included in "ExternalSoilModel.for" and in ModReadMaterialData included in "ReadMaterialData.for". For the sake of completeness, hereafter the example of ModExternalSoilModel is provided:

```
module ModExternalSoilModel
!*********************************************************************
!
! Function: Contains the routines related to calling user-defined soil models.
! $Revision: 9064 $
! $Date: 2021-02-20 10:27:53 +0100 (sab, 20 feb 2021) $
!
!*********************************************************************

use ModMPMData
use ModGlobalConstants
use ModReadCalculationData
use ModReadMaterialData
use ModMPMInit
use user32
use kernel32
use ModMeshInfo
use ModLinearElasticity
use ModMohrCoulomb
use ModBingham
use ModNonLinearElasticity !add your module here!
```

5.5.3 Definition of the pointer to the ESM in "ReadMaterialData.for"

The ModReadMaterialData included in "ReadMaterialData.for", should be modified when in the case (ESM_EXTERNAL_SOIL_MODEL). To do this, search for "ESM_EXTERNAL_SOIL_MODEL" case and add the line defining the pointer diraction in case your model is associated to the materials. Note that you can have different material names pointing to different ESM:

if (MatParams(I)

```
                    case (ESM_EXTERNAL_SOIL_MODEL)

!if (MatParams(I)%MaterialName=="MyModel") MatParams(I)%ESM_POINTER => ESM_Mymodel
!Example of the pointer definition
if (MatParams(I)%MaterialName=="MyModel") MatParams(I)%ESM_POINTER => ESM_NONLINEAR
```

Note that "MyModel" should match the name the user should put in GiD when the material is created.

Compile the code as is described in Chapter 2.

5.5.4 Definition of a External Soil Model in Gid

In order to use the ESM, the option External Soil Model has to be selected in the material dialogue box and in which the parameters of the model parameters and initial state variables are specified (Figure 5.14). Note that the name of the model ("MyModel" in our example) has to be properly defined in the specified option box. Based on the ESM formulation (3D or 2D), the size of the stress tensor should be specified.

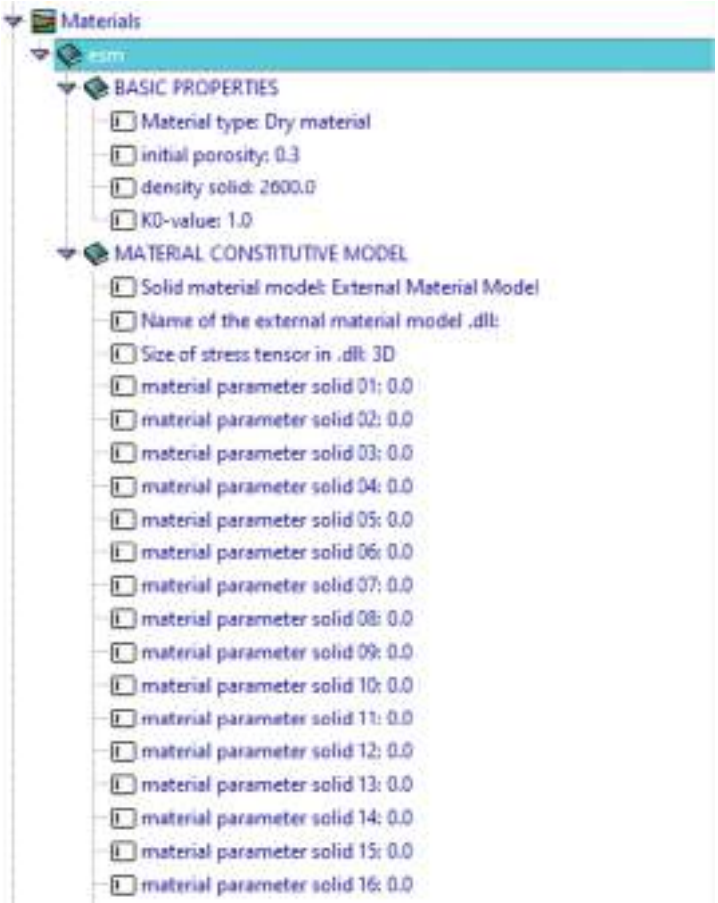


Figure 5.14: *Definition of external soil model in GiD.*

Results from the example of External Soil Model application

The same simulation as the previous example is run with this ESM. As the stiffness of the material changes in the computation, the critical time step is decreases as shown in Figure 5.15.

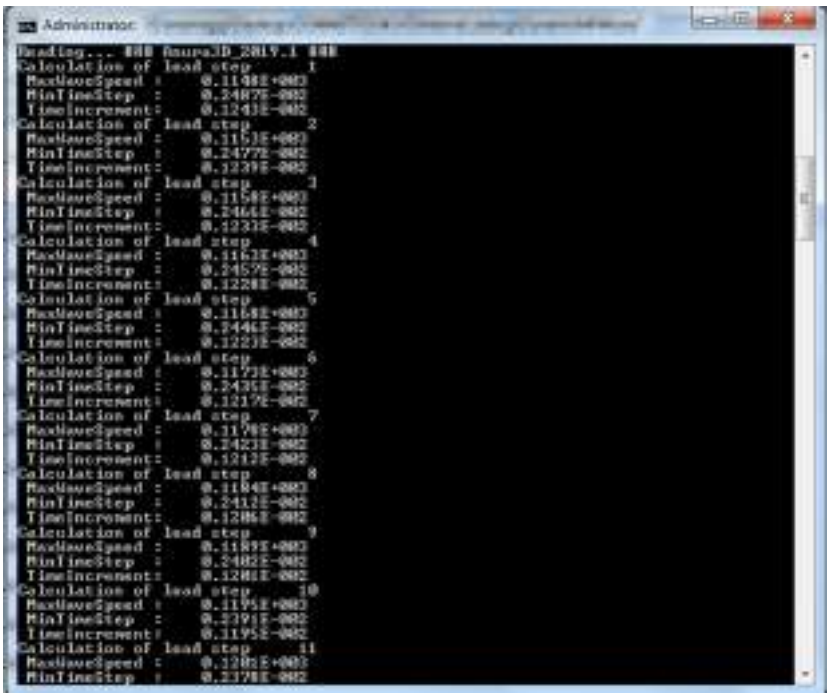


Figure 5.15: *Time step evolution during calculations.*

The increase in stiffness can be detected even plotting the average mean pressure over the sample with the average volumetric strain, as can be seen in (Figure 5.16).

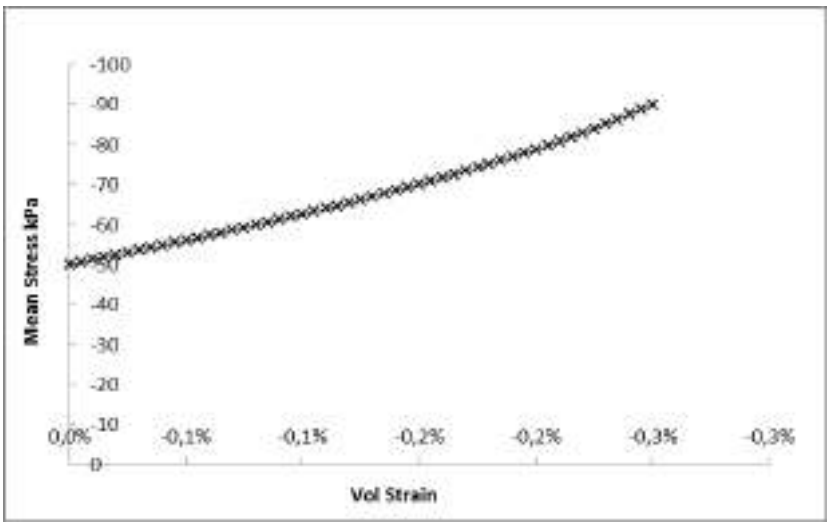


Figure 5.16: *Mean pressure agains volumetric strain. Increase of the material stiffness.*

6 Sliding blocks

Authors: Alba Yerro; Alexander Chmelnizkij; Alexander Rohe; James Fern; John Murphy; Luis E. Zambrano-Cruzatty; Tong Zhang

This tutorial demonstrates the use of the contact formulation with three blocks sliding on an inclined plane. Different friction and adhesion parameters will be used illustrating their influence on the sliding blocks. Finally, the kinematics of the blocks could be compared with the analytical solution for an infinite slope problem.

6.1 Problem definition

Three blocks are initially located at different heights on an inclined plane. The plane is divided in two different sections – the lower part and the upper part. The inclined plane ends in a horizontal plane. The geometry and the dimensions of the problem are given in Figure 6.1 and Table 6.1. Large deformation MPM models require defining empty spaces in which the material points are free to move in and these spaces have to be defined in the model and meshed. The contact algorithm will be applied between each sliding block and the plane. The contact formulation implemented in Anura3D allows the different contact parameters along the contact surface. Hence, different friction angles and adhesion values will be specified for each combination. All materials in this analysis are considered as linear elastic and in dry conditions. The properties for the materials are given in Table 6.2.

The calculation will be carried out in two stages.

(1) initialisation of stresses by quasi-static gravity loading,
(2) sliding of blocks down the plane by switching on the contact formulation.

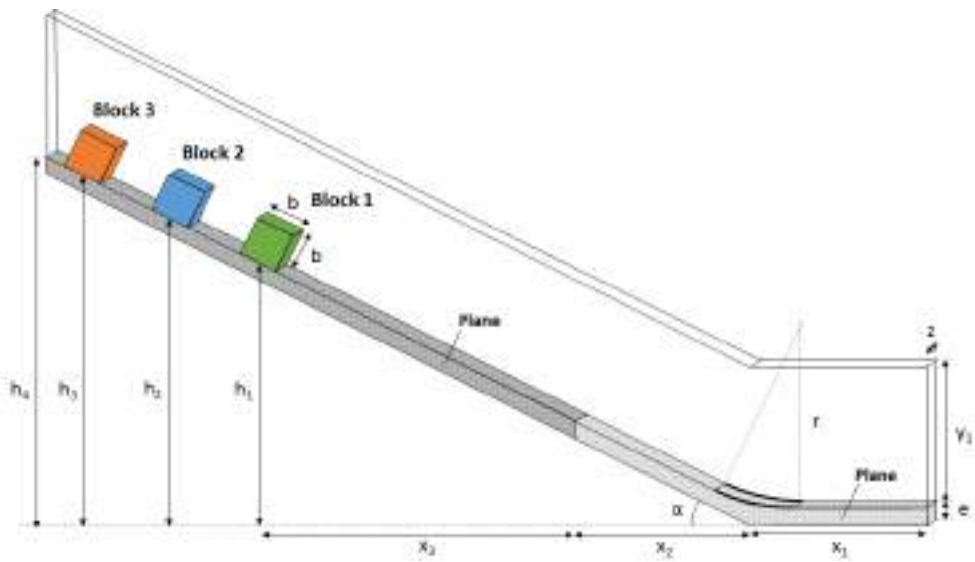


Figure 6.1: *Problem definition for the sliding blocks problem.*

Table 6.1: *Dimensions of the sliding blocks problem.*

dimension	value
r	5m
e	0.5m
z	0.0m
h_1	7m
h_2	8.25m
h_3	9.5m
h_4	10m
x_1	5m
x_2	5m
x_3	9m
y_1	4m
b	1.118m
α	26.56°

Table 6.2: *Material parameters for the sliding blocks and the soil.*

material	**block 1**	**block 2**	**block 3**	**plane**
material type	dry material	dry material	dry material	dry material
initial porosity [–]	0.0	0.0	0.0	0.0
density solid [kg/m^3]	2000	2000	2000	4000
K0-value [–]	0.0 (*)	0.0 (*)	0.0 (*)	0.0 (*)
material model solid	Linear Elasticity	Linear Elasticity	Linear Elasticity	Linear Elasticity
Poisson ratio [–]	0.33	0.33	0.33	0.33
Young modulus [kPa]	20000	20000	20000	40000

(*) not used

6.1.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "SlidingBlocks").

6.1.2 Defining geometry

Create the geometry considering the dimensions given in Table 6.1 and follow the steps described below.

(1) Define points and lines of one the faces of the model (Figure 6.2) using the point coordinates indicated in Table 6.3.

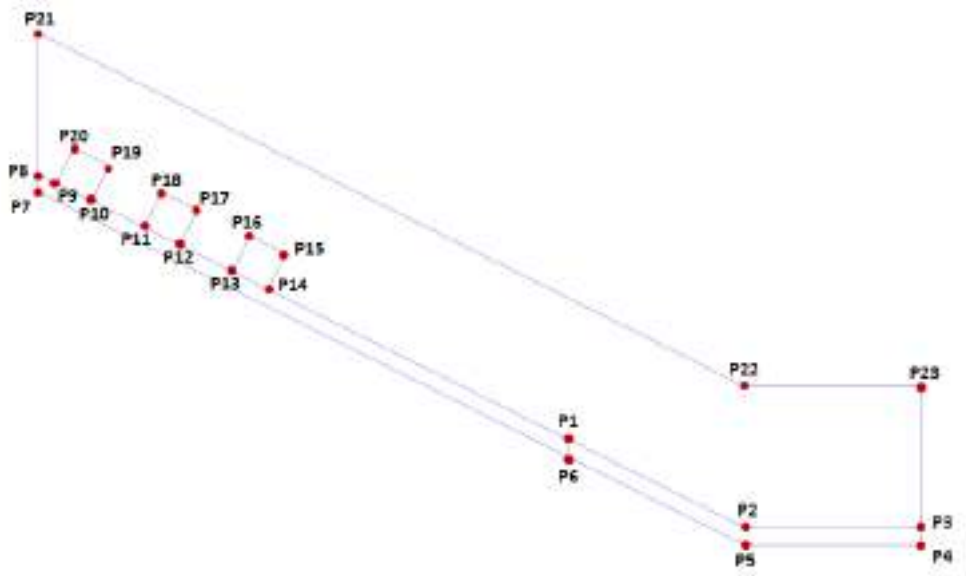


Figure 6.2: *Geometry of the front face of the model. Key points are indicated.*

Table 6.3: *Key point coordinates of front faces of the sliding blocks problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	-5	2.5
P2	0	0
P3	5	0
P4	5	-0.5
P5	0	-0.5
P6	-5	2
P7	-20	9.5
P8	-20	10
P9	-19.5	9.75
P10	-18.5	9.25
P11	-17	8.5
P12	-16	8
P13	-14.5	7.25
P14	-13.5	6.75
P15	-13	7.75
P16	-14	8.25
P17	-15.5	9
P18	-16.5	9.5
P19	-18	10.25
P20	-19	10.75
P21	-20	14
P22	0	4
P23	5	4

(2) Smooth the sharp kink at the toe of the slope (Figure 6.3) by creating an arc tangent to two lines. Choose <Geometry> <Create> <Arc> <Fillet curves> as presented in Figure 6.4 to create an arc tangent to two lines.

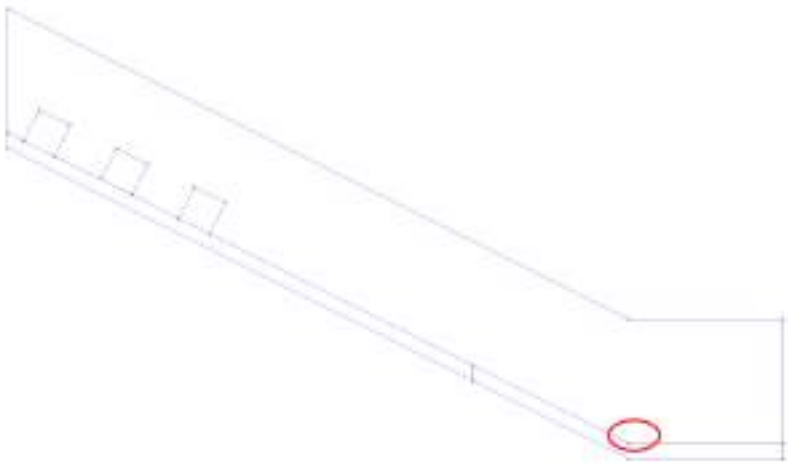


Figure 6.3: *Sharp kink at the toe of the slope.*

Figure 6.4: *Dialog box to create an arc the with the "Fillet curves" option.*

(3) Write the radius of the arc r indicated in Table 6.1 in the command line (Figure 6.5) and press <ENTER>.

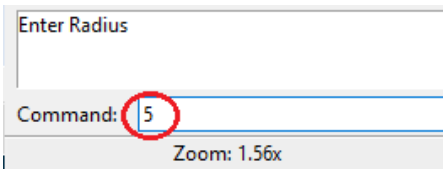


Figure 6.5: *Enter radius of the arc* r*.*

(4) Select the first tangent line (Figure 6.6) and press <ENTER>. Select the second tangent line and press <ENTER> (Figure 6.7). The sharp kink has been replaced by an arc as indicated in Figure 6.8.

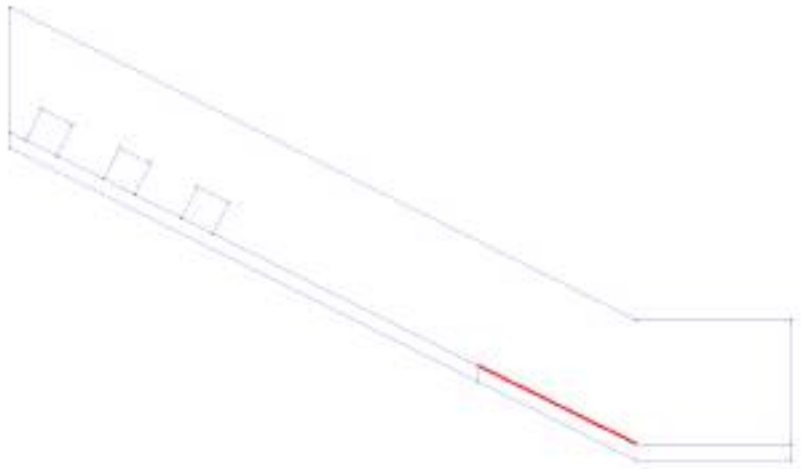


Figure 6.6: *Select first tangent line of the arc.*

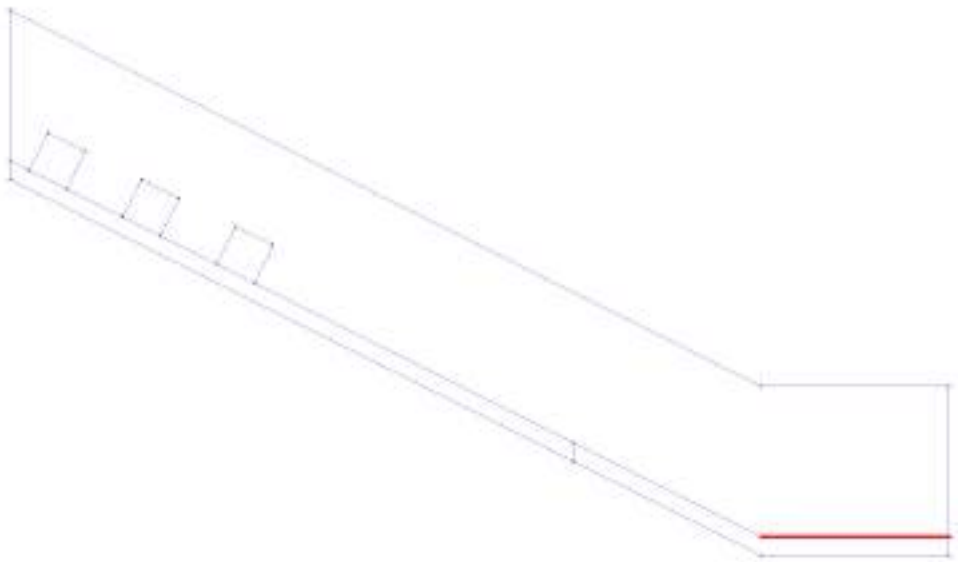


Figure 6.7: *Select second tangent line of the arc.*

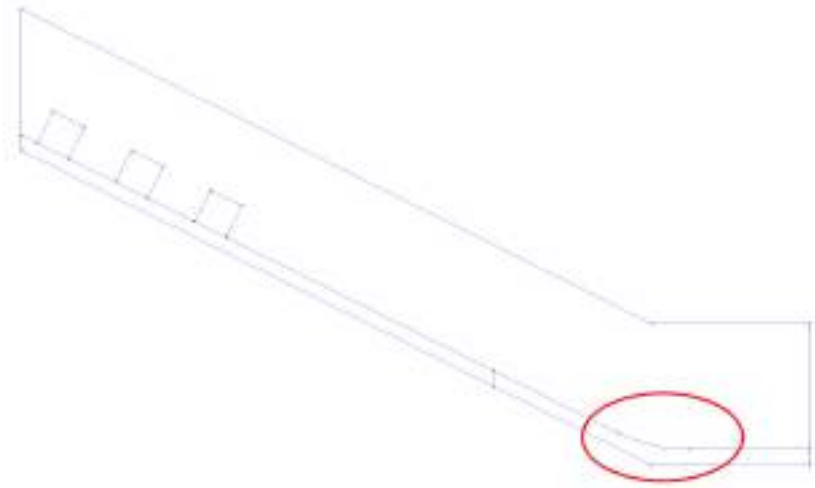


Figure 6.8: *Rounded kink at the toe of the slope.*

(5) Create surfaces following the explanations in Chapter 3.2.2. The final geometry should look like shown in Figure 6.9.

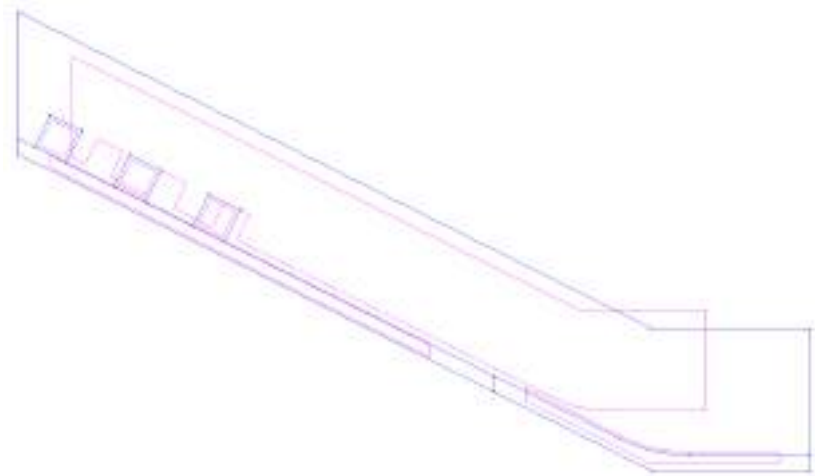


Figure 6.9: *Geometry of the sliding blocks problem.*

6.1.3 Defining general data

Under the *General data* menu go to *Problem dimensions* and set it to *2D:plane-strain*. Next, go to *MPM discretization type* and choose *single_point*, as the single point formulation will be used. In this example, ParaView post-processing visualization software is selected, as shown in Figure 6.10.

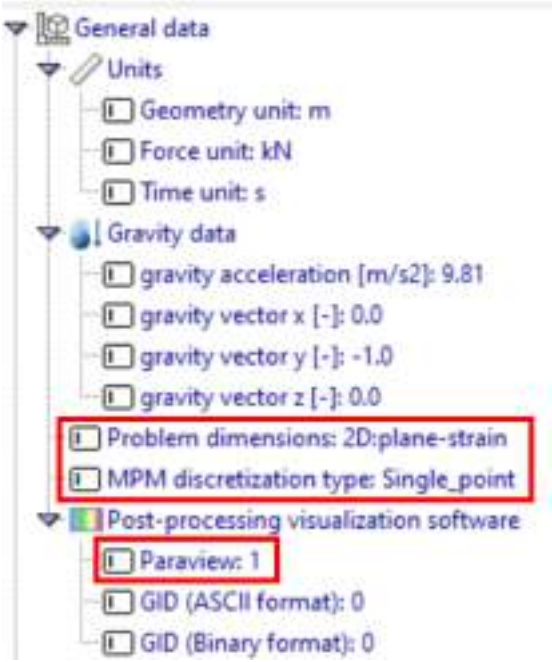


Figure 6.10: *General data for the sliding blocks example.*

6.1.4 Defining calculation stages

The calculation of the sliding blocks problem will be carried out in two stages. In the first stage (Stage 0) stresses are initialized by gravity loading, in the second stage (Stage 1) the contact formulation is applied and the blocks slide down the plane.

6.1.4.1 Stage 0: Gravity loading

In this stage, material properties, number of material points per element, and boundary conditions of the model are defined to initialize stresses by applying a gravity loading.

6.1.4.1.1 Defining material properties

The sliding blocks model consists of four different materials – one for each block ("block1", "block2", and "block3") and another one for the material conforming the slope ("plane"). Follow the instructions detailed in Section 3.2.5 to define each material. Material parameters are detailed in Table 6.2. Finally, ensure that the defined materials match the configuration shown in Figure 6.11.

Figure 6.11: *Materials of the sliding blocks problem.*

6.1.4.1.2 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. For this case, choose 3 material points per element for the plane and each of the blocks. Check if the material points are assigned correctly (Figure 6.12).

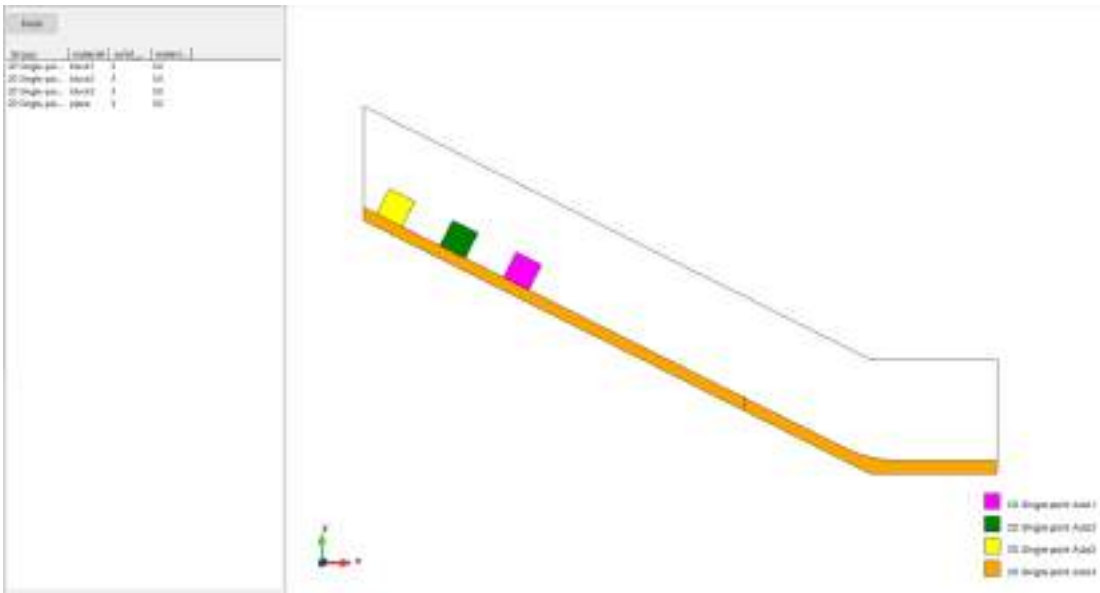


Figure 6.12: *Specification of the number of material points per element in the sliding blocks problem.*

6.1.4.1.3 Defining boundary conditions

Define the boundary conditions following the steps as described previously in Section 3.2.7.

Applying fixities

As the material is dry, only the solid phase needs to be considered. The solid displacement at the boundaries will be fixed as follows.

(1) Left side: horizontal fixity in x-direction on both lines
(2) Right side: horizontal fixity in x-direction on both lines
(3) Top side: vertical fixity in y-direction on both lines
(4) Bottom side: full fixities in x- and y-direction on all three lines
(5) Check if the x- and y- fixities are assigned correctly (Figure 6.13)

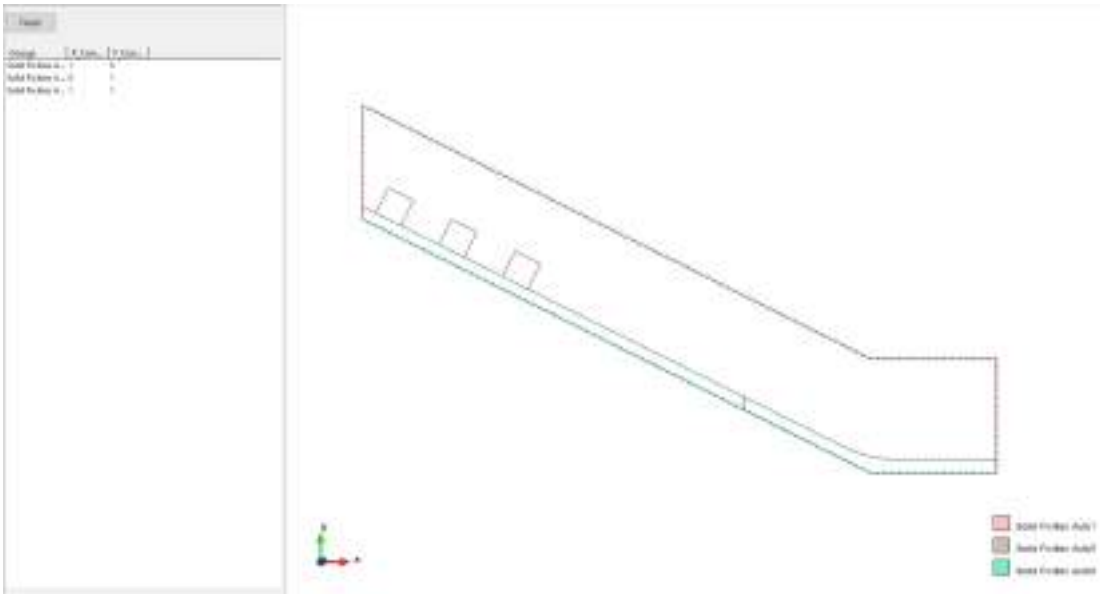


Figure 6.13: *Solid fixities of the sliding blocks problem in GiD.*

6.1.4.1.4 Defining calculation data

Computation method
The computation method used is the Gauss point integration. Therefore the option "MPM - mixed integration" should be selected.

Calculation step data
The duration of the initial stage of the calculation corresponding to the stresses initialisation will be automatically determined by the convergence criteria. The value for the time per calculation step is, therefore, not needed in this case. A Courant number of 0.9 is recommended to ensure stability of the solution.

Gravity load
Select *apply gravity load - stepwise*, initial stresses are zero by default and need to be initialised. In this example, the gravity is applied suddenly but the model will be heavily damped. Therefore, set the multipliers for gravity loading as 1.0 (initial and final value). This is maintained constant throughout the calculation. In this example, there is no additional external load. Therefore, the multipliers for external load can be kept at default values.

Quasi-static convergence
The determination of the end of the gravity loading is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition. The convergence criteria are the tolerated error of the out-of-balance force, the tolerated error of the kinetic energy or the maximum number of time steps.

Homogeneous local damping
In order to converge to the quasi-static solution, an overall local damping coefficient for all active elements of 0.75 is applied during the first stage of the calculation.

Default values
All other calculation options are not used in this example and the provided default values can

be kept, i.e. gravity data, smoothing algorithms, quasi-static convergence, etc.

To define the initial calculation parameters, follow the instructions detailed in Section 3.2.9. For this example, enter the values given below as shown in Figure 6.14.

- ◇ select "COMPUTATION METHOD" as "MPM - mixed integration" from the drop down menu
- ◇ set "number of calculation steps [-]" to "1"
- ◇ set "time per calculation step [s]" as "1.0". Note that any value except 0.0 can be entered as the quasi-static convergence criterion is used.
- ◇ set "Courant number" to "0.9"
- ◇ select "GRAVITY LOAD" as "apply gravity load - stepwise"
- ◇ set "gravity multiplier (initial, final) [-]" to "1.0 1.0"
- ◇ set "QUASI-STATIC CONVERGENCE" as "apply convergence criteria" from the drop down menu
- ◇ set "HOMOGENEOUS LOCAL DAMPING" as "apply homogeneous local damping" from the drop down menu
- ◇ set "local damping coefficient [-]" to "0.75"

All other calculation parameters can be kept at their default values.

Figure 6.14: *Dialogue box to define calculation parameters in GiD for the sliding blocks problem Stage 0.*

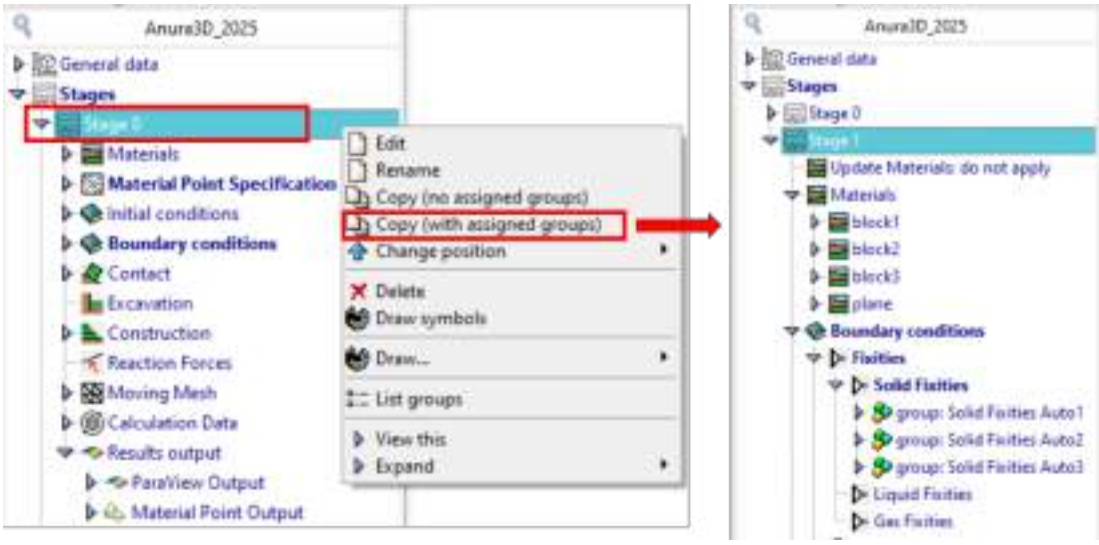


Figure 6.15: *Creation of Stage 1.*

Table 6.4: *Contact parameters between the sliding blocks and each surface of the base plane.*

		base plane	
		upper part	**lower part**
block 1	friction	0.3	0.45
	adhesion [kPa]	0.0	0.0
block 2	friction	0.325	0.45
	adhesion [kPa]	0.0	0.0
block 3	friction	0.35	0.45
	adhesion [kPa]	0.0	0.0

6.1.4.2 Stage 1: Sliding of the blocks

Since all materials, material point settings, and model boundary conditions does not change from *Stage 0*, these settings will continue to be used in *Stage 1*. By right-clicking on *Stage 0* and selecting *Copy (with assigned groups)*, the data tree of *Stage 1* can be generated (Figure 6.15). In this stage, the contact will be applied, allowing the blocks to slide along the plane. It should be noted that in *Stage 1*, only *Contact* needs to be defined and some calculation parameters need to be modified, while the others features remain unchanged.

6.1.4.2.1 Defining contact properties

The contact formulation implemented in Anura3D allows the definition of different contact properties (friction angle and adhesion) for each sliding material. These properties can also change in each segment of the contact surface. For this exercise, there are three different sliding materials (block 1, block 2, and block 3) and two sections in the base plane (the upper part and the lower part).

Follow the instructions as detailed below to assign the contact properties between blocks and base plane.

(1) Choose <Contact → Contact Properties> from the menu as shown in Figure 6.16.

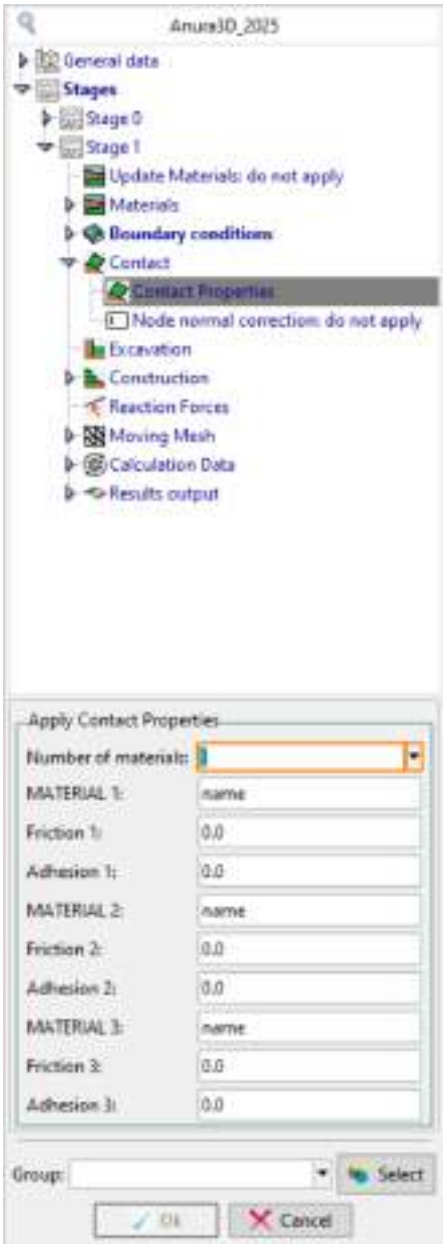


Figure 6.16: *Select <Contact Properties>.*

(2) Select the "number of materials" as "3" since there are three sliding blocks as shown in Figure 6.16.

(3) Name the contact Materials "MATERIAL 1" etc. according to the the three blocks. The name must be **exactly** the same name of the material defined in Section 6.1.4.1.1. Otherwise, the contact will not be properly assigned. Define the friction and adhesion between each block and the upper part of the slope filling the spaces below each material name. The contact properties are specified in Table 6.4. Note that the term included in "MATERIAL" must be **exactly** the same name of the material defined in Section 6.1.4.1.1. Otherwise, the contact will not be properly assigned (Figure 6.17).

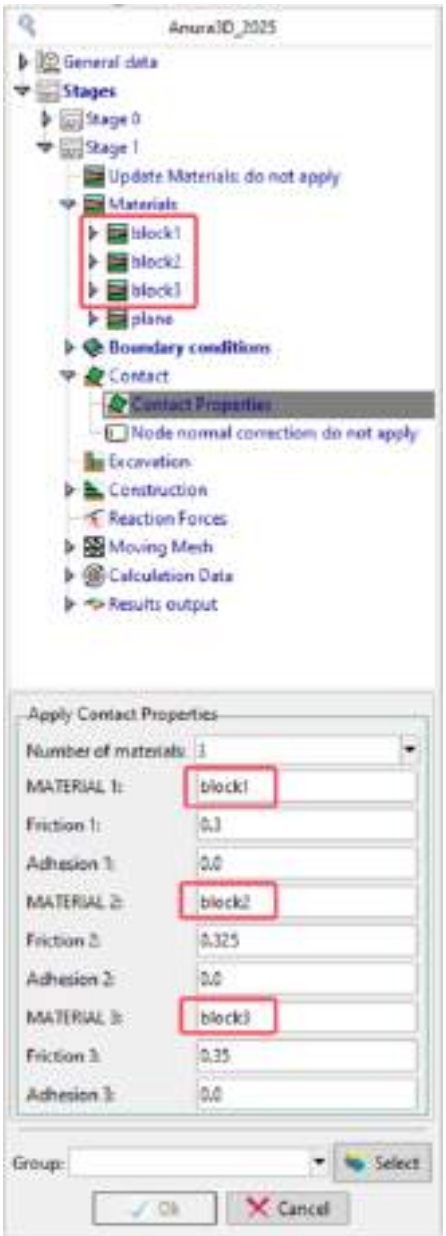


Figure 6.17: *Select "Number of materials" and specify the name.*

(4) Click on the button <Select> and select the surface corresponding to the upper part of the slope as shown in Figure 6.18.

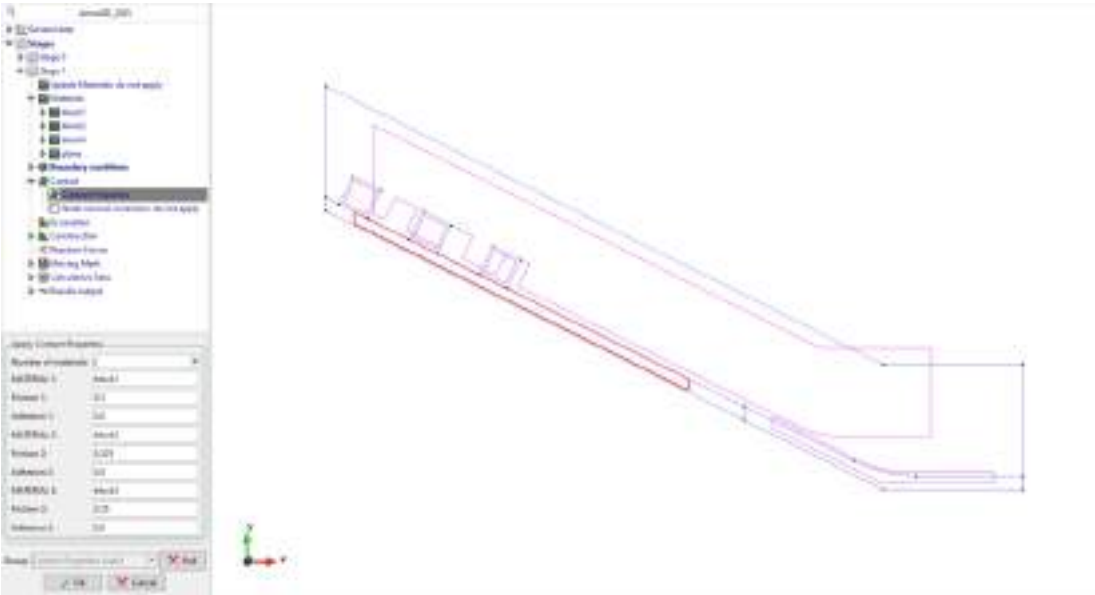


Figure 6.18: *Assign contact properties between each sliding block and the upper part of the slope.*

(5) Press <Ok> to save the contact properties for the upper part of the slope.

(6) Define the friction and adhesion between each block and the lower part of the slope by rewriting the contact properties as shown in Figure 6.19 and according to the values specified in Table 6.4.

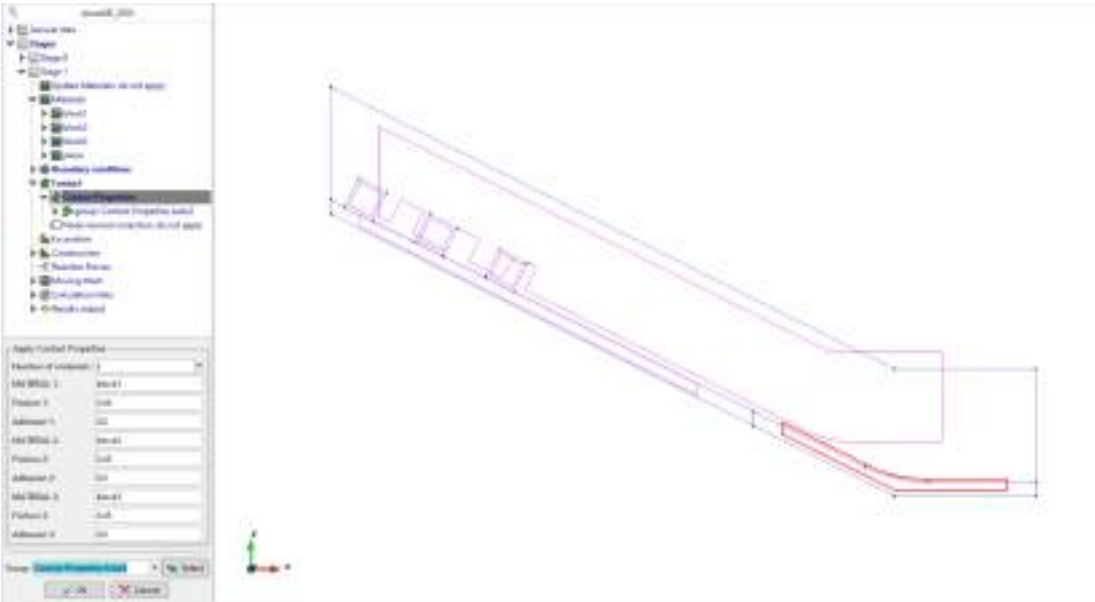


Figure 6.19: *Assign contact properties between each sliding block and the lower part of the slope.*

(7) Click on the button <Seletc> and select the lower surface of the soil layer. The selected surface should turn red.

(8) Press <Esc> and click on <Ok> to save the contact properties for the lower part of the slope.

(9) Right-click on <Contact Properties> an select <Draw → Draw values> to check if the contact properties are properly assigned. It should look like Figure 6.20.

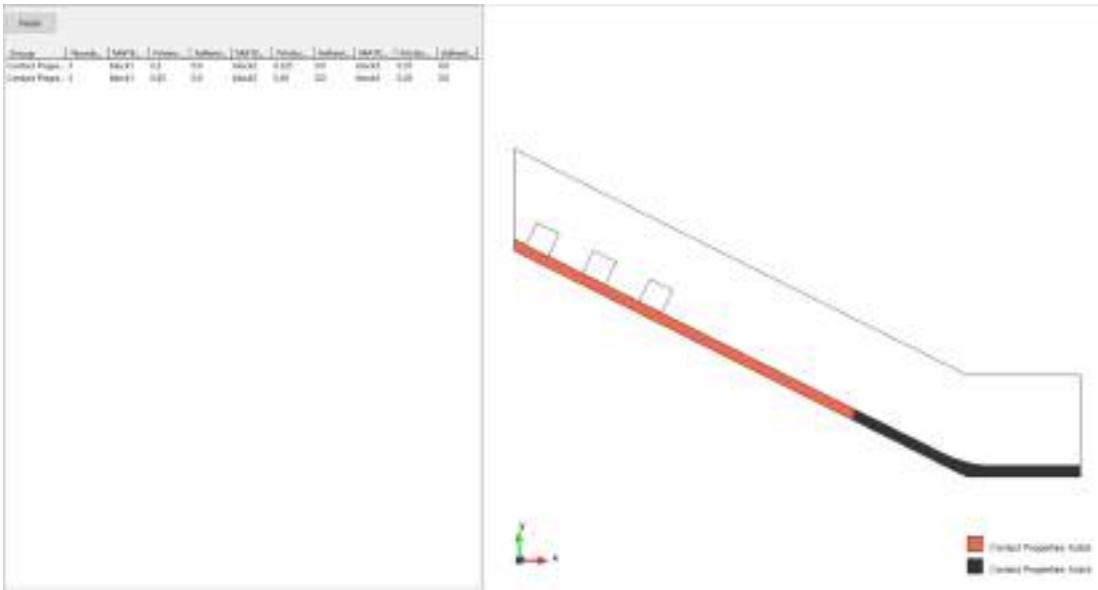


Figure 6.20: *Definition of the contact properties between each sliding block and the slope.*

6.1.4.2.2 Defining calculation data

Figure 6.21 shows the calculation data of *Stage 1*.

Calculation step data
In this stage, 70 additional load steps will be calculated. The time per load step is set as 0.1 seconds to be able to follow (and visualize) the sliding of the blocks in small steps. Set *number of calculation steps [-]* to 71 and *time per calculation step [s]* to 0.1.

Quasi-static convergence
This stage is performed through a dynamic calculation. Thus, set the *QUASI-STATIC CONVERGENCE* to *do not apply convergence criteria*.

Homogeneous local damping
The local damping coefficient is reduced to 0.01.

Figure 6.21: *Dialogue box to define calculation parameters in GiD for the sliding blocks problem Stage 1.*

6.1.5 Creating mesh

It is possible to generate meshes in many different ways in GiD. A unstructured mesh is used for this example. To generate the mesh follow the steps detailed below.

(1) Choose <Mesh> <Generate mesh...> from the menu as shown in Figure 6.22.

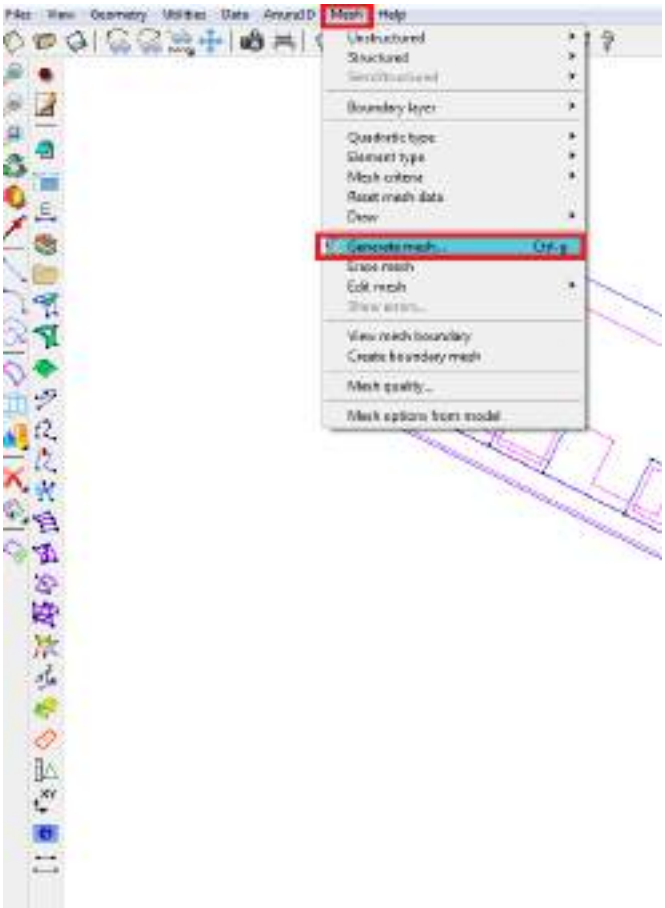


Figure 6.22: *Select <Mesh> <Generate mesh...>.*

(2) Enter the value "0.3" in the new dialogue box to indicate the mesh size for the lines as shown in Figure 6.23.

Figure 6.23: *Enter the number of divisions and assign it to semi-structured volumes.*

(3) Click the <OK> button. The mesh will be generated and the progress is shown in the "Progress in meshing" dialogue box as shown in Figure 6.24. The number of generated tetrahedral elements can be seen as well as the number of generated nodes.

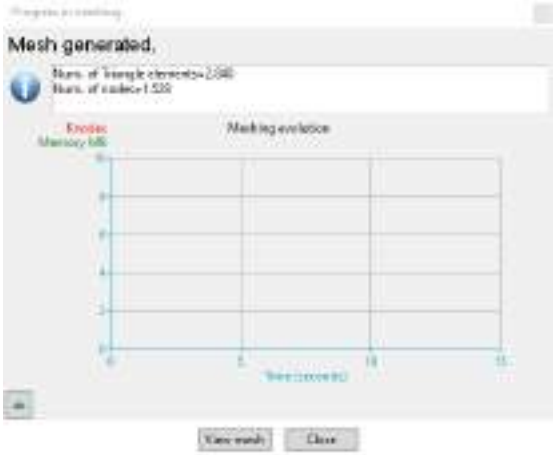


Figure 6.24: *Mesh generated dialogue box.*

(4) Click on the <View mesh> button to quit the meshing process and see the generated mesh. It should look like Figure 6.25

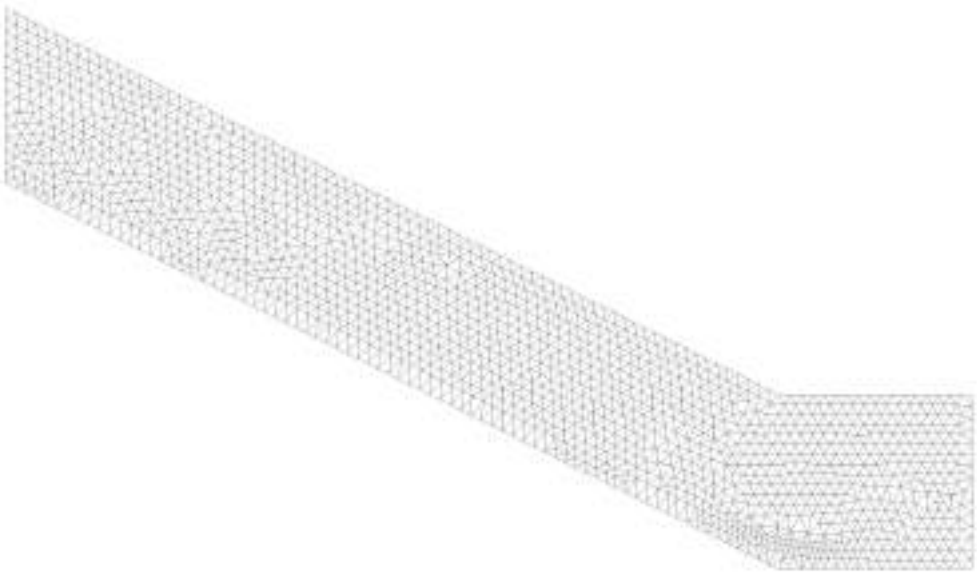


Figure 6.25: *Computational mesh of the sliding blocks problem.*

6.2 Performing calculation in Anura3D

The calculation will be carried out in two stages as follow:

1) initialisation of stresses by quasi-static gravity loading,
2) sliding of blocks down the plane by switching on the contact formulation.

To start the calculation go to <Anura3D → Calculate> (Figure 6.26). The two stages will be computed automatically.

Figure 6.26: *Calculation of the stages.*

The generated input files for *Stage 0* of Anura3D simulation can be found in the newly created folder `SlidingBlocks.A3D`, within the directory specified earlier by the user (Sec. 6.1.1):

D:\..\SlidingBlocks.A3D\SlidingBlocks.GOM_stage0

D:\..\SlidingBlocks.A3D\SlidingBlocks.OPD_stage0

D:\..\SlidingBlocks.A3D\SlidingBlocks.CPS_001

The calculation of *Stage 0* should be finished in a few seconds depending on the performance of your PC. Once calculation of *Stage 0* is finished the necessary files for *Stage 1* are generated automatically:

D:\..\SlidingBlocks.A3D\SlidingBlocks.GOM_stage1

D:\..\SlidingBlocks.A3D\SlidingBlocks.OPD_stage1

D:\..\SlidingBlocks.A3D\SlidingBlocks.CPS_002

The calculation of *Stage 1* should be finished in about three minutes depending on the performance of your PC. The output files are generated and stored in the project directory `D:\..\SlidingBlocks.A3D`.

6.3 Visualization of results in ParaView

After each step of calculation, a corresponding output file (VTK file) will be generated, which can be visualized in ParaView. Instructions presented in Section 3.4 can be followed to open and analyse the results.

In *Stage 1*, the contact formulation is applied allowing the blocks to slide along the plane. Sliding displacements at calculation step 35 and the final step are shown in Figure 6.27 and Figure 6.28, respectively.

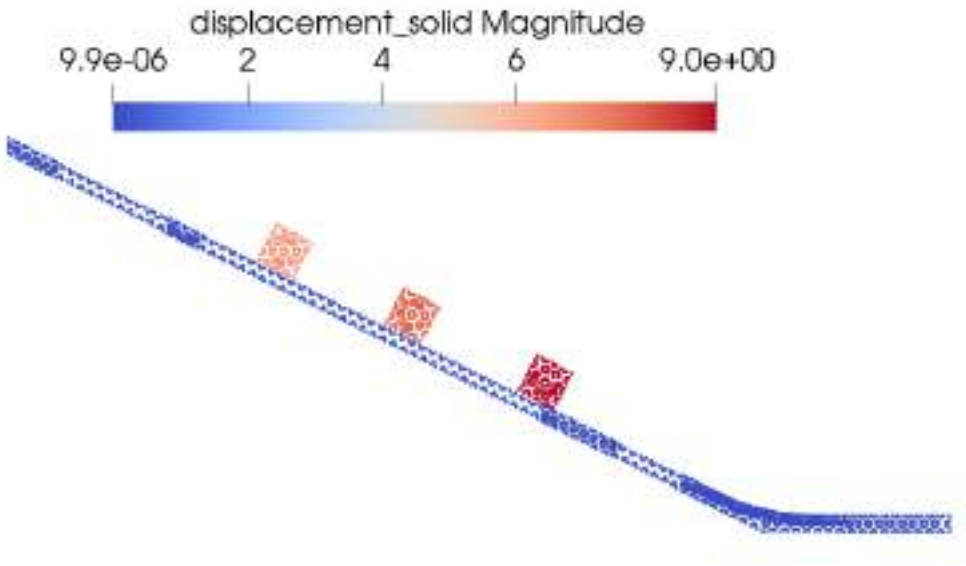


Figure 6.27: *Total displacement at calculation step 35.*

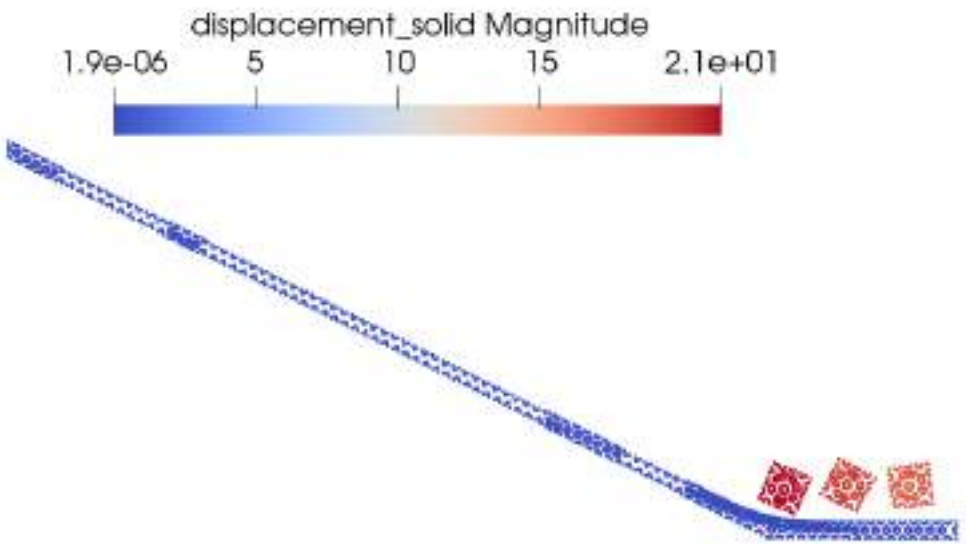


Figure 6.28: *Total displacement at the end of the simulation.*

Following the steps described in Section 3.4, the three material points, as shown in Figure 6.29, will be examined.

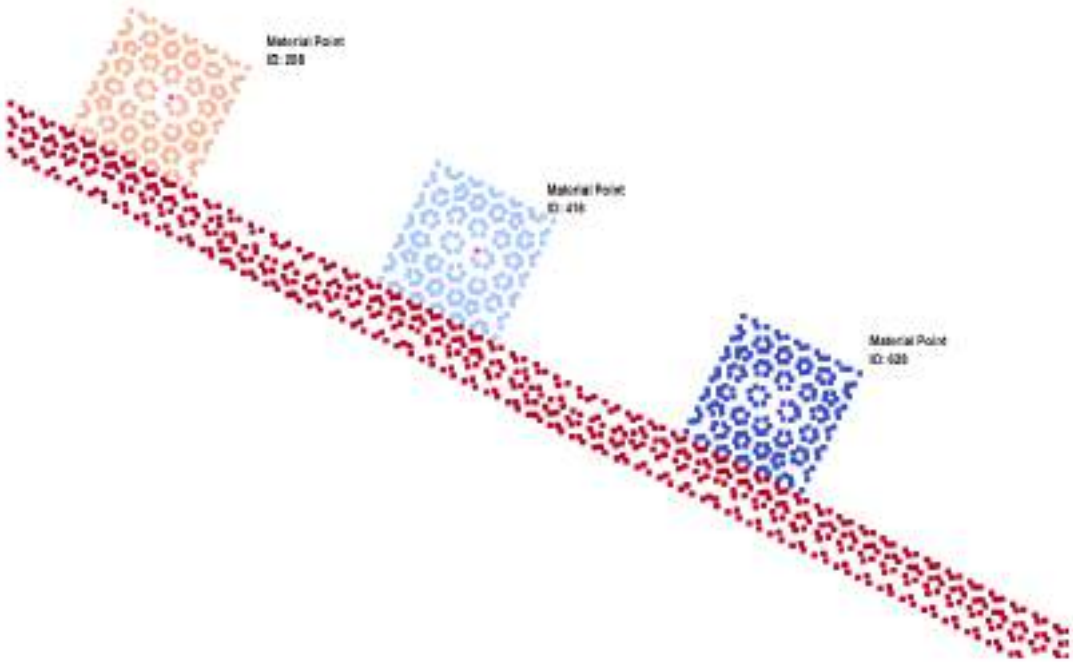


Figure 6.29: *Location of material points 208, 418 and 628.*

The output in ParaView can be exported to different types of files. To compare the numerical results of the sliding block problem with analytical solutions for an infinite slope the results are saved as .csv files. This can be done by choosing <File> <Save Data...> from the main menu. The output file can be generated either for one specific or for all load steps (Figure 6.30).

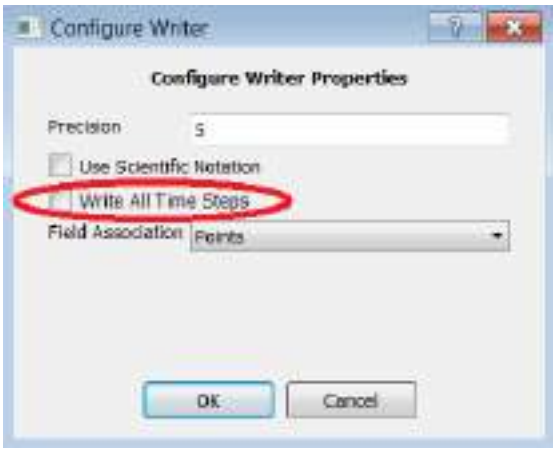


Figure 6.30: *Options for saving output files in ParaView.*

These files can be imported for example in Matlab or MS Excel. The analytical solution for the horizontal displacement of a sliding block on an infinite slope with an inclination α and a friction coefficient μ is

$$s_x = 0.5 \cdot \Big(g \cdot \big(\sin(\alpha) - \mu \cdot \cos(\alpha)\big)\Big) \cdot \cos(\alpha) \cdot t^2 \tag{6.1}$$

where g denotes the gravity.

6.4 Variations

The variations proposed for the sliding blocks problem are listed below.

(1) Examine the influence of different values of friction angle and adhesion in the sliding kinematics of the blocks.
(2) Examine the behaviour of the blocks depending on their Young's moduli.
(3) Use different material constitutive models for the blocks (i.e. Mohr-Coulomb).
(4) Explore the behaviour of the blocks depending on the curvature at the kink, by changing the radius of the arc.

To perform each variation, you can copy the original project, and save it using another name (e.g. `blocks_var1.A3D`, `blocks_var2.A3D`, ...). Then, change the corresponding contact properties, material properties or geometry in GiD following instructions from Section **??**, and repeat the calculation process in two stages described in Section 6.2.

Furthermore the presented 2D problem can be expanded to a 3D calculation using the "3D-Body contact" and defining all conditions on surfaces and volumes instead an lines and surfaces. In Figure 6.31 the 3D version of the presented problem, with a depth of 0.5 m, is shown.

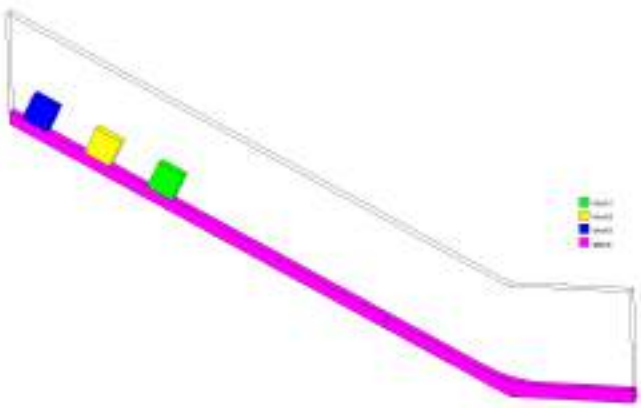


Figure 6.31: *Three-dimensional variation of the sliding blocks problem.*

6.5 Discussion

Note that the contact formulation implemented in Anura3D considers rigid contact surfaces. This means that to use the contact formulation in a proper manner, you should be able to distinguish two types of material in your problem: the "base" material (which should not deform), and the "sliding" material or materials (which slide on top of the "base"). With this configuration, potential contact surfaces are determined at the very beginning of the calculation. The contact surface geometry is not updated during the calculation, hence the computation time is optimised.

7 Column collapse

Authors: Alba Yerro; Alexander Rohe; James Fern; Amine Aboufirass; Marc Stapelfeldt; Gaia Di Carluccio; John Murphy; Pietro Marveggio; Veronica Girardi; Francesca Ceccato

This tutorial explains how to simulate the collapse of a granular column. The granular column collapse is a well-established experiment which consists in releasing a column of sand on a flat surface and looking at the liquid-like flow and final deposition profile.

The current version of Anura3D provides two-dimensional plane strain and axisymmetric as well as three-dimensional analyses. This exercise contains a two-dimensional example, which can be modified for axisymmetric or three dimensional calculations.

7.1 Problem description

The dimensions of the problem are given in Figure 7.1. Note that the area into which the material is supposed to flow has to be included in the mesh discretisation to allow material points to move freely. A base layer is included for basal friction. The dimensions are given in Table 7.1.

This analysis consists of a one-phase calculation. Therefore, both the column and base will be modelled as dry materials. The material properties are given in Table 7.2.

The calculation will be performed in two stages:

1) stress initialisation with quasi-static gravity loading
2) failure of the column by removal of column fixities

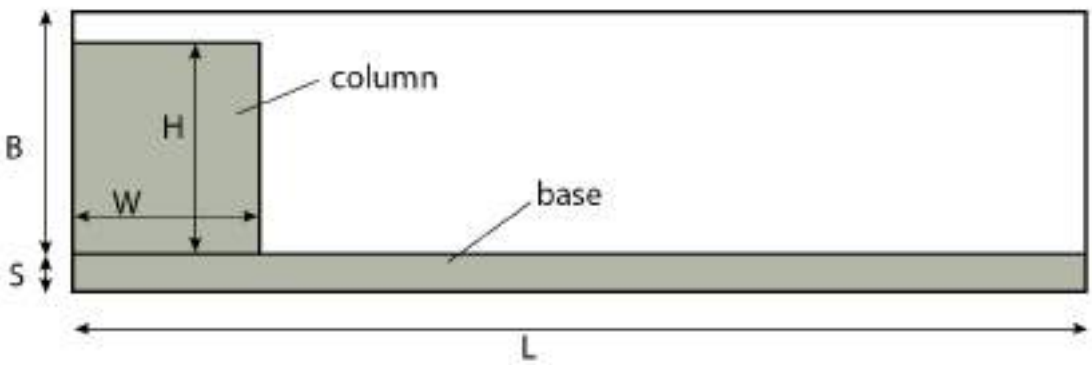


Figure 7.1: *Problem definition for the case 2D of a collapsing column including base layer.*

Table 7.1: *Dimensions of the collapsing column.*

ratio	column height H [m]	column width W [m]	depth D [m]	box height B [m]	box width L [m]	base layer S [m]
2:1	2.0	1.0	0.1	2.1	10.0	0.2

Table 7.2: *Material parameters of the collapsing column.*

material	**column**	**base**
material type	dry material	dry material
initial porosity [–]	0.4	0
solid density [kg/m^3]	2650	2650
K_0-value [–]	0.0 (*)	0.0 (*)
material model solid	Mohr-Coulomb	Linear Elasticity
Poisson ratio [–]	0.3	0.3
Young's modulus [kPa]	10000	10000
cohesion [kPa]	0	-
friction angle [°]	30	-
dilatancy angle [°]	0	-
tensile strength [kPa]	0	-

**not used*

7.2 Creating input data in GiD

The geometry, boundary conditions, material properties, and mesh are defined using the GiD pre-processor. The calculation options will be defined in the *CPS* file. Both follow the same procedure as previous examples. Note that this geometry is defined in 2D as the case described in Section 3.2.

7.2.1 Starting pre-processor and selecting problemtype

Start the GiD pre-processor following the steps as described previously in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e. g. "Column-Collapse") as shown in Figure 7.2.

Figure 7.2: *Projectname of the project "columncollapse" in GiD.*

Set the general data as shown in Figure 7.3.

Figure 7.3: *Define general settings*

7.2.2 Defining geometry

Create the geometry by following the steps as described in Section 3.2.2. The dimensions of the problem are given in Table 7.1. Unlike for Lagrangian FEM models, MPM models require the definition of empty volumes in which the material points can move into. One should think of the possible large deformed configuration of the material when defining the model geometry.

In this example, a two-dimensional column with an initial aspect ratio of 2 (= height/base) will be created by creating 6 rectangles as shown in Figure 7.4. A structured mesh will be utilised. The origin of coordinates will be the bottom left of the soil column (point P4 in Figure 7.4). The coordinates of the points is given in Table 7.3.

Figure 7.4: *Geometry of the two-dimensional column collapse problem for ratio 2:1 including empty entities on the right and top side of the column.*

Table 7.3: *Coordinates of points.*

point	X	Y	point	X	Y	point	X	Y
P1	0	0	P5	10.0	0	P9	10	2.2
P2	1.0	0	P6	10.0	0.2	P10	10	2.3
P3	1.0	0.2	P7	0	2.2	P11	1.0	2.3
P4	0	0.2	P8	1.0	2.2	P12	0	2.3

7.2.3 Defining calculation stages

This problem has two calculation stages:

1) Stress initialization. This is a quasi static gravity loading stage in which the column is constrained horizontally in order to initialize stresses. Gravity is applied in one load step.
2) Column collapse. In this stage the fixities on the right side of the column are removed and the material collapses propagating along the base. The collapse dynamic of the column is simulated with 50 load steps.

Start with the definition of the fist stage as described in the following.

7.2.4 Definition of the first stage (gravity loading)

At the first stage you have to define all material parameters and material point specifications as well as output material points if appropriate. In addition, the geometry should be created.

7.2.4.1 Defining material properties

Define the material properties as described previously in Section 3.2.5. The first example is for a column of dry soil. Two materials are defined – one for the base layer and one for the column. The base layer is linear elastic and the column has an elastic-plastic Mohr-Coulomb model. The model parameters are given in Table 7.2 and shown in Figure 7.5.

Figure 7.5: *Defining material properties for the column collapse problem in GiD.*

7.2.4.2 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. Choose 3 material points per element for the column and base. Check if the material and the number of material points are assigned correctly (Figure 7.6).

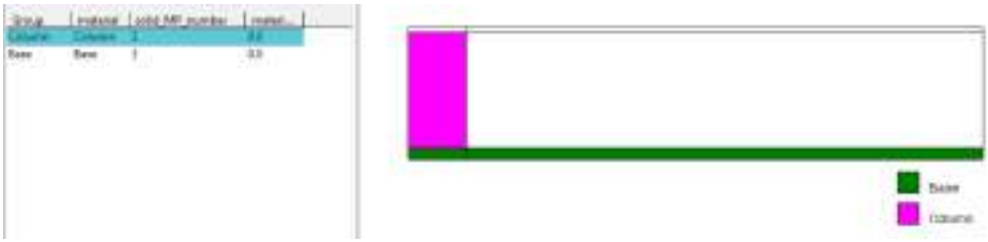


Figure 7.6: *Specification of the material and number of material points per element in the column collapse problem.*

7.2.4.3 Defining boundary conditions

Define the boundary conditions following the steps detailed below.

Applying fixities

Apply the fixities for the column collapse problem following the steps described previously in Section 3.2.7. Because this first example is a one-phase calculation, the boundary conditions are applied only on lines for solid phase.

(1) Left side at x = 0.0: horizontal fixity in x-direction on all three lines
(2) Right side at x = 10.0: horizontal fixity in x-direction on all three lines
(3) Right side of column at x = 1.0: horizontal fixity in x-direction on the middle surface
(4) Top side at y = 2.1: vertical fixity in y-direction on both lines
(5) Bottom side at y = -0.2: full fixities in x-, y-direction on both lines

Note that at the top boundary of the column itself no fixities are applied. The assigned solid fixities of the column collapse problem should look like shown in Figure 7.7.

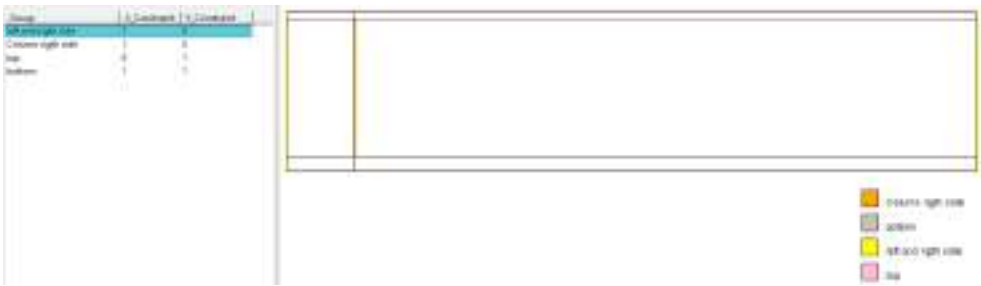


Figure 7.7: *Fixities of the two-dimensional column collapse problem in GiD.*

7.2.4.4 Creating mesh

A structured mesh will be used with an element size of 0.1m in each coordinate direction. The surfaces have to be selected in which the structured mesh should be created. Afterwards, the mesh size has to be assigned to the bounding lines by specifying the element length in each direction.

Double check that the quadratic type is <normal> in the menu <Mesh> <Quadratic type> as shown in Figure 7.8.

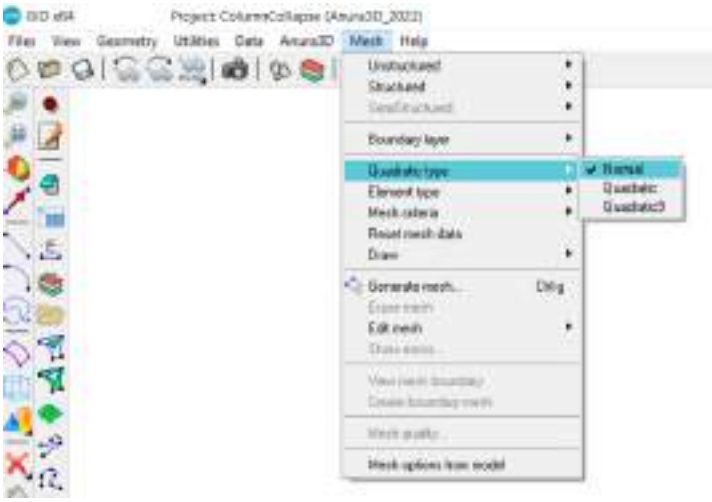


Figure 7.8: *Definition of the quadratic type in GiD.*

Choose the element type to be <Triangle> in the menu <Mesh> <Element type> as shown in Figure 7.9 and select all the surfaces to assign triangular elements.

Figure 7.9: *Definition of the quadratic type in GiD.*

The properties of the structured mesh are defined by selecting <Mesh> <Structured> <Surfaces> <Assign size to surface lines> from the menu as shown in Figure 7.10.

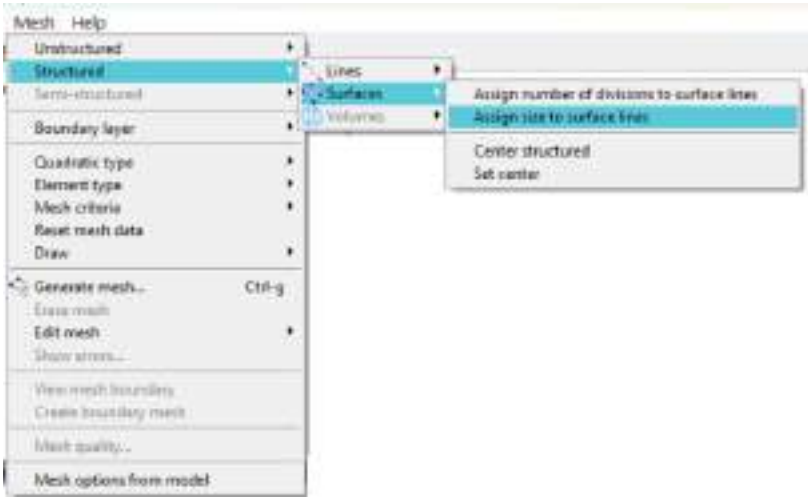


Figure 7.10: *Definition of structured mesh properties in GiD.*

With the appearing cursor, select all surface entities of the geometry. Each surface selected should turn red. When all surfaces are selected, press <ESC> to leave. In the appearing dialogue box, enter the element size that should be assigned to all lines, i. e. 0.1. Click on the button <Assign> and select all line entities bounding the volume entities with the appearing cursor. Each selected line should turn red, as shown in Figure 7.11.

Figure 7.11: *Assigning element size to entities.*

When all lines are selected, press <ESC> to leave. In the appearing dialogue box, click on the button <Close> to finish the definition of structured element size.

Continue the generation of the mesh as described in Section 3.2.8. The structured mesh with element size of 0.1m should look like shown in Figure 7.12.

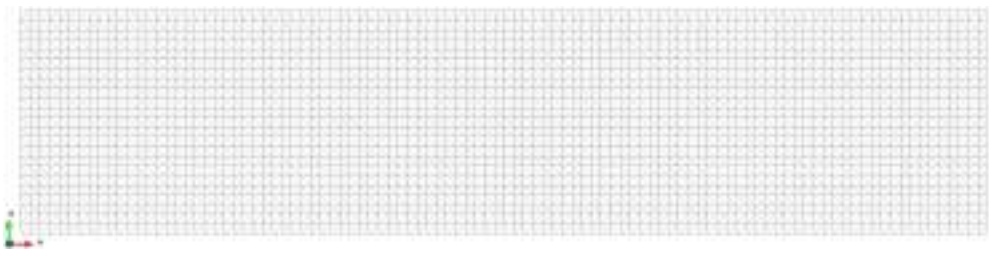


Figure 7.12: *Mesh in GiD for the case of a column with ratio 2:1 including empty entities.*

7.2.4.5 Definition of calculation data

This Section explains the calculation settings of the first stage of this problem, that corresponds with gravity loading for the generation of initial stresses.

Computation method
The computation method used in this example is the Gauss point integration. Therefore, the option "MPM - mixed integration" should be selected.

Calculation step data
The end of the calculation step will be determined by the convergence criteria (see below in the "Quasi-static convergence"), thus the "time per calculation step" is not used. A Courant number of 0.98 is recommended for the stability of the solution.

Gravity load
Apply gravity load stepwise and set the multipliers for gravity loading as 1.0 (initial) and 1.0 (final value).

Quasi-static convergence
Apply the quasi-static convergence and leave the default values. The end time of the gravity loading is achieved when a quasi-static equilibrium condition is reached. The convergence criteria are the tolerated error of the out-of-balance force, the tolerated error of the kinetic energy, or the maximum number of time steps. Only tolerated error for solid phase are used in one-phase simulation.

Homogeneous local damping
Apply homogeneous local damping with a local damping coefficient of 0.75. This is used to dissipate energy and converge to quasi static equilibrium faster.

Strain smoothing
Apply strain smoothing algorithm to minimize kinematic locking, which might occur for volumetric strains.

The other calculation parameters can be kept at their default values. Double check that the parameters are set in accordance to Figure 7.13.

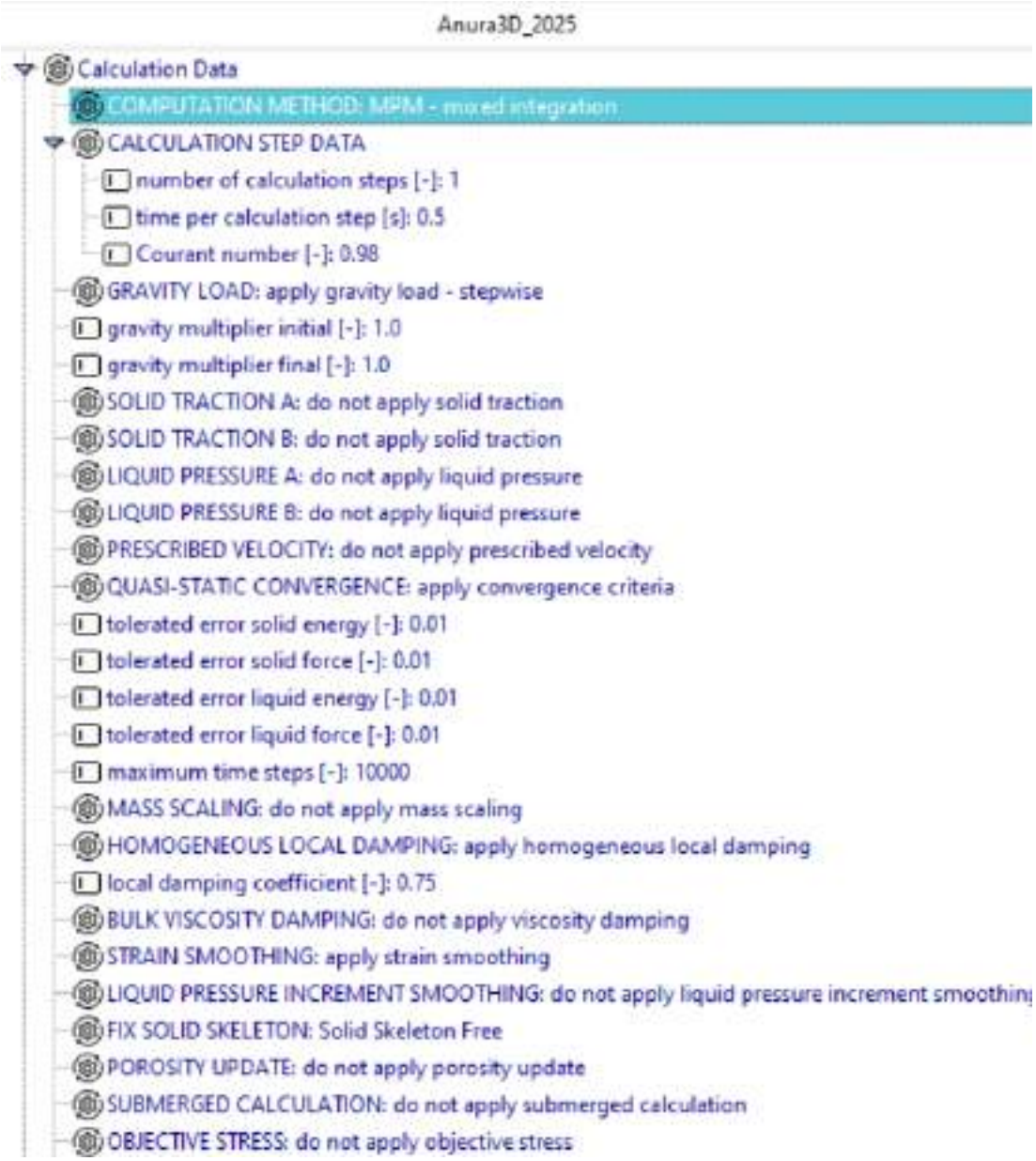


Figure 7.13: *Dialogue box to define calculation parameters in GiD.*

7.2.4.6 Define results output

With this menu you can select the desired quantities to inspect in the Paraview of GiD output. For this example you can leave the default values. In addition, you can define material points to follow throughout the simulation. To this end, make the mesh visible, then open the menu "Material Point Output" and click "Select elements for MPs output". Active the option "select" and clic on tre elements: one at the bottom of the column, one at the middle and one at the top. Set the "Number of output MPs per element" to 1 and leave the default value of "Time between output". See Figure 7.14 as a reference. Anura3D 2025 will automatically select 1 MP inside the selected elements and write a text file (.PAR) with relevant data for selected time intervals. Note that the selection of material Point Output can be done only in the first stage of the simulation. This option is disabled in the following stages. Do not regenerate the mesh after this assignement or the information will be lost and you need to redefine it.

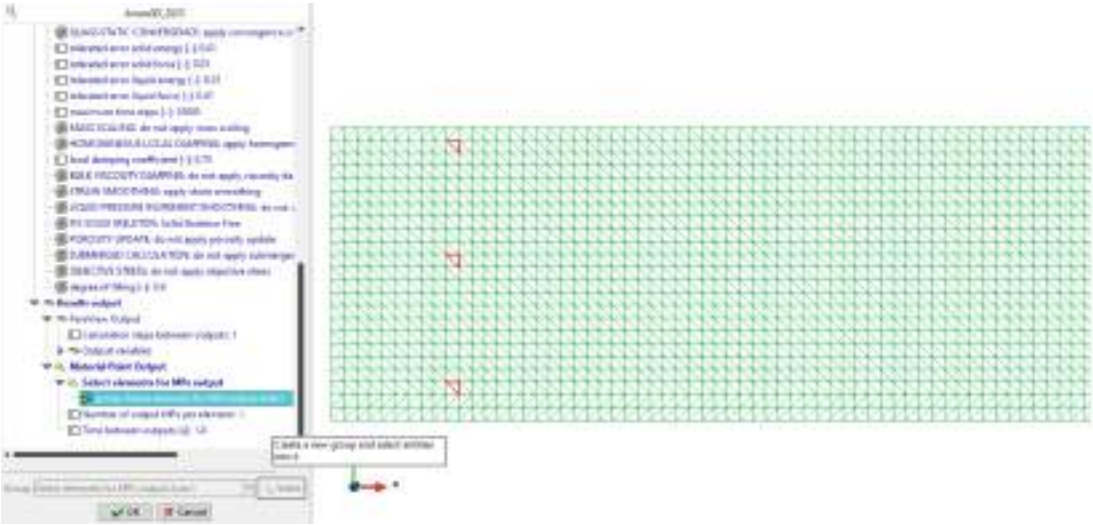


Figure 7.14: *Dialogue box to define calculation parameters in GiD.*

7.2.5 Definition of the second stage (column collapse)

Create a second stage and name it "Column collapse". In order to copy all the properties from the previous stage, rigth click on the first tage and select *Copy (with assigned group)* .

In this stage the fixities on the right side of the column are removed to release the material. This can be done by removing the corresponding group form the solid fixity menu. Mouse right click on the correct group, and choose "Unassing group" and then OK (see Fig. 7.15). If you inspect the fixities, you will see that the boundary condition on the right side of the column is not there anymore (Fig. 7.16).

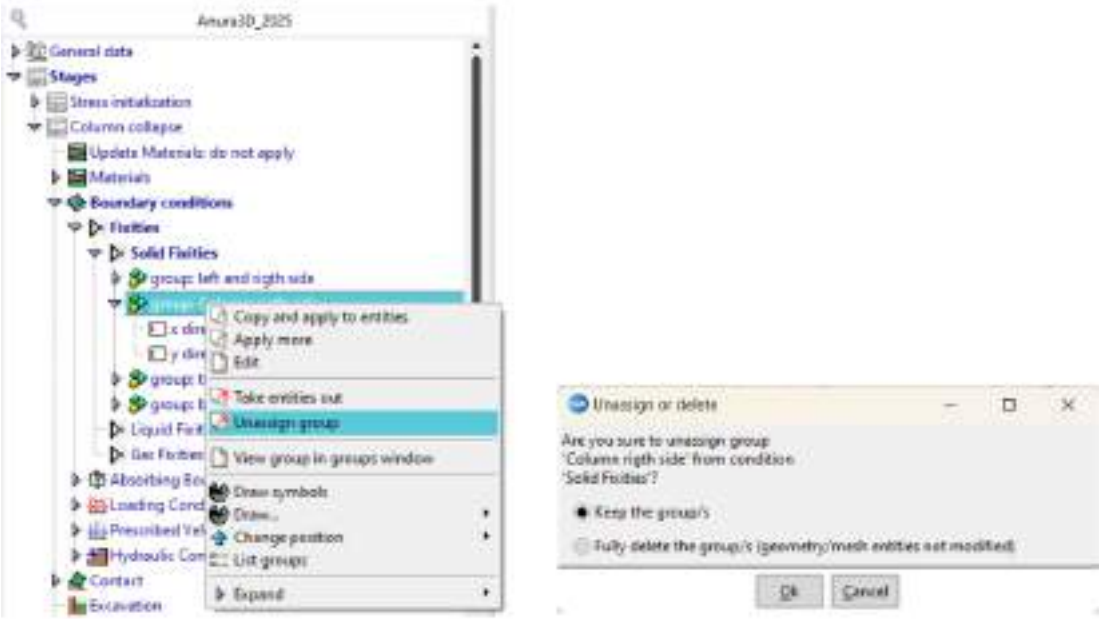


Figure 7.15: *Dialogue box to remove fixities.*

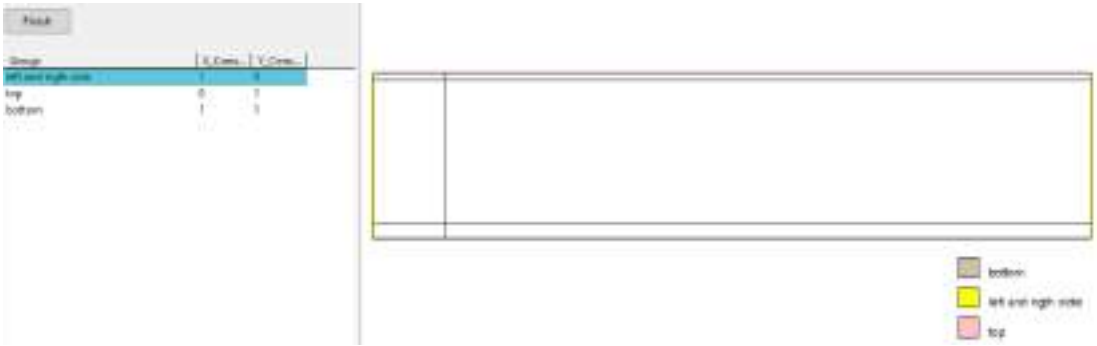


Figure 7.16: *Fixities stage 2.*

In the second phase 50 additional load steps will be calculated. The time per load step is set as 0.05 seconds to be able to follow (and visualise) the collapse of the column in small steps. Switch off the convergence to the quasi-static equilibrium as now the full dynamic behaviour of the column will be analyzed. Reduce the local damping coefficient to 5%. Therefore set:

number of calculation steps: 51
time per calculation step: 0.05
quasi-static convergence: do not apply convergence criteria
homogeneous local damping: apply homogeneous local damping
local damping coefficient: 0.05

All other calculation options in this example should be kept with the provided default values, see Figure 7.17.

Finally we reduce the "time between outputs" of the output MPs to have a better visualization of the process. In the "Material Point Output" menu set the time to 0.001s (Fig. 7.18).

7.3 Performing calculation in Anura3D

The calculation will start with the Calculate function in the Anura3D menu.

The calculation of the column failure phase should be finished in approximately 3 minutes depending on the performance of your PC.

7.4 Visualisation of results in ParaView

The results for the dry column example are presented for the end of the gravity loading phase and the end of the column failure phase.

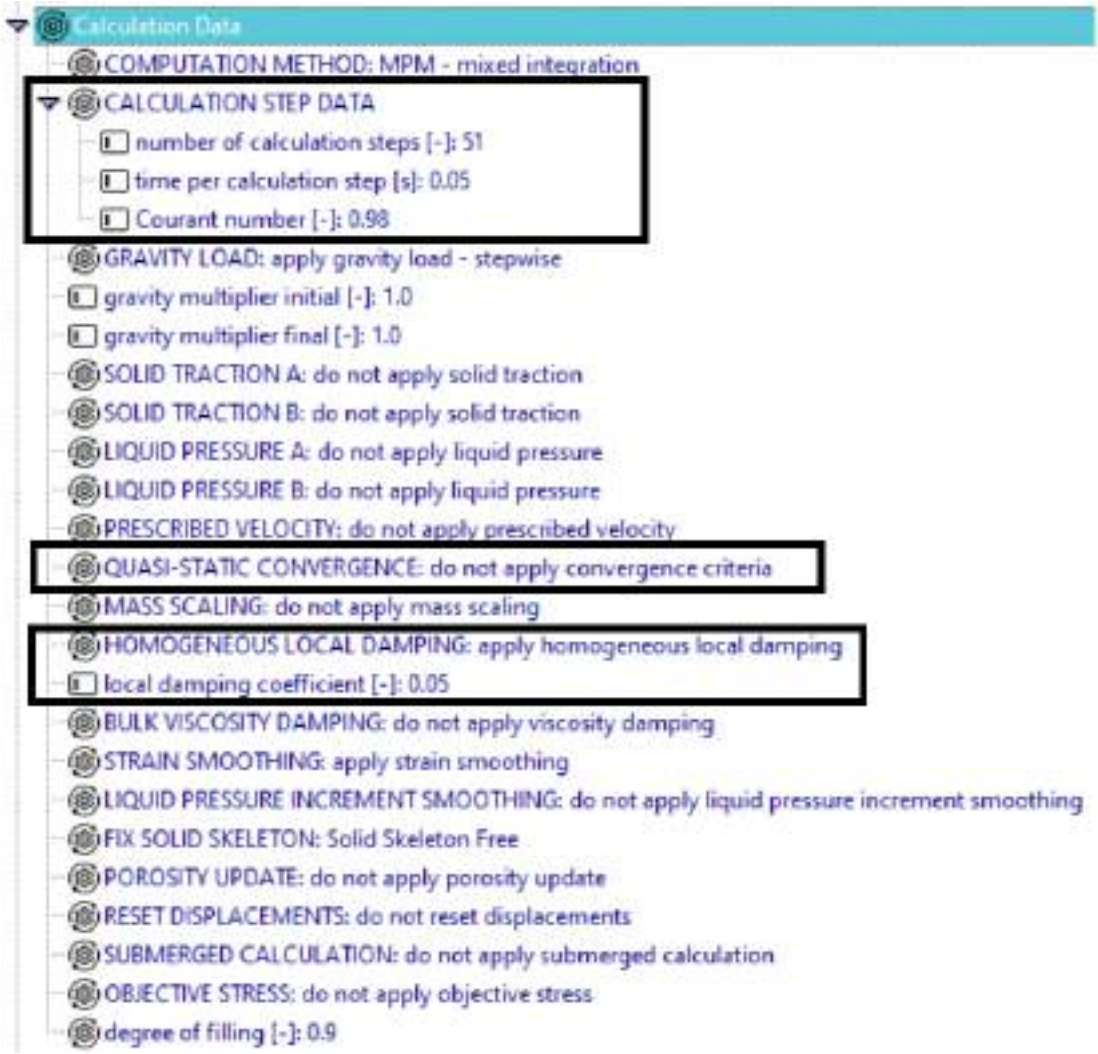
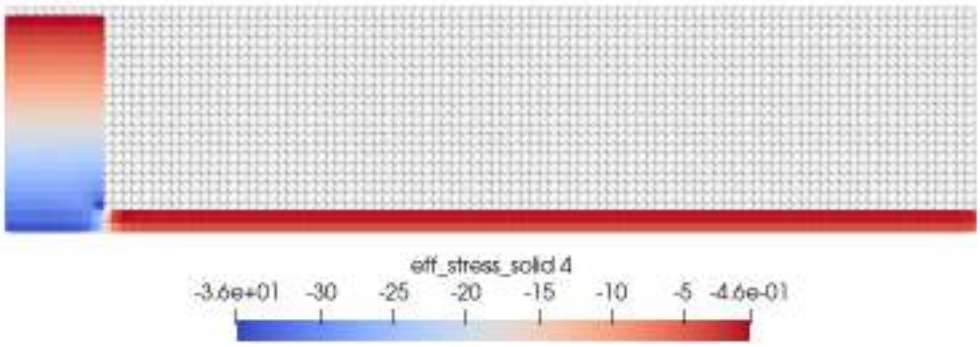


Figure 7.17: *Calculation data stage 2.*

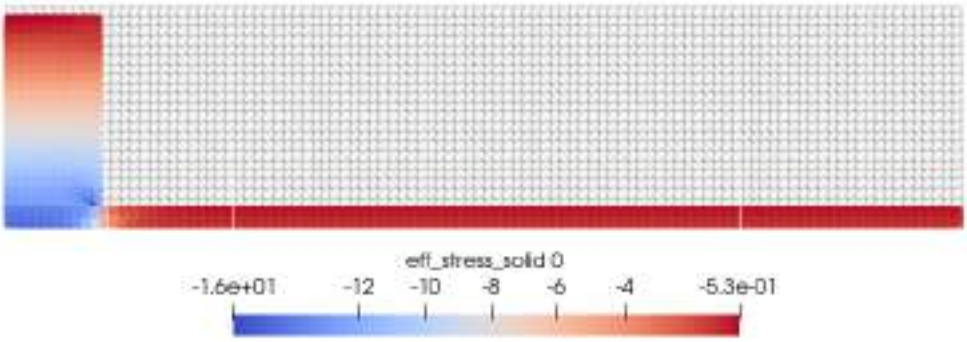
7.4.1 Gravity loading

The vertical and horizontal effective stresses after gravity loading are shown in Figure 7.19.

Figure 7.18: *Calculation data stage 2.*



(a)



(b)

Figure 7.19: *Effective stresses [kPa] after gravity loading for one-phase dry soil. a) Vertical stresses. b) Horizontal stresses.*

7.4.2 Column collapse

The final total displacements after collapse are shown in Figure 7.20a. The deviatoric strain after collapse is shown in Figure 7.20b. The vertical effective stress in [kPa] after collapse is shown in Figure 7.20c.

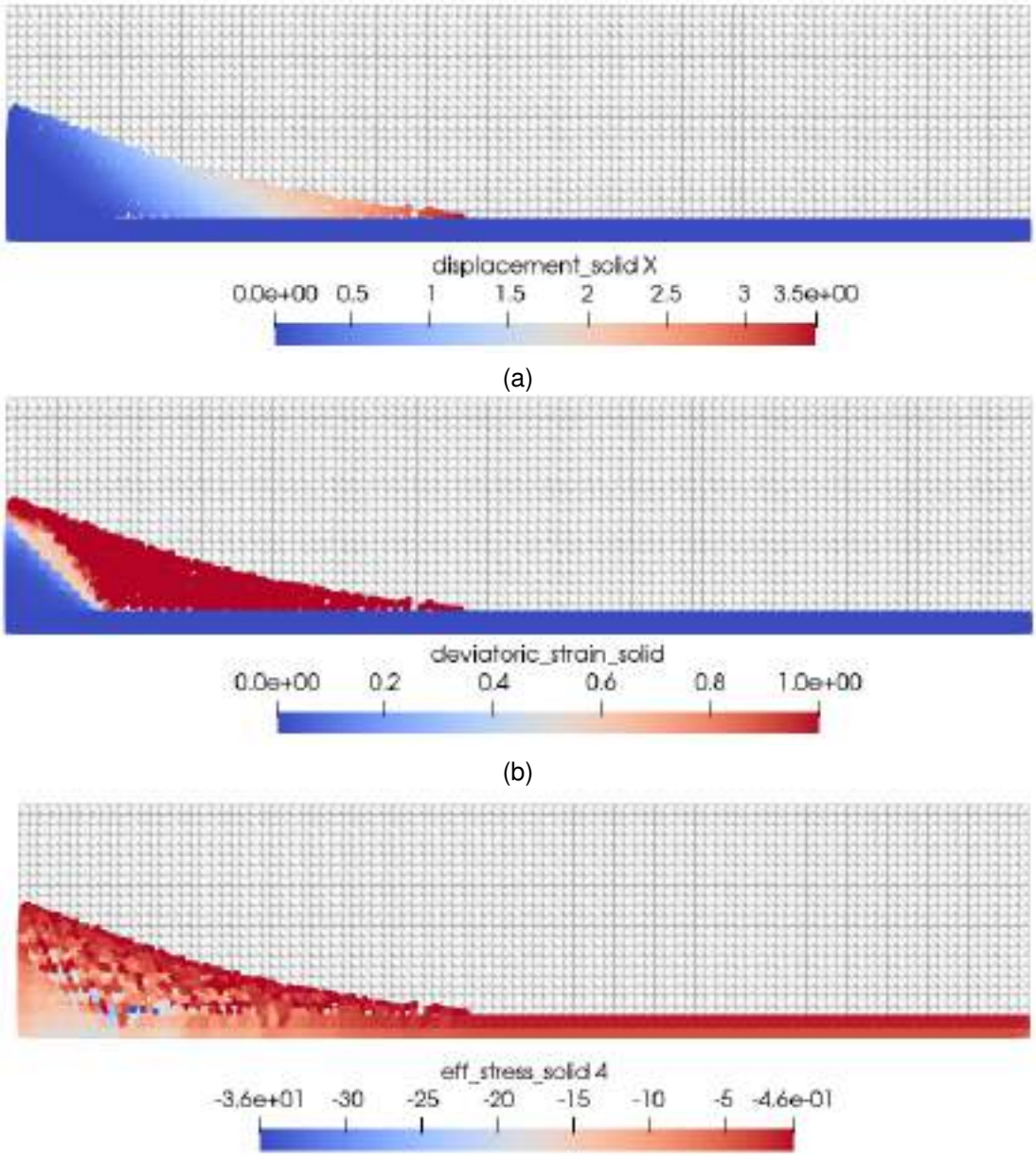


Figure 7.20: *Total displacements (a), Deviatoric strain (b) and Vertical stresses (c) after the collapse of one-phase dry soil.*

7.4.3 Visualization of results of text files

Relevant data stored at the selected MPs are written in the .PAR files. In this example we selected 3 MPs as shown in Sec. 7.2.4.6 and they can be visualized with any text editor, or imported in Excel or other calculation software.

For the column collapse problem it is interesting to plot the X and Y coordinates of the three points as shown in Figure 7.21.

Figure 7.21: *X and Y coordinates of 3 MPs during column collapse.*

8 Shallow foundation

Authors: Alba Yerro; Francesca Ceccato; Alexander Rohe; Bruno Zuada; James Fern; John Murphy; Luis E. Zambrano-Cruzatty; Abdelrahman Alsardi

This tutorial explains the use of Anura3D 2025 for the following functionalities:

- ◇ Load multipliers (2 systems named 'A' and 'B')
- ◇ Prescribed velocity
- ◇ Reaction forces
- ◇ Moving mesh.

A shallow foundation problem is simulated in 2D-plane strain conditions to show the above functionalities. The soil reaction to the structure movement is measured during the calculation with the option *Reaction Forces*. The exercise is solved with two approaches:

(1) a load is applied at the top of the foundation, and its magnitude increases progressively by means of the *load multipliers*. A variation of this will be presented where a surcharge will be included across the soil ground surface, which will be progressively decreased after the foundation load is applied.

(2) the foundation moves downward into the soil with a *prescribed velocity* bringing soil to failure

The *moving mesh* feature [4–6] exploits the fact that in MPM, all properties of the continuum are stored at the material points. Therefore, the mesh can be freely redefined at the end of each time step because the mesh does not store any permanent information. The moving mesh feature adjusts the mesh to the movement of a reference material (usually the structure, i.e. the foundation in this case) after each time step, ensuring that the surface of the structure coincides with element boundaries throughout the computation. With this approach, the mesh area adjacent to the foundation moves with the same average displacement as the structure. The elements in this moving area keep the same shape during the analysis, while the elements in a compressed area reduce their vertical length. The discretization of the compressed zone should be defined in such a way that the elements keep a reasonable aspect ratio until the end of the simulation.

The moving mesh feature is particularly useful in combination with the contact formulation and when reaction forces need to be computed. Indeed, if a body or structure is expected to move during the simulation, it is necessary to keep the element boundaries aligned with the object (i.e. the object should be very stiff or rigid). Similarly, if the reaction forces on a predefined surface, for example the soil–foundation interface, are computed, it is necessary that this interface follows the movement of the structure. The reaction forces on a surface are computed as the sum of the nodal reaction forces over the nodes belonging to the selected surface.

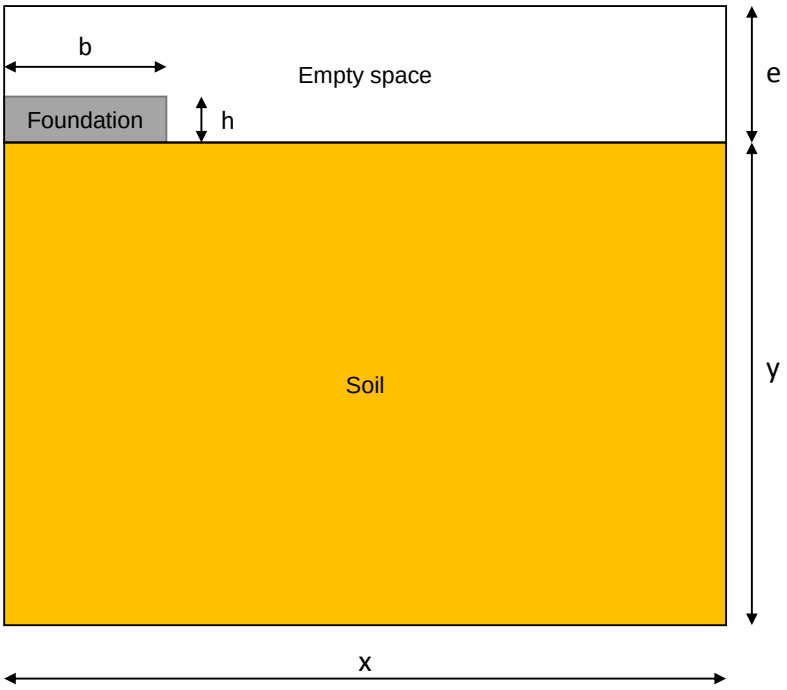


Figure 8.1: *Problem definition for failure of a shallow foundation.*

Table 8.1: *Dimensions of the shallow foundation problem.*

dimension	value
b	2m
h	1m
x	15m
y	10m
e	3m

8.1 Problem definition

A rectangular shallow foundation is considered on top of a horizontal layer of soil. Taking advantage of the symmetry of the problem, only half of the model is discretized in order to reduce the computational time. The geometry and the dimensions of the problem are given in Figure 8.1 and Table 8.1. The problem is simulated in 2D-plane strain conditions. Note that the boundaries have to be far enough from the foundation. The single point formulation will be used.

The soil is modelled with the Mohr-Coulomb model and the foundation is elastic and stiffer than the soil. The properties for both materials are given in Table 8.2. This information will be used for the material definition in Section 8.2.4.

Table 8.2: *Material parameters of the foundation and the soil.*

material	foundation	soil
material type	dry material	dry material
initial porosity [–]	0.0	0.4
density solid [kg/m^3]	2000 (*)	2650
K0-value [–]	0.5	0.5
material model solid	Linear Elasticity	Mohr-Coulomb
Young modulus [kPa]	50000	20000
Poisson ratio [–]	0.33	0.33
cohesion [kPa]	-	0.5
friction angle [°]	-	30
dilatancy angle [°]	-	0.0
tensile strength [kPa]	-	0.0

8.2 Creating input data

The geometry, discretization, material properties, and all other input parameters will be defined using the GiD pre-processor. This follows a similar procedure as described in Section 3.2. In this chapter, only the differences with respect to previous examples will be highlighted.

8.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1, ensuring that you select the problemtype. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "foundation").

8.2.2 Defining units and dimensions

The problem is simulated in 2D-plane strain conditions; therefore, under "Units and dimensions" in "Problem dimensions", select "2D:plane-strain". Note that the boundaries have to be far enough from the foundation. The single point formulation will be used; so, under "Units and dimensions" in "MPM discretization type", select "single_point". Also, unsure that in "Units", the geometry, force and time units are in m, kN, and s.

8.2.3 Defining geometry

To generate the geometry follow the instructions given in Section 3.2.2 considering the dimensions given in Table 8.1. As a reference the key coordinates indicated in Figure 8.2 and Table 8.3 can be used.

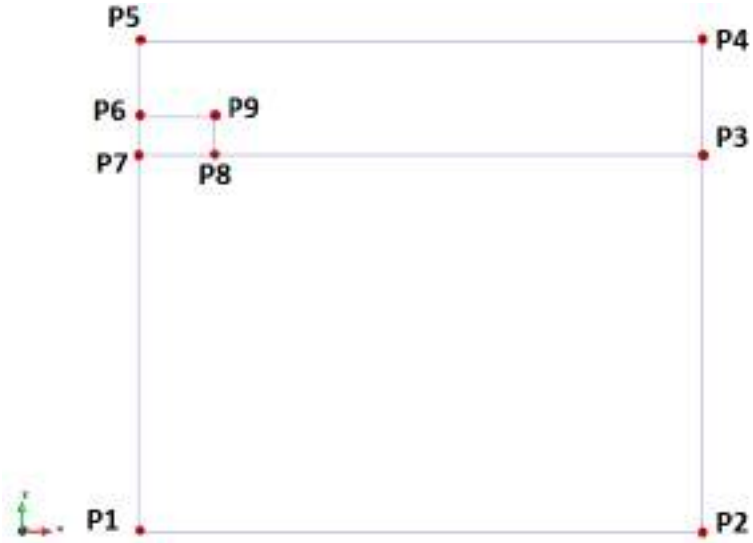


Figure 8.2: *Geometry of the front face of the shallow foundation model. Key coordinates are indicated as given in Table 8.3.*

Table 8.3: *Key coordinates of the front face of the shallow foundation problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	0	0
P2	15	0
P3	15	10
P4	15	13
P5	0	13
P6	0	11
P7	0	10
P8	2	10
P9	2	11

8.2.4 Defining material properties

The shallow foundation model consists of two different materials: the foundation and the soil. A 1-phase calculation will be performed because both materials are considered "dry". The foundation is elastic and the material model for the soil is Mohr-Coulomb. Material parameters are given in Table 8.2. Note that the Young modulus of the foundation used in this model is small compared with real foundation materials such as concrete, but it is significantly larger than the soil to ensure a rigid behaviour of the structure. A smaller Young modulus allows a larger critical time step and results are achieved faster, usually after a few minutes.

Follow the instructions given in Section 3.2.5 to define each material. Checking the assigned materials, it should look like shown in Figure 8.3.

Figure 8.3: *Materials of the shallow foundation problem.*

Figure 8.5: *K0 initialization for foundation problem in GiD.*

8.2.5 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. In this example choose 3 material points per element and assign it to the soil and the foundation materials. Check if the material points are assigned correctly by right clicking on "Material Point Specification" and selecting "Draw" and "Draw values" as shown in Figure 8.4.

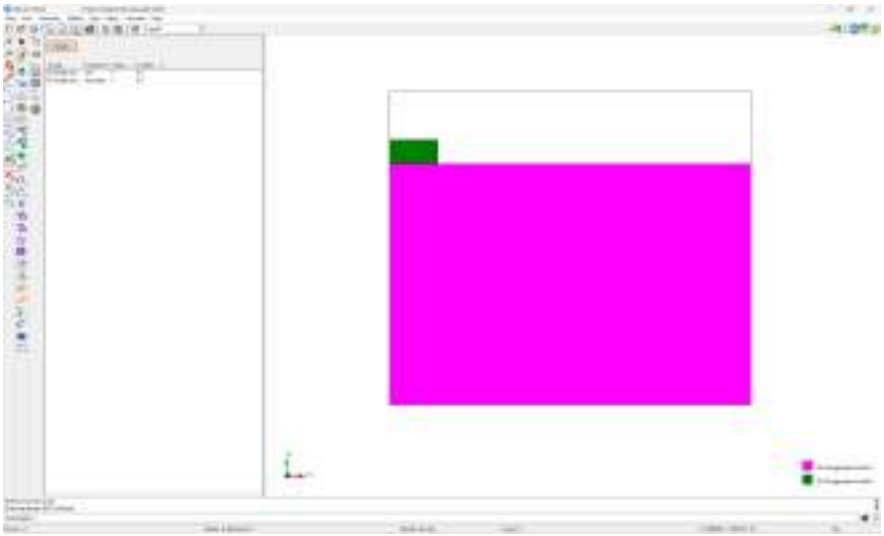


Figure 8.4: *Specification of the number of material points per element in the shallow foundation problem.*

8.2.6 Defining initial conditions

The stresses in the soil will be initialised at the very beginning of the calculations by means of the K_0-procedure. Activate the K_0-procedure by selecting "Yes" in the "Apply K0-procedure" option. This option is found in the Anura3D tree menu under the section "Initial conditions" → "Stress initialization" → "Horizontal layered soil". Keep the initial vertical load to "0kPa" and set "Soil surface elevation" to "10.0m" (Figure 8.5).

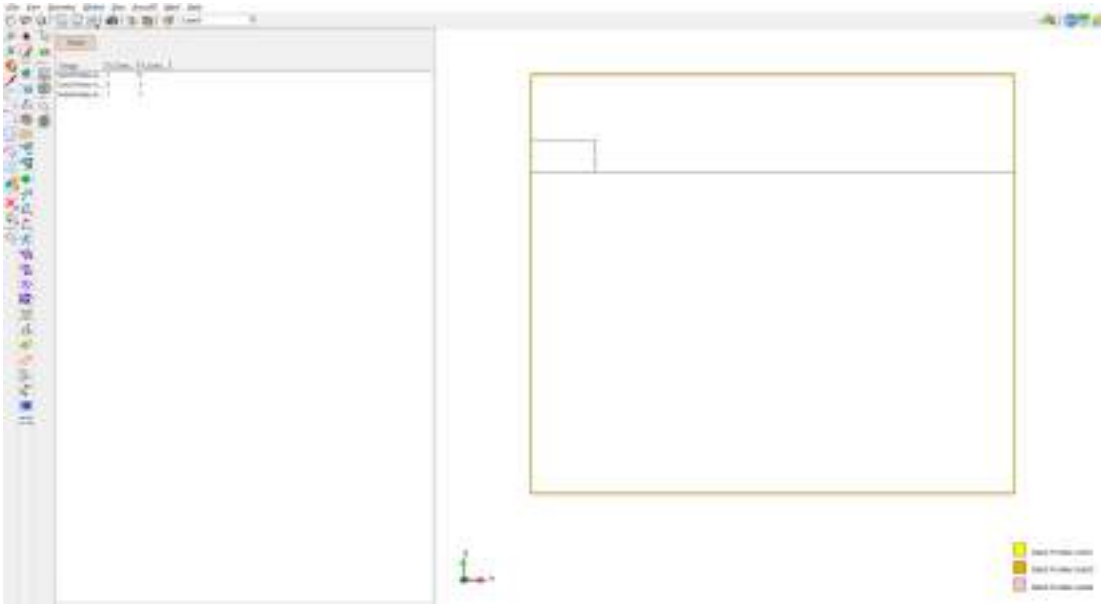


Figure 8.6: *Solid fixities of the shallow foundation problem in GiD.*

8.2.7 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying fixities
Since in this example materials are in dry conditions, only the solid phase needs to be considered. The solid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction on all three lines;
(2) Right side: horizontal fixity in x-direction on both lines;
(3) Top side: vertical fixity in y-direction on the top line;
(4) Bottom side: full fixities in x- and y-direction on the bottom line;
(5) Check if the x- and y- fixities are assigned correctly by right clicking on "Boundary Conditions" and selecting "Draw" and "Draw values" as shown in Figure 8.6;

Applying prescribed velocity
A prescribed velocity can be applied to the nodes or to the material points. In the first example, the prescribed velocity is assigned at material points (in the code, the material point velocity is mapped at the nodes of the element, thus the body moves rigidly). In the second example, the prescribed velocity is applied at the nodes. Because the foundation is very stiff, it is recommended to use this feature in combination with the moving mesh. This will keep the boundary of the moving body always aligned with the element boundary. Moreover, the prescribed velocity must be aligned with the *moving mesh direction* and the *moving mesh reference material* must be the material on which velocity is prescribed (the foundation in this case) (see 8.2.9).

Table 8.4 shows the available options for prescribed velocity in 2D and 3D models. In Anura3D 2025 only one of these options can be applied in the same project.

In this problem we prescribe a velocity of –0.1m/s in y-direction at the material points of the foundation, while the horizontal direction is free. This can be done through the menu

Table 8.4: *Prescribed velocity options.*

	Material Points	Nodes
2D	Surfaces	Surfaces
		Lines
		Points
3D	Volumes	Volumes
		Surfaces
		Lines
		Points

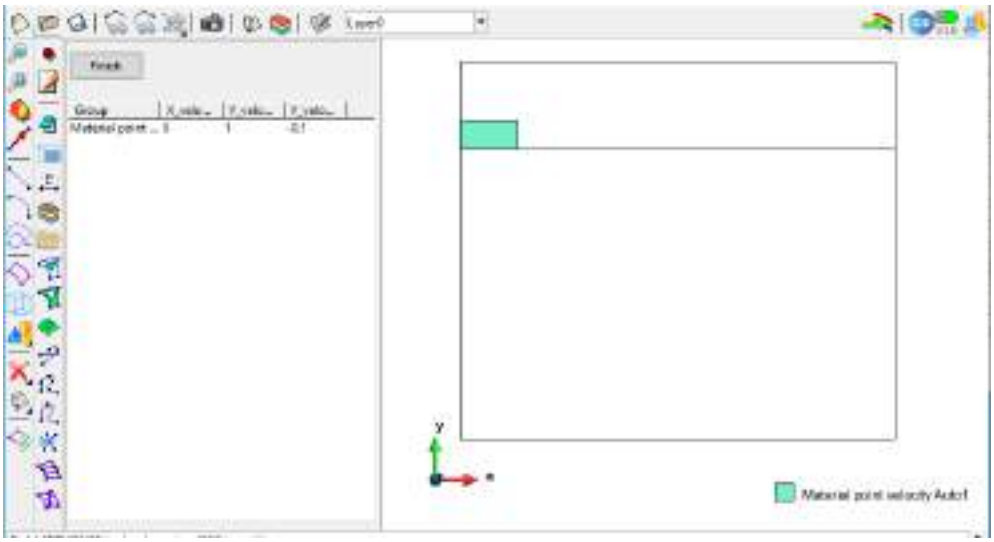


Figure 8.7: *Applied prescribed velocity on node.*

<Boundary Conditions><Prescribed Velocity><Material point Velocity>:

(1) In Apply Material point velocity, select *y direction*.
(2) (you might need to scrall down).
(3) Enter –0.1m/s as the prescribed velocity.
(4) Click on the <Select> button and choose the surface identifying the foundation, then press <OK>.

To check the assigned property right click on <Material point velocity>, select <Draw><Draw values> and you will get the result in Figure 8.7. Then, click <Finish>. Note that x-direction is free, i.e. at the selected nodes the velocity will result from the solution of the governing equations, while y-direction is prescribed, i.e. the velocity is –0.1m/s.

Applying loading conditions

A vertical load 'A' equal to –100 kPa will be applied in y-direction on the top of the foundation as shown in Figure 8.8 (for more information on how to apply a load, please refer to "Applying loading conditions in 2D and 3D" in section 3.2.7). Additionally, a vertical load 'B' equal to –100kPa will be applied in y-direction to represent a surcharge extending across the soil next to the footing. Note load A and B are specified in the GiD pre-processor, but the user can specify to turn them on or off independently in the CPS file as will be presented subsequently.

In this exercise the moving mesh approach will be used. This means that the surface on which

the load is applied will move together with the foundation, and therefore, load A can be applied on the nodes. On the other hand, if a significant relative displacement between the loaded surface and the material points is expected, the load should be applied on the material points instead, so the load is carried by them throughout the simulation. This is the case of load B, that will be applied on the material points of the soil material. Follow instructions in section 3.2.7 to apply the loading conditions.

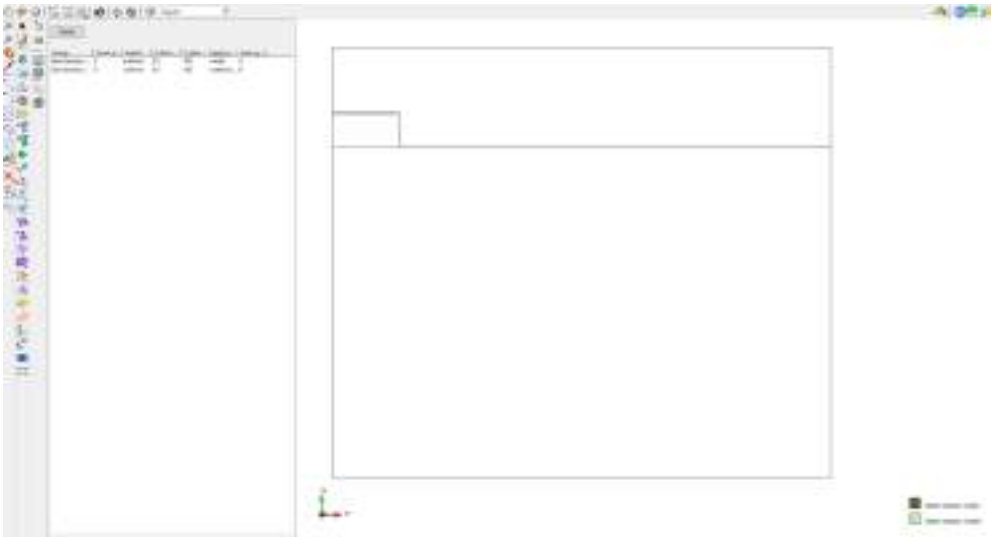


Figure 8.8: *Applied load on top of the foundation at the nodes (load system A) and on top of the soil at the material points (load system B).*

8.2.8 Defining a surface for computation of reaction forces

This feature allows the user to generate a text output file with extension .RSurf_XXX, where XXX is a number that depends on how many surfaces are selected. This file collects, for each time step, the sum of reaction forces over the nodes of the selected surface for different materials (see Section 8.3.5).

The following steps have to be taken to select a surface for computation of reaction forces:

(1) Double click on <Reaction Forces> from the menu as shown in Figure 8.9a
(2) Enter a name for the "line identifier", for example, "reaction".
(3) Click on the <Select> button and select one (or more lines) for which the reaction forces should be computed, in this case select the soil–foundation interface, as shown in Figure 8.9b. Note that only one output file will be created for surfaces with the same name.
(4) Check if the surfaces are assigned correctly. Right click on <Reaction Forces> and select <Draw><Draw values>.
(5) Click on <Finish> to exit.

The reaction forces can be computed for multiple surfaces by repeating the steps above. The maximum number of surfaces that can be created is 50.

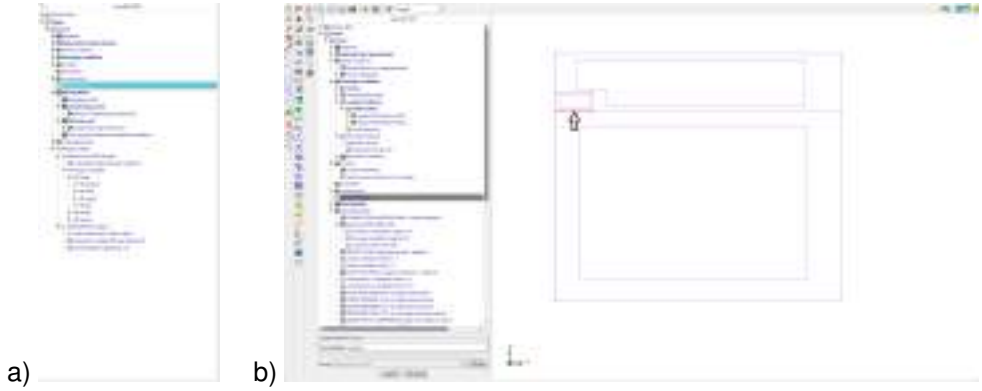


Figure 8.9: *Definition of the surface for computing reaction forces.*

8.2.9 Defining the moving mesh

The moving mesh feature can be accessed through the menu <Moving Mesh> as shown in Figure 8.10. Three domain areas can be defined, i.e. extending mesh, compressing mesh, and moving mesh. The elements of the moving mesh domain translate as a whole in one specified direction following a reference material, while the elements of the extending/compressing mesh domains will extend/compress during the computation as a result of the movement of the moving mesh domain. The three domain areas must be connected with each other and all elements of the discretised domain must belong to either one of them. At least two domain areas must be defined, i.e. moving mesh + extending mesh, moving mesh + compressing mesh, or moving mesh + extending mesh + compressing mesh). The shape of the mesh domains must be rectangular. The different mesh domains must be connected.

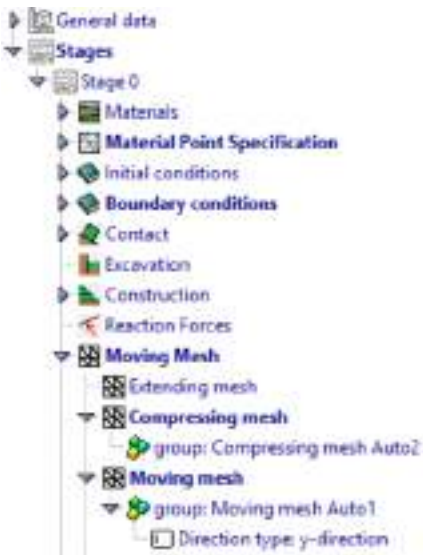


Figure 8.10: *Definition of the mooving mesh*

The following steps have to be taken to define the moving mesh feature. Note that in this problem, only compressive mesh and moving mesh domains will be defined.

(1) Double click <Compressing Mesh> from the menu.

(2) Click <Select> and select the 4 corner nodes that define the zone that will be compressed by the movement of the moving mesh as shown in Figure 8.11. Select <OK> in the Com-

pressing Mesh window to finish assigning compressing mesh. Note that the surface defined by the corner nodes must be a rectangle, thus exactly 4 nodes must be selected of which 2 must be on an external boundary. Alternative shapes of the compressing/extending/moving mesh are not allowed.

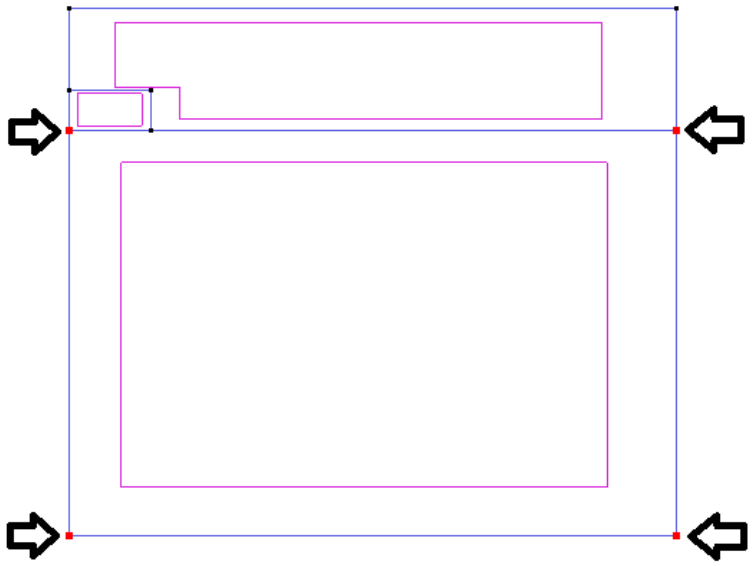


Figure 8.11: *Definition of compressing mesh corner nodes.*

(3) In this example no "Extending Mesh" will be used, thus no corner nodes must be selected for this option.

(4) Double click <Moving Mesh> from the menu.

(5) Specify the direction of movement. In this example the foundation should move in vertical direction, thus choose "y-direction" next to "Direction type".

(6) Click <Select> and select the 4 corner nodes that define the moving mesh region as shown in Figure 8.12. Two of the moving mesh nodes must coincide with the 2 nodes of the compressing mesh, i.e. the nodes of the top soil surface. The remaining 2 nodes must be on the free boundary, i.e. top surface. Select <OK> in the Moving Mesh window to finish assigning moving mesh. Note that when using both compressing and extending mesh, it is necessary that exactly 2 nodes of the moving mesh coincide with 2 nodes of the compressing mesh and the remaining 2 nodes coincide with 2 nodes of the extending mesh.

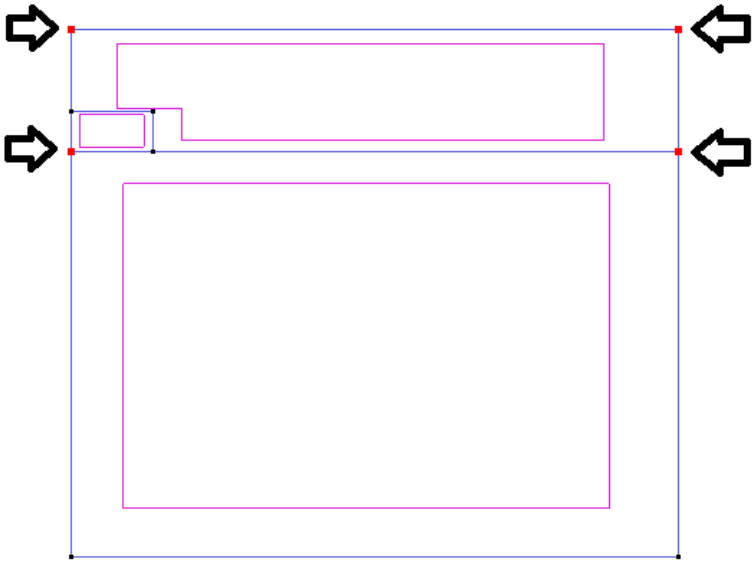


Figure 8.12: *Definition of moving mesh corner nodes and direction of movement.*

(7) Define the reference material that will determine the displacement of the moving mesh. Right click on the <Moving mesh reference material> and select <Edit> to activate the draw down menu. Now, all materials defined in the model are visible. Select "foundation".

(8) Click <Assign> and select the structure surface, i.e. the foundation, as shown in Figure 8.13.

Figure 8.13: *Definition of moving mesh reference material.*

(9) Check if the compressing and moving mesh domains are assigned correctly. Right click on <Compressing mesh> or <Moving mesh> and select <Draw><Draw values>.

(10) Click on <Finish> to exit.

8.2.10 Creating mesh

In GiD it is possible to generate meshes in a very flexible way. An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example set the mean element size to "0.5". The mesh should look as shown Figure 3.2.8.

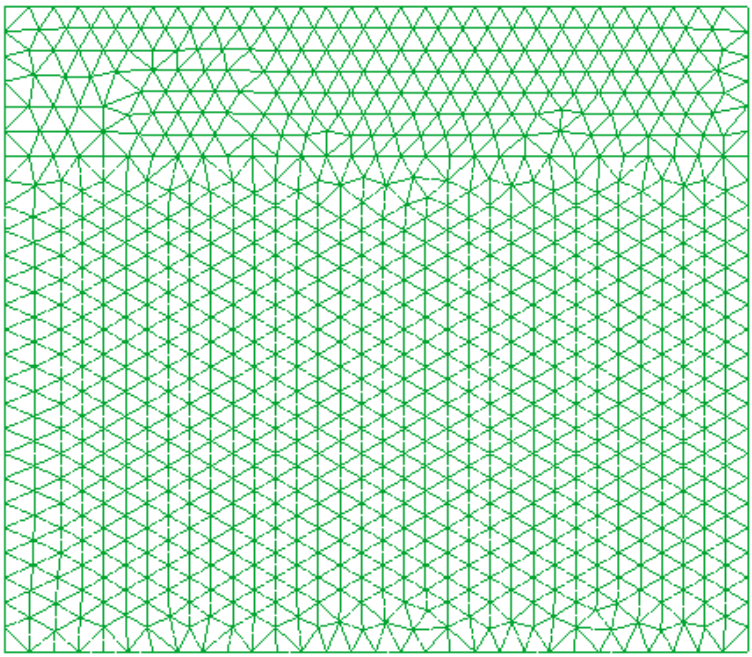


Figure 8.14: *Computational mesh of the shallow foundation problem.*

8.3 Approach 1: External load A and use of load multipliers

The input generated in the previous section is identical for both approaches 1 and 2; the differences between these two methods are highlighted in the following. In the first approach, the foundation is brought to failure by increasing the load applied on the footing to –1000kPa in 6 load steps by means of the load A multipliers. Load A is increased at the beginning of the load step and kept constant until quasi-static convergence is reached.

8.3.1 Initialisation of calculation data

To initalise the calculation data, select <CalculationData> from the menu and consider the following options.

Computation method
The option "MPM - mixed integration" should be selected.

Calculation step data
This example will be defined as a quasi-static problem and 6 load steps will be used. The end of each step will be determined by the quasi-static convergence criteria. Set "number of calculation steps [-]" to "6". A Courant number of 0.98 is advised to ensure the stability of the solution. Set "Courant number [-]" to 0.98. Because this is a quasi-static problem, "time per calculation step [s]" will not affect the calculation.

Gravity data
Use standard "gravity acceleration" value: $\mathbf{g} = 9.81(0, -1.0, 0)$, and gravity load vector as $(x, y, z) = (0, -1, 0)$

Gravity load
Gravity is applied from the beginning; therefore, choose *Apply gravity load - stepwise* and set the multipliers for gravity loading as 1.0 (initial and final value). The gravity multipliers will be maintained constant throughout the calculation.

Solid traction
In this example the external load A will be increased from –100kPa to –1000kPa in steps. Therefore, for solid traction A, choose *Apply solid traction - stepwise* and the multipliers for external load must be 1.0 and 10.0. This means that the incremental load A is applied at the first time step and kept constant for the entire load step computation. In this case, at load step 1 the external load A multiplier is

$$M_{step} = M_{initial} + (M_{final} - M_{initial})\frac{N_{current} - N_{previous}}{N_{totstep} - N_{previous}} = 1.0 + (10 - 1)\frac{1 - 0}{6 - 0} = 2.5 \quad (8.1)$$

where M_{step} is the load multiplier of the current load step, $M_{initial}$ is the initial load multiplier, M_{final} is the final load multiplier, $N_{current}$ is the number of the current load step, $N_{previous}$ is the number of the previous load step, and $N_{totstep}$ is the total number of load steps. In this case, the applied vertical load is $2.5 * (-100\text{kPa}) = -250\text{kPa}$.

Load B representing the soil surcharge will be turned off in this approach. Therefore, for solid traction B, keep the default *do not apply solid traction*.

Prescribed velocity
In the first example the foundation is brought to failure increasing the load. Keep the default *do not apply prescribed velocity* option because this example does not use the prescribed velocity feature.

Quasi-static convergence
This calculation will be performed using the quasi-static convergence approach; select *apply convergence criteria*. In this type of analysis, the determination of the end of each load step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition. The convergence criteria are the tolerated error of the kinetic energy, the tolerated error of the out-of-balance force, or the maximum number of time steps. In general, to obtain an accurate solution it is suggested to set the kinetic error and the force error to 0.01; however, in this example, in order to reduce the computational time, use a kinetic error of 0.01 and a force error of 0.05. This is a dry calculation, hence liquid errors can be left with default values. If the calculation does not converge, it will stop when the macimum numer of time steps is reached. You can leave the default value of 10000.

Homogeneous local damping
In order to damp oscillations to reach quasi-static convergence a local damping coefficient for all active elements of 0.7 is applied. Select *apply homogeneous local damping* and enter the coefficient 0.7. This value of local damping will be kept constant during the whole calculation since the quasi-static solution should be reached.

Strain smoothing
Select "apply strain smoothing" to minimize the kinematic locking.

Output data

One material point located at the bottom of the foundation will be selected for which additional output files will be created, (see Section 3.4.7 how to determine the ID number of a material point).

Default values

All other calculation options are not used in this example and the provided default values can be kept.

Figure 8.15: *Dialogue box to define calculation parameters in GiD for the approach 1 of shallow foundation problem. Note only load system A is used here.*

8.3.2 Generating Input Files

After the geometry, material properties, boundary conditions, and calculation parameters are defined and the mesh is generated the input files for the Anura3D analysis, calculation can be performed. Go to Anura3D menu in taskbar and click <Calculate>. Individual *GOM* and *CPS* file can also be edited later on.

8.3.3 Performing calculation in Anura3D

The calculation can be performed as explained in Section 3.3.

Note that in `foundation.CPS_002` the load multipliers are stored as,

```
$$SOLID_TRACTION
 step 2.50 10.0
```

which means that the initial load multiplier is $M_{initial} = 2.5$ and the final load multiplier is $M_{final} = 10.0$, thus the load multiplier of the current load step (number 2) is:

$$M_{step} = M_{initial}+(M_{final}-M_{initial})\frac{N_{current}-N_{previous}}{N_{totstep}-N_{previous}} = 2.5+(10-2.5)\frac{2-1}{6-1} = 2.5+1.5 = 4.0 \quad (8.2)$$

and therefore the applied vertical load is $4.0 * (-100\text{kPa}) = -400\text{kPa}$.

Similarly, in `foundation.CPS_003` the load multipliers are stored as,

```
$$SOLID_TRACTION
 step 4.0 10.0
```

8.3.4 Visualisation of results in ParaView

The results for the shallow foundation problem are presented in this section. The recommended way to visualise Anura3D results is by using the ParaView software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results from the shallow foundation problem.

In order to appreciate the effect of the moving mesh feature, the material index will be plotted and the Axis Grid will be switched on as shown in Figure 8.16. Note that in this example `material_id=1` is the foundation and `material_id=2` is the soil. The material index can be retrieved from the *GOM* file.

```
$$MATERIAL_INDEX
1
$$MATERIAL_NAME
foundation
```

```
$$MATERIAL_INDEX
2
$$MATERIAL_NAME
soil
```

Figure 8.17 shows the displacement of the moving mesh area, which moves downwards following the displacement of the foundation, and the deformation of the compressing mesh area. Note that the material points of the foundation do not cross element boundaries and the soil–foundation interface remains aligned with the element boundaries. In Figure 8.18 the evolution of displacements is presented.

8.3.5 Visualisation of text output files

Following the instructions from Section 3.6, the evolution in time of vertical displacement of a material point located at the bottom of the foundation (3957) can be plotted. Figure 8.19 shows that the vertical displacement of MP #3957 increases rapidly at the beginning of each load step, due to the sudden increase of the applied load, and then stabilizes approaching the quasi-static equilibrium. The load multiplier for solid traction increases in step during the calculation (blue dashed line in Figure 8.19) while gravity is constant (red dot-dashed line in Figure 8.19).

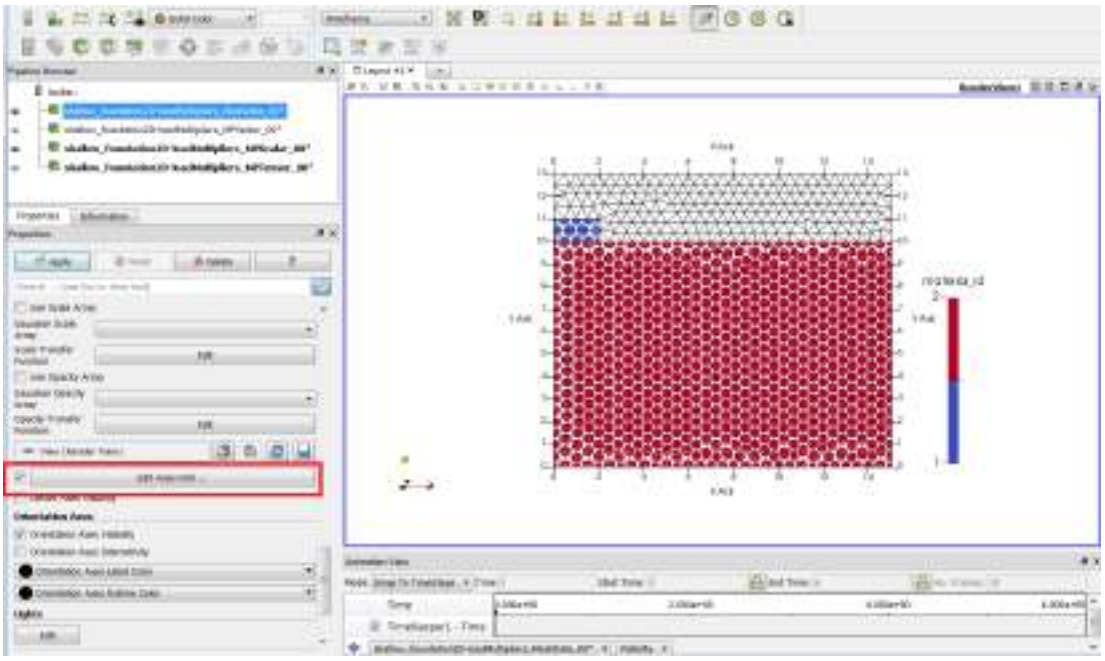


Figure 8.16: *Initial mesh and plotting the material index.*

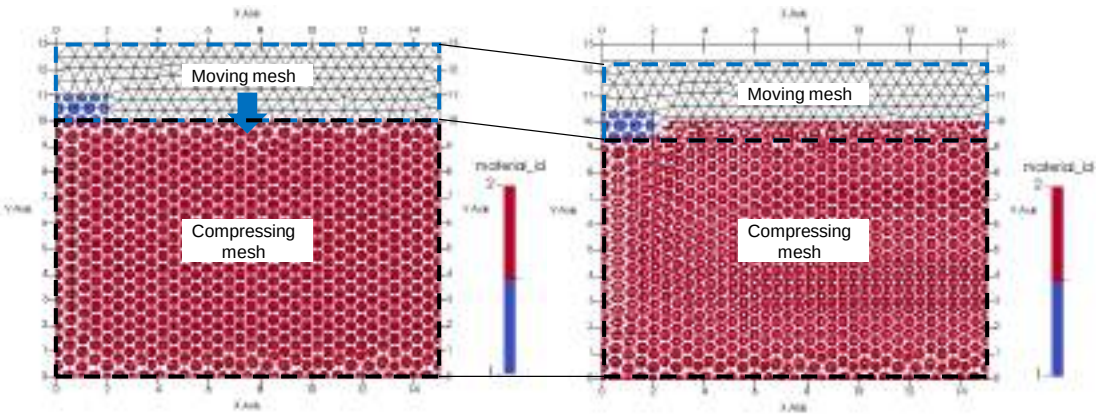


Figure 8.17: *Displacement of the mesh at the beginning of the computation and after load step 6.*

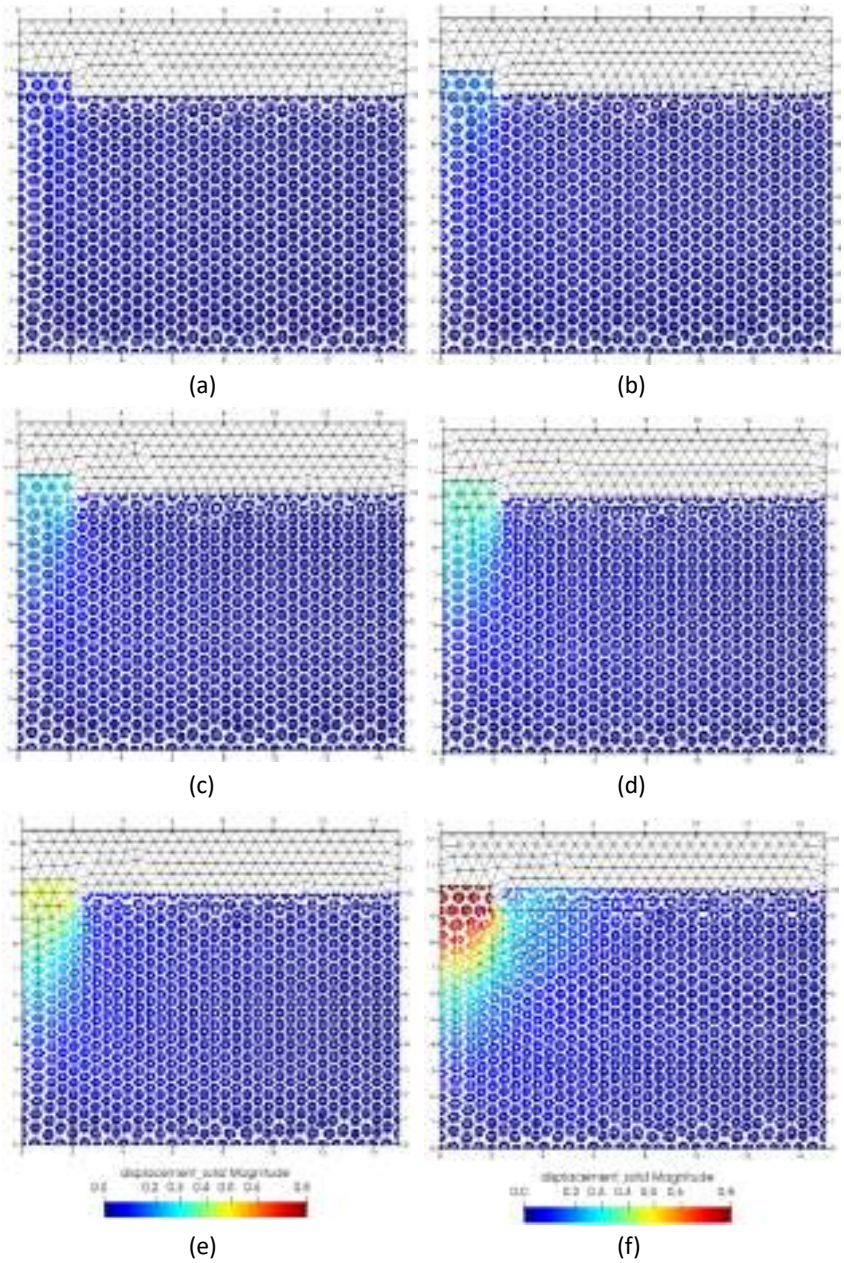


Figure 8.18: *Displacement after: a) load step 1, b) load step 2, c) load step 3, d) load step 4, e) load step 5 and f) load step 6.*

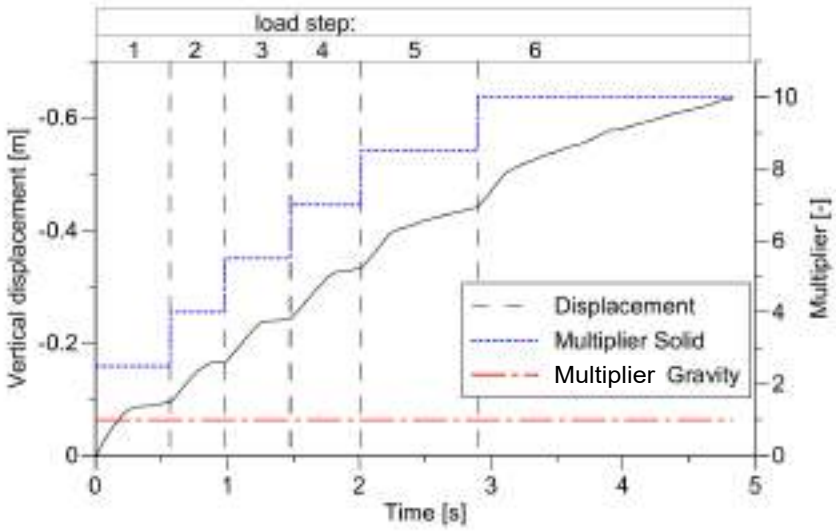


Figure 8.19: *Evolution of vertical displacement for a material point located at the foundation bottom, and value of load multipliers.*

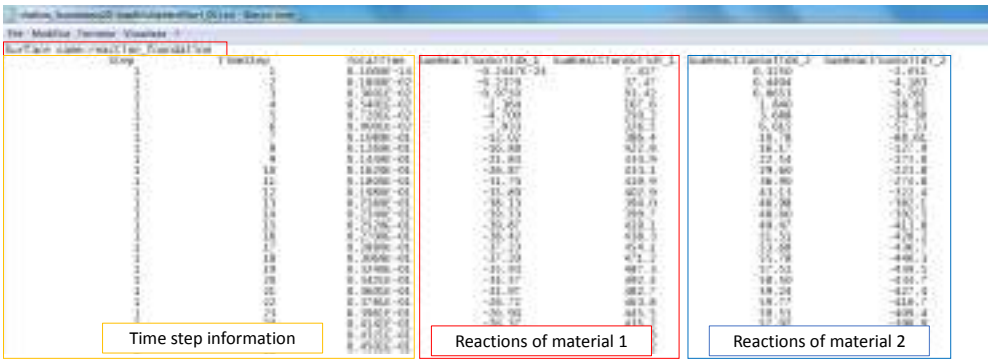


Figure 8.20: *Output file for reaction forces .RSurf_001 .*

The results file `foundation.RSurf_001` can be opened using Excel or with NotePad, see Figure 8.20. The first line of the file indicates the surface name, i.e. the name that has been assigned in GiD. The first 3 columns of the file contain time information. The following 4 columns contain the sum of the nodal reaction forces on the surface in x- and y-direction for material 1 and 2, respectively. The maximum number of materials that can be handled in this output file is 3. For 2-phase analyses, the reaction forces due to the pore pressure are given.

The results of this file can be plotted with any chart making program. Figure 8.21 shows the vertical reaction forces for material 1 (foundation) and 2 (soil); the forces have similar values but opposite sign. As expected, for each load step the reaction force is:

$$R = M_{step} qb = M_{step}(100kPa)(2m) \tag{8.3}$$

and it maximum value is 2000kN/m.

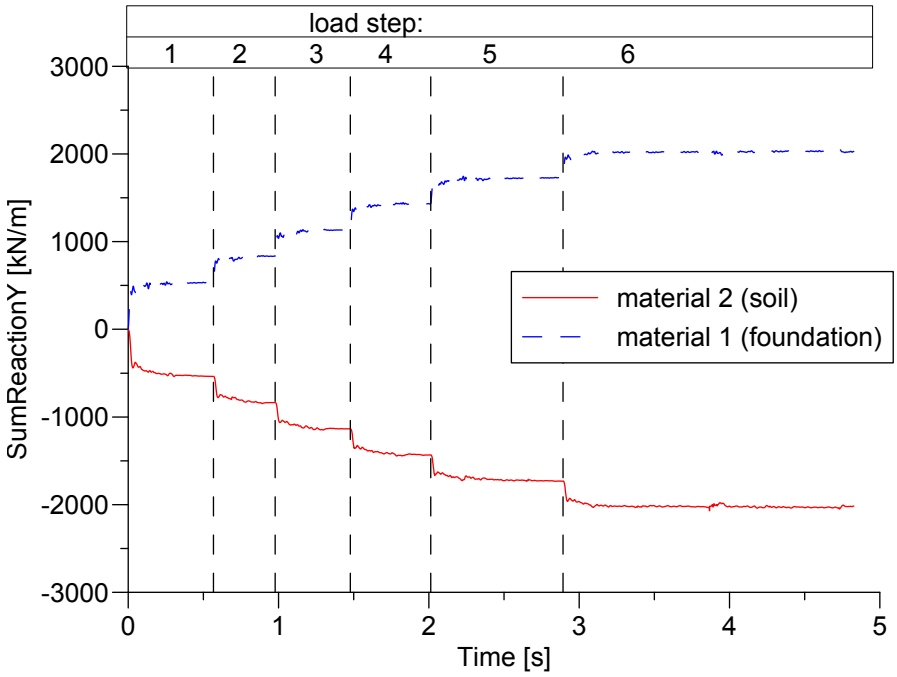


Figure 8.21: *Vertical reaction forces.*

8.3.6 Approach 1 - Variation: add load system B to simulate surcharge on soil

Anura3D 2025 is equipped with two independent load systems that have been defined in section 8.2.7. To showcase this feature, Approach 1 (section 8.3.1) is varied and the surcharge load across the soil near the footing is activated (load B). The load system conditions are shown in Figure 8.8. For load A (associated with the footing), the multipliers are identical to what was previously presented in the original Approach 1 (section 8.3.1). The multipliers for load B are also presented in Figure 8.22 (initially constant followed by a reduction of the load). In this section, the effect of having two load systems A and B corresponding to the footing and the surcharge, respectively, is investigated.

The file initialization in GiD is identical to Approach 1 except for the <Calculation Data> (section 8.3.1). The load multiplier B is turned on and its value is kept at 100kPa for the first six load steps as shown in Figure 8.23. In order to do so, one can modify the <Calculation Data> in the original GiD project as seen in Figure 8.23. Another option is to directly open the `foundation.CPS_001` text file generated in Approach 1 and modify the corresponding flag input as follows:

```
$$SOLID_TRACTION_B
step 1.0 1.0
```

The calculation is performed as discussed in Section 3.3. Subsequently, after the calculation is done, four more load steps are adopted to decrease load B to zero as shown in Figure 8.23. To do so, the number of load steps in `foundation.CPS_007` is directly edited and increased to a total of 10, as follows:

```
$$NUMBER_OF_LOADSTEPS
10
```

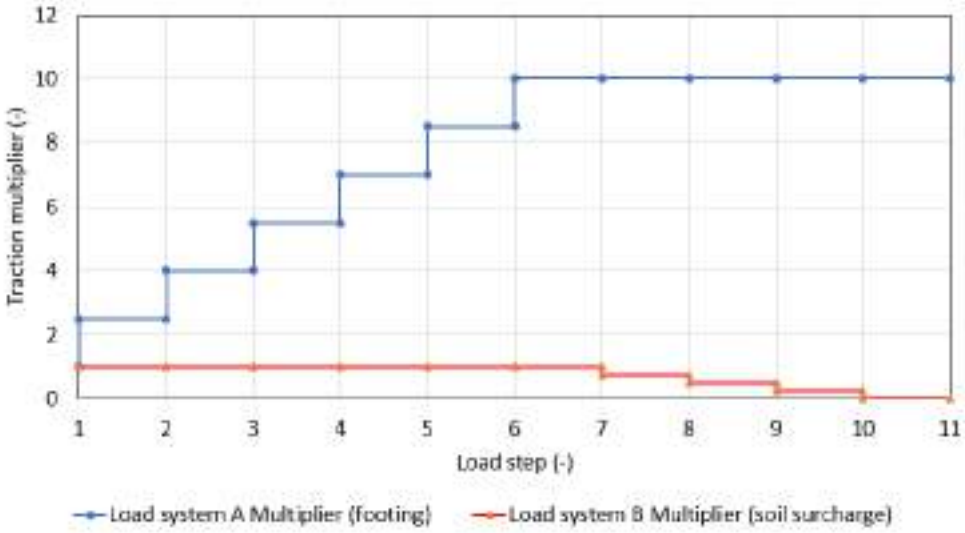


Figure 8.22: *Evolution of load multipliers*

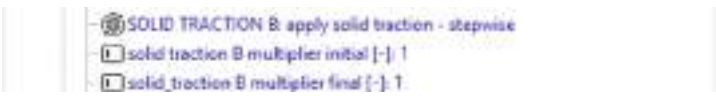


Figure 8.23: *Dialogue box showing the activation of load system B associated with the soil surcharge.*

Additionally, the load B multiplier is set to zero at the end of the simulation as follows:

```
$$SOLID_TRACTION_B
step 1.0 0.0
```

The calculation is then performed for the four additional load steps. Vertical displacement results for the material point directly below the footing is shown in Figure 8.24. Additionally, the external force vector applied on the material points can be visualized in Paraview as shown in Figure 8.25.

8.4 Approach 2: Prescribed velocity

With this approach, the foundation penetrates into the soil with a prescribed vertical velocity of 0.1m/s and the reaction offered by the soil is measured at the soil-foundation interface. We will not activate the solid traction on the foundation and we will not use the quasi-static convergence criteria.

8.4.1 Initialisation of calculation data

The file initialization in GiD is identical to Approach 1 except for the <Calculation Data> (section 8.3.1). All calculation parameters are identical to the previous case, except for those listed below:

Calculation step data

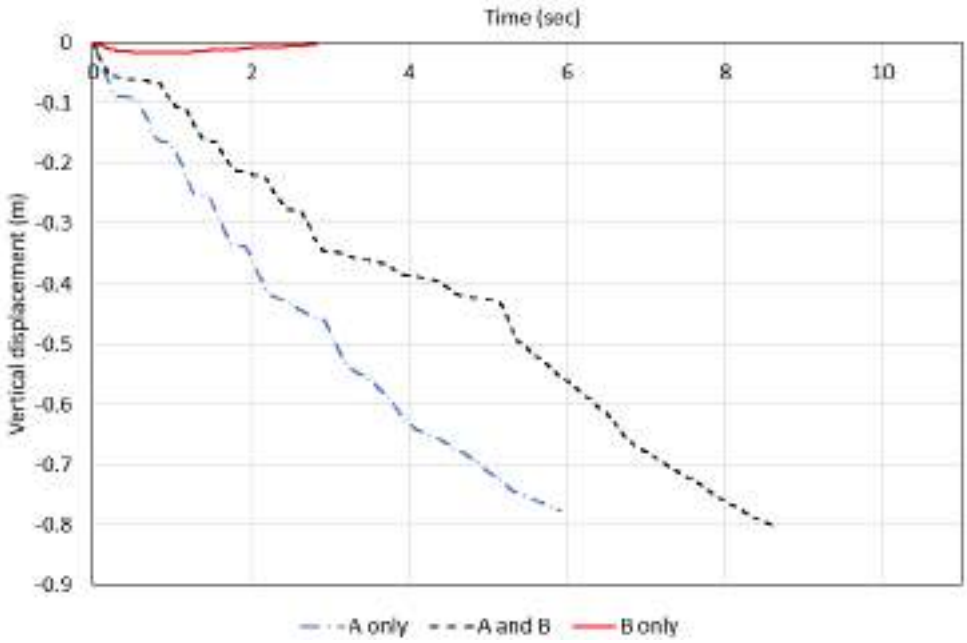


Figure 8.24: *Evolution of vertical displacement with different load system combinations. Note the simulation time is not consistent for all three simulations because it depends on the quasi-static convergence criteria in each load step.*

In this example 8 load steps of 0.5s will be used. Set "number of calculation steps [-]" to "8" and "Time per calculation step [s]"' to "'0.5"'.

Solid traction
In this example we will not apply solid traction, thus the option *do not apply solid traction* must be selected.

Prescribed velocity The velocity is constant throughout the calculation, thus select *apply prescribed velocity - stepwise* and set the multipliers to 1.0 and 1.0.

Quasi-static convergence
Set *do not apply convergence criteria*.

Homogeneous local damping
Set *no homogeneous local damping*. This is a slow process, thus a very small amount of damping (e.g. 5%) can be used to damp dynamic oscillations.

8.4.2 Generating input files

The generation of input file is identical to the previous case, see Section 8.3.2.

8.4.3 Performing calculation in Anura3D

The calculation can be performed as explained in Section 3.3.

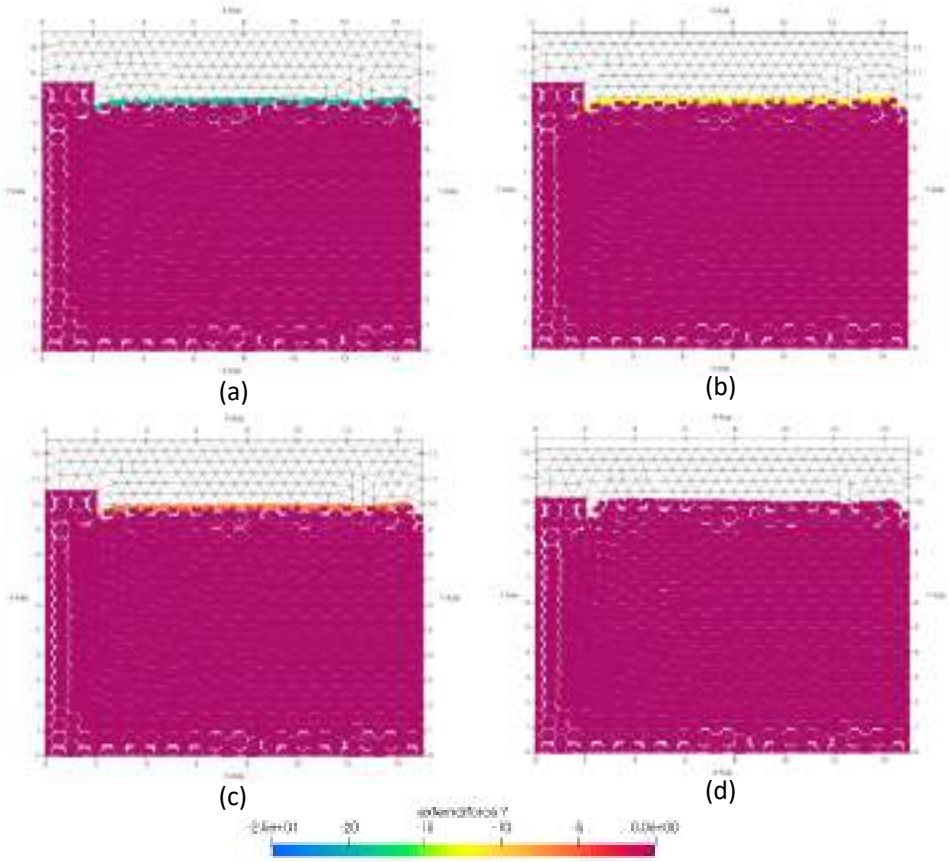
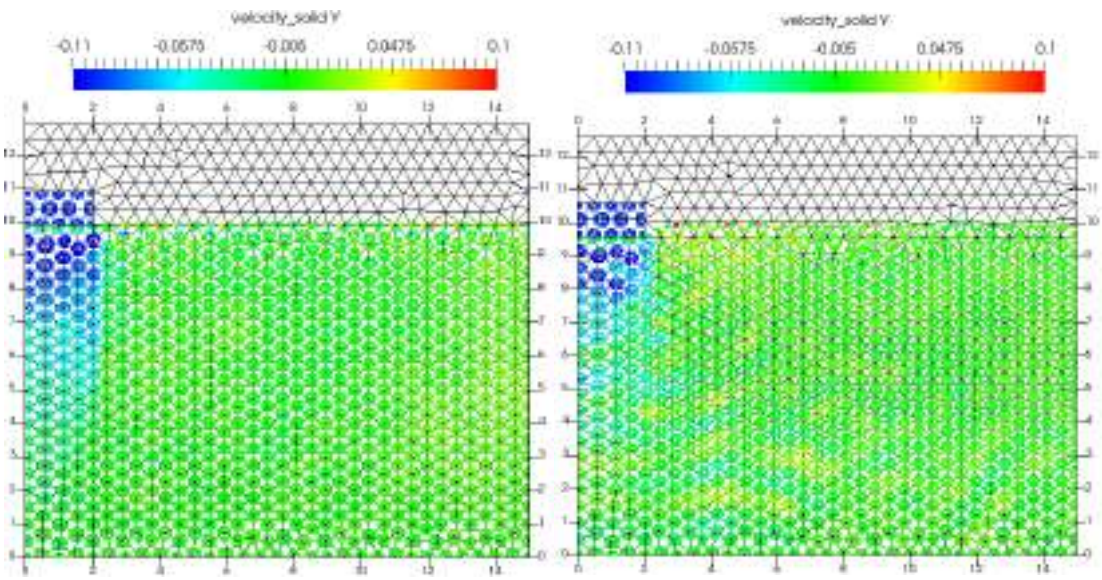


Figure 8.25: *Evolution of external force carried by the material points as load B is decreased from* 100kPa *to zero.*

8.4.4 Visualisation of results

To open the results with ParaView software (see Sections 2.5 and 2.1.3) proceed as explained in Section 3.4. Figure 8.27 shows the vertical velocity of material points, where it could be see than the velocity of the foundation is -0.1m/s.

Figure 8.26: *Dialogue box to define calculation parameters in GiD for the shallow foundation problem.*



(a) t=0,5s (b) t=4s

Figure 8.27: *Vertical velocity at MP.*

8.5 Variations

The variations proposed for the shallow foundation problem are listed below:

(1) Examine the influence of the fricton angle of the soil on the failure mechanism and the value of the ultimate bearing capacity.
(2) Use the contact formulation and study the effect of soil–foundation contact properties on the results.
(3) Consider a square footing and solve the problem with the 3D model.
(4) Consider a circular footing and solve the problem with the 3D-cylindrical formulation (see A).

9 Impact problem

Authors: Luis Zambrano-Cruzatty; Alba Yerro; John Murphy

This tutorial shows how to use the features of Anura3D to simulate impact on soils. The goals of this example are:

- ◇ Apply initial velocity on the material points.
- ◇ Use the moving mesh feature.
- ◇ Record reaction forces.
- ◇ Use the rigid body algorithm.
- ◇ Create an axisymmetric domain using 3D cylindrical boundary conditions.

First, a wall impacting soil is simulated using 2D plane strain conditions (Figure 9.1a). Afterwards, a variation is proposed considering an axisymmetric domain using 3D cylindrical coordinates simulating free fall impact into soil (Figure 9.1b).

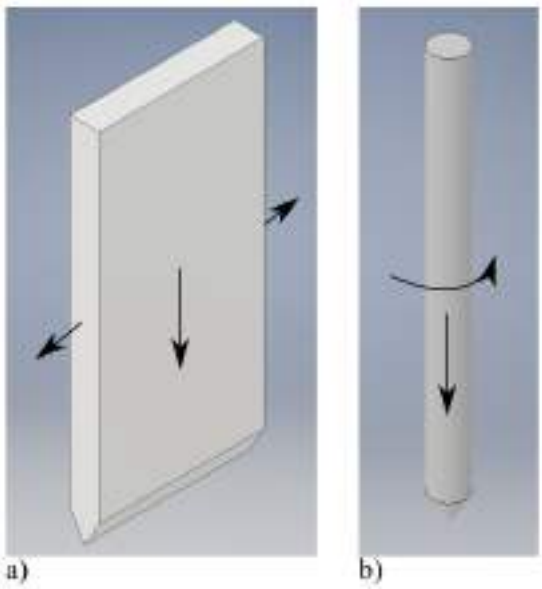


Figure 9.1: *Impacting body. a) long wall, b) free fall penetrometer.*

The moving mesh feature is explained in chapter 8. It is important in this problem because of the following reasons:

- ◇ The moving mesh prevents mixing of material points in elements. Therefore, the material points of the impacting body are always on the same element thought the calculation.
- ◇ The contact surface is well defined. Thus, the contact algorithm and the reaction forces features can be used.
- ◇ Fine meshing can be defined along the impacting body, avoiding unnecessary refinement and reducing computational resources in other areas.

Likewise, the rigid body algorithm is introduced in this chapter. The use of the rigid body algorithm can accomplish the following:

- ◇ Computational resources are saved because stresses are not integrated in the rigid material domain.

◇ Further computational resources are saved because the critical time step is not longer affected by the stiffness of the rigid body.

9.1 Problem definition

First, the problem is simulated in 2D-plane strain condition (in 9.4 a 3D cylindrical axisymmetric simulation is proposed). The geometry and the dimensions of the problem are given in Figure 9.2 and Table 9.1. Taking advantage of the symmetry of the problem, only one-half of the model is discretized in order to reduce the computational time.

The impacting body is simulated using i) linear elasticity in this section, and ii) as a rigid body in section 9.3. The soil is modeled with a cohesionless Mohr-Coulomb model. The properties for both materials are given in Table 9.2.

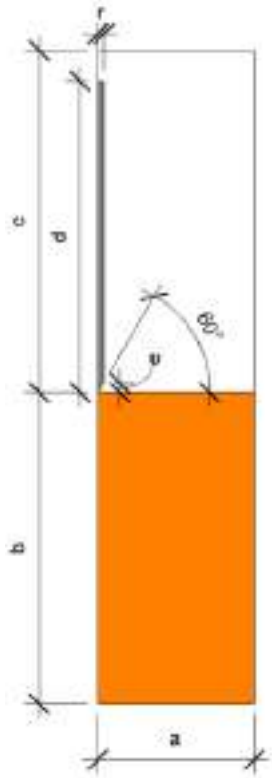


Figure 9.2: *Geometry of the impact problem.*

Table 9.1: *Dimensions of the impact problem.*

dimension	value
a	0.5m
b	1m
c	1.1m
d	1m
e	0.0309m
r	0.0179m

Table 9.2: *Material parameters of the impacting body and the soil.*

Material	Impacting body	Soil
Material type	Dry material	Dry material
Initial porosity [–]	0.0	0.5
Density solid [kg/m^3]	3500	2650
K0-value [–]	0.5	0.6
Material model solid	Linear Elasticity	Mohr-Coulomb
Young modulus [kPa]	50000	5000
Poisson ratio [–]	0.33	0.33
Cohesion [kPa]	-	0.5
Friction angle [°]	-	40
Dilatancy angle [°]	-	0.0
Tensile strength [kPa]	-	0.0

9.2 Creating input data

The geometry, material properties, fixities, number of material points, initial calculation parameters, and mesh are defined using the GiD pre-processor. This example follows a similar procedure as described in sections 3.2 and 8. The application of initial velocity on the material points is illustrated below.

9.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. In the “General Data” tab, set the dimension to “2D: plane strain”. In addition, activate the output for ParaView and deactivate outputs for GiD post-processors as indicated in Figure 9.3.

Figure 9.3: *Starting the GiD pre-processor.*

Save the project in the desired directory on the hard disk with an appropriate name (e.g. "impact_simulation").

9.2.2 Defining geometry

The geometry can be generated following instructions given in Section 3.2.2 considering the dimensions given in Table 9.1. The key coordinates indicated in Figure 9.4a and Table 9.3 can be used for reference.

9.2.3 Defining material properties

Before specifying material properties ensure that the problem type has been set as described in Section 3.2.1 step (2).

This impact problem uses two materials; 1) the soil which is simulated as a non-cohesive material, and 2) the impacting body that is usually rigid compared to the soil. The impacting body is considered elastic, and the material model for the soil is the Mohr-Coulomb. Material parameters are given in Table 9.2. As described in section 8.2.4, the Young modulus of the

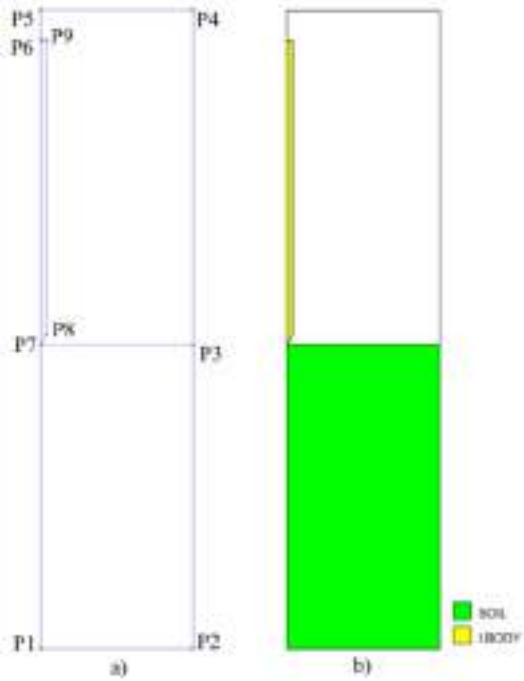


Figure 9.4: *Geometry and material specification of the impacting cone. a) key points (Table 9.3); b) material specification.*

Table 9.3: *Key coordinates for the impact problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	0	-1
P2	0.5	-1
P3	0.5	0
P4	0.5	1.1
P5	0	1.1
P6	0	1
P7	0	0
P8	0.0179	0.0309
P9	0.0179	1

impacting body is quite small compared with its real stiffness.

First, create the materials dictionary as explained in section 3.2.5. After completion the list of materials should look like it is shown in Figure 9.5a.

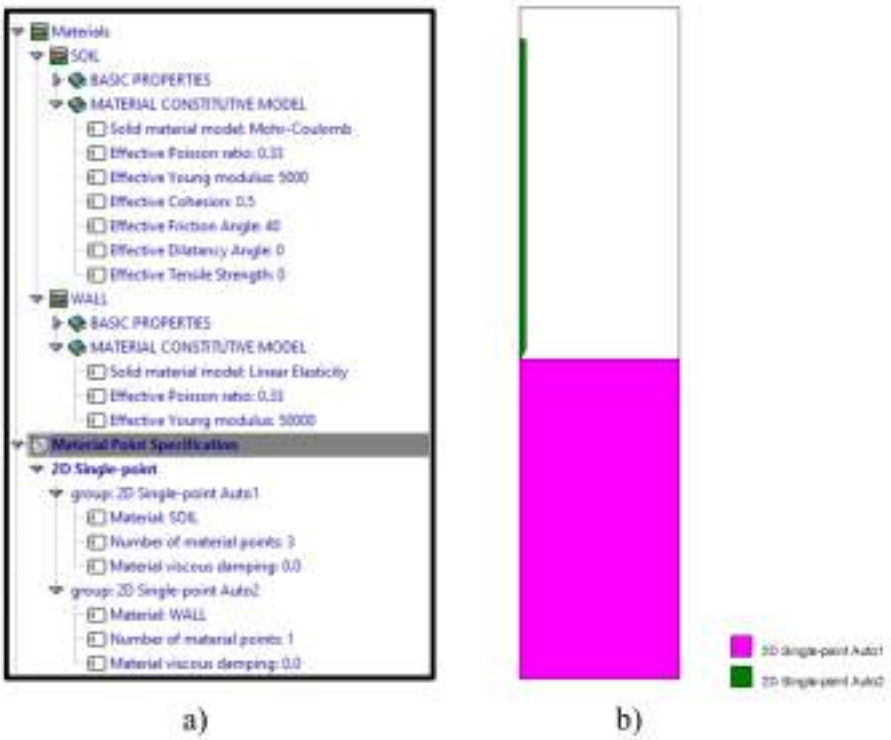


Figure 9.5: *Creating and assigning materials for the simulation of impact simulations.Subplot a) shows the view of the problem type tree, and b) materials are assigned to their corresponding areas.*

9.2.4 Specifying material points and assigning materials

Follow the instructions in Section 3.2.5 to define each material and assign them to their corresponding areas. Once the materials are assigned the problem should look like is shown in Figure 9.4b. The properties of the materials are also shown in the problem type tree in Figure 9.5a.

Specify the initial material point distribution following instructions from Section 3.2.6. In this example choose one (1) material point per element for the impacting body, and three (3) material points per element for the soil. Check if the material points are assigned correctly as shown in Figure 9.5b.

9.2.5 Defining initial conditions

The stresses in the soil will be initialized at the very beginning of the calculations employing the K_0-procedure. The "Apply K0-procedure" option has to be switched to "Yes" and "Soil surface level" must be set to "0.0 m" (Figure 9.6). This option is found in the Anura3D tree menu under the section "Initial conditions" $\rightarrow$ "Stress initialization" $\rightarrow$ "Horizontal layered soil"

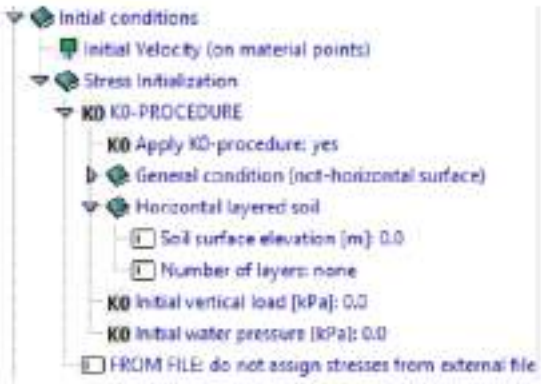


Figure 9.6: *Initial stress conditions definition.*

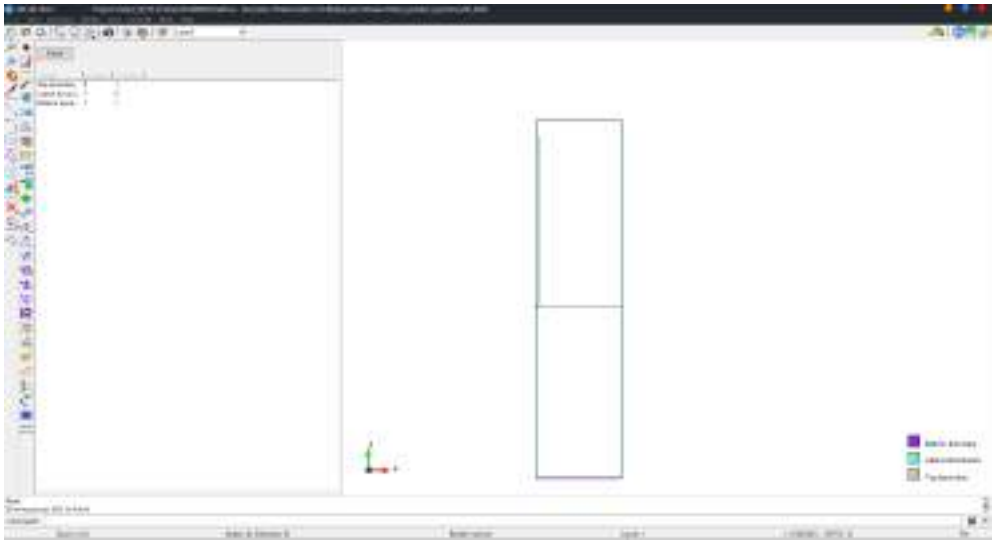


Figure 9.7: *Solid fixities of the impact problem in GiD.*

9.2.6 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying fixities

Since the materials in this example are in dry conditions, only the solid phase conditions are needed. The solid displacement at the boundaries are fixed as follows:

(1) Left side: horizontal fixity in x-direction on all three lines;
(2) Right side: horizontal fixity in x-direction on both lines;
(3) Top side: vertical fixity in y-direction on the top line;
(4) Bottom side: vertical fixities in y-direction on the bottom line;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 9.7;

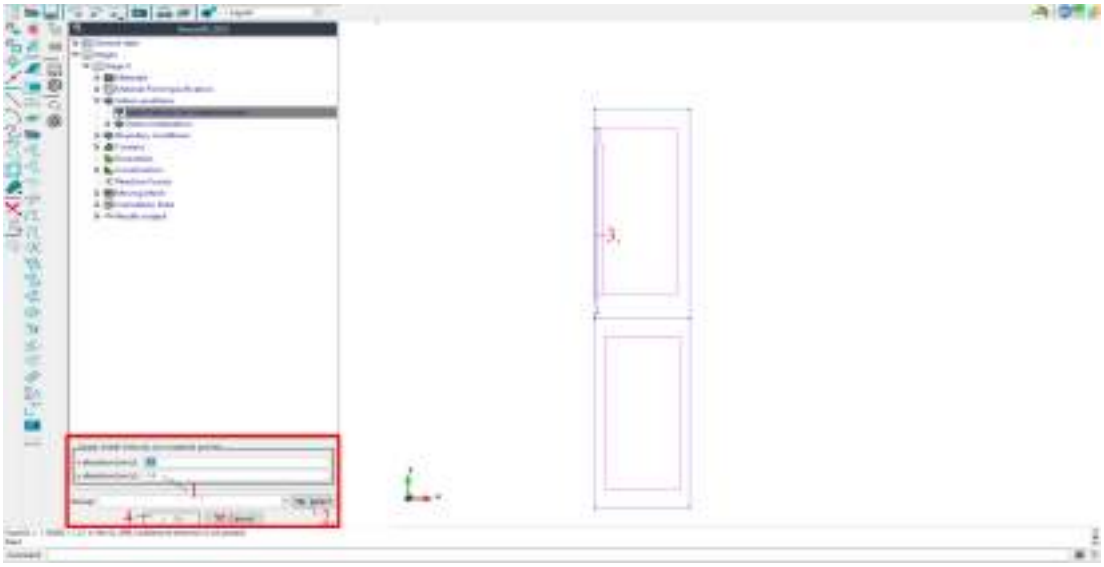


Figure 9.8: *Steps required to apply initial velocity to material points.*

9.2.7 Defining initial velocity on material points

The initial impact velocity is assigned to the material points belonging to the impacting body (note that the impacting body tip is initially placed at contact with the soil). An initial velocity of 5 m/s is assigned to the falling body expanding the "Initial conditions" drop-down menu and clicking on the "Initial Velocity (on material points)" option as shown in Figure 9.8:

(1) Specify 5 m/s in the y-direction text box as shown in Figure 9.8;
(2) CLick on the select button as shown in Figure 9.8.
(3) Assign the condition to the region belonging to the impacting body;
(4) Click on the "OK" and a new group condition will appear under the "Initial Velocity (on material points)" menu as shown in Figure 9.8.

You can check that the condition was correctly imposed doing right click over the new group and then clicking on the "draw values" option.

9.2.8 Defining a surface to record reaction forces

Two surfaces are defined for computation of reaction forces: 1) The surface at the tip of the impacting body, and 2) the lateral surface along the impacting body. Follow the instructions explained in section 8.2.8 to assign the reaction surfaces. Figure 9.10 shows the considered reaction surfaces.

Figure 9.9: *Visualizing the application of initial velocity on material points.*

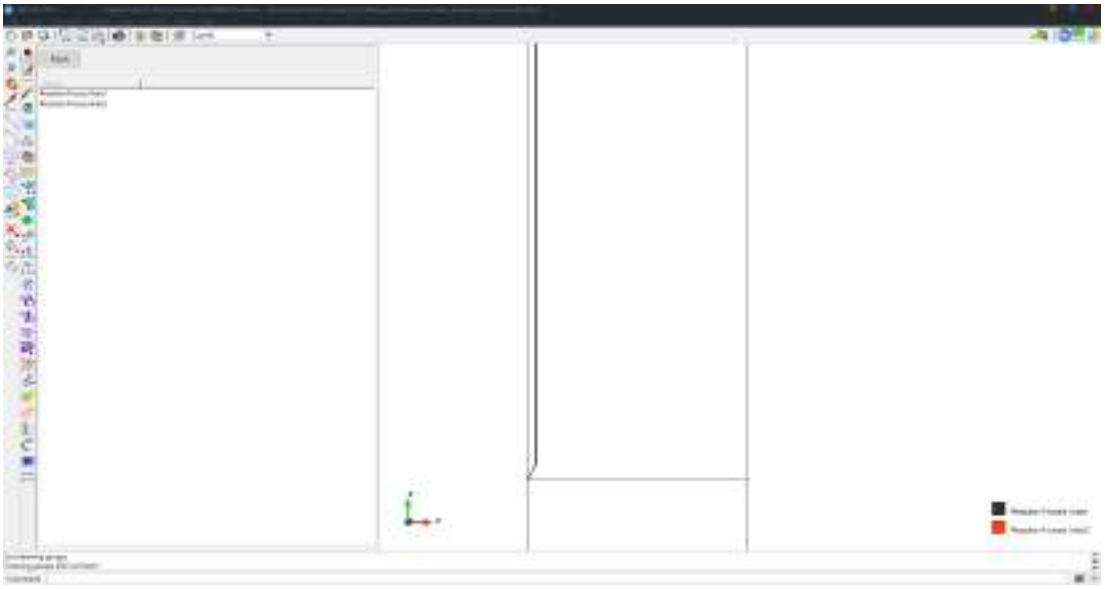


Figure 9.10: *Definition of the surface for computing reaction forces.*

9.2.9 Defining the moving mesh

In this problem, the moving mesh feature is used. The compressing mesh, the moving mesh, and the reference material can be defined as shown in section 8.2.9. Figure 9.11 shows the definition of the moving mesh for the impact problem.

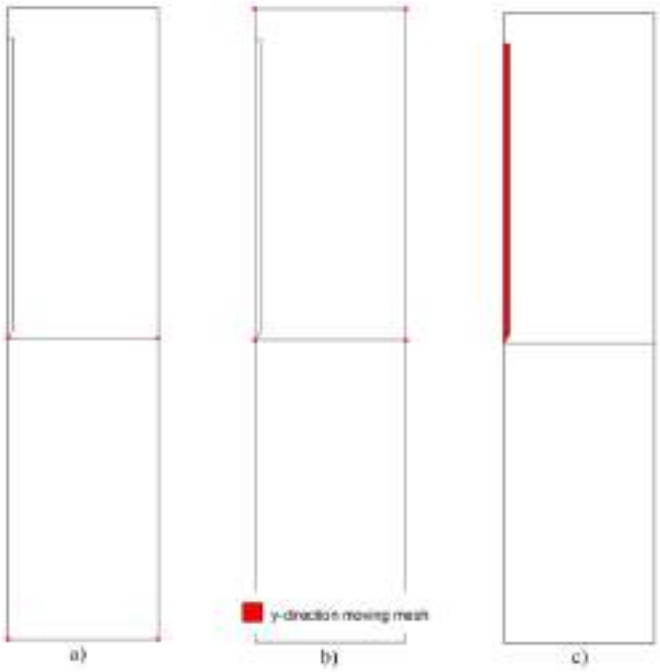


Figure 9.11: *Definition of the moving mesh. a) compressing mesh corners; b) moving mesh corners; c) reference material*

9.2.10 Creating mesh

An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example set the mean element size to "0.02". The mesh should look as shown in Figure 9.12.

9.2.11 Calculation parameters

The calculation will be carried out in a single stage. For this example, the following configuration is used.

Computation method
The computation method used in this example is the Gauss point integration. Therefore the option "MPM - mixed integration" should be selected.

Calculation step data
The simulation is carried out in 25 calculation steps. Each step will be 0.005 s long. A Courant number of 0.98 is advised to ensure the stability of the solution.

Gravity Load and Multipliers
The K_0-procedure is used to initialize the stress in this example, the gravity is fully applied from the beginning. The gravity multipliers will be maintained constant throughout the calculation. Therefore, either "apply gravity load - step" or "apply gravity load - linear" are acceptable Gravity Load selections. Set the multipliers for gravity loading as 1.0 for both the initial and final values. In this example the external load will not be used; therefore the multipliers for external load must be 0.

Strain smoothing

Figure 9.12: *Computational mesh of the impact problem.*

The strain smoothing setting is switched to "apply strain smoothing" to minimize the kinematic locking. Strain smoothing is applied to the volumetric strains.

Output data

Set the “Calculation steps between output” under the “ParaView Output” tab to 1 as shown in Figure 9.13. This will generate output files for each calculation step. The output files will be used to visualize the results in ParaView.

Three material points located at the bottom, middle, and top part of the impacting body will be selected for which additional output files will be created. First, make sure the mesh is visible and then click on “Select elements for MPs output” as shown in Figure 9.13. A new window will appear where the material points can be selected. Select the material points as shown in Figure 9.13. Select 1 material point per element and change the “Time between outputs” to 1×10^{-4} s, to ensure good data resolution.

Default values
All other calculation options are not used in this example, and the provided default values can be kept.

To define the initial calculation parameters follow the instructions detailed in Section 3.2.9. For this example, enter the values given below as shown in Figure 9.13.

- select "COMPUTATION METHOD" as "MPM - mixed integration" from the drop down menu
- set "number of calculation steps [-]" to "25"
- set "time per calculation step [s]" as "0.005".
- set "Courant number" to "0.98"
- set "GRAVITY LOAD" to "apply gravity load - linear"
- set "gravity multiplier initial and final [-]" to "1.0" and "1.0"
- set "STRAIN SMOOTHING" as "apply strain smoothing"
- select "number of material points" as "3" from the drop down menu

All other calculation parameters can be kept at their default values.

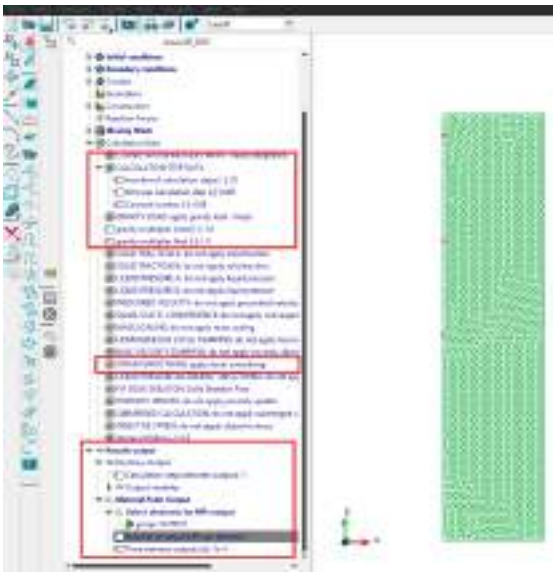


Figure 9.13: *Dialogue box to define calculation parameters in GiD for the impact problem.*

9.2.12 Performing calculation in Anura3D

The calculation can be performed as explained in Section 3.3.

9.2.13 Visualisation of results in ParaView

ParaView software (see Sections 2.5 and 2.1.3) is used to visualize the results of the simulation. Instructions presented in Section 3.4 can be observed.

Figure 9.14 presents the evolution of the deviatoric strain during the impact problem.

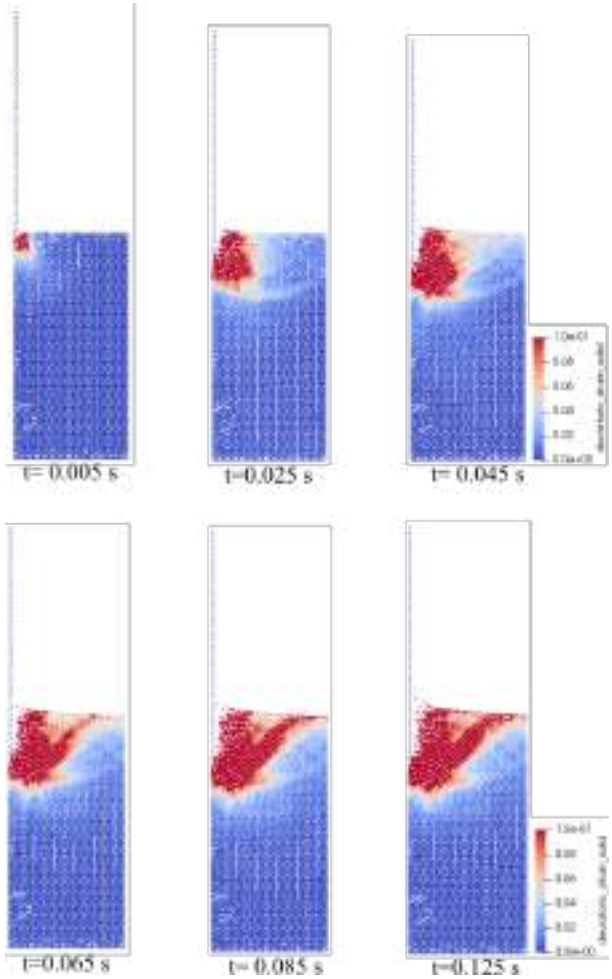


Figure 9.14: *Evolution of the deviatoric strain.*

9.2.14 Visualisation of text output files

Following the instructions from Section 3.6, the evolution of the penetration depth vs. time can be plotted for points belonging to the impacting body. Results are presented in Figure 9.15.

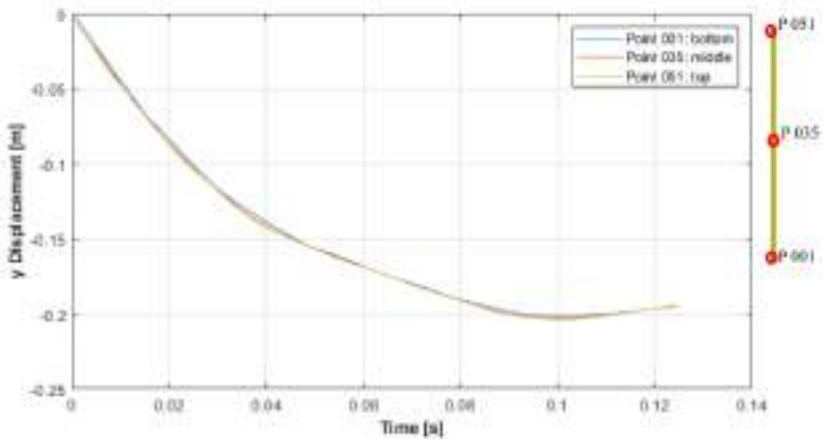


Figure 9.15: *Penetration depth vs. time.*

The reaction surfaces chosen for this example are those required to observe the tip (RSurf_001) and shaft resistance (RSurf_002). Section 8.3.5 presents detailed instruction to open and visualize Rsurf files. Figure 9.16 shows the tip and shaft force developed as function of the penetration depth. Note that the results contained in a Rsurf file are force vs. time, but because the PAR and Rsurf file contain results for the same time domain, their results can be plotted together.

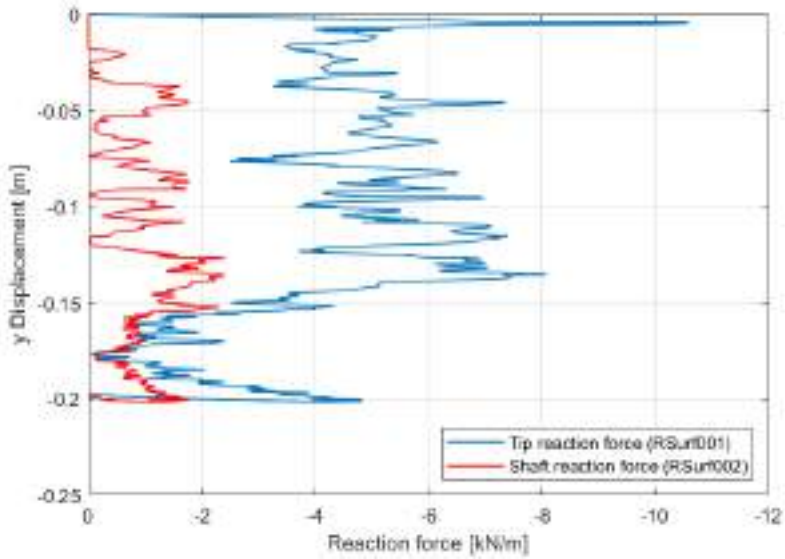


Figure 9.16: *Tip and shaft reaction forces (from RSurf files) vs. penetration depth (from PAR files).*

9.3 Variation: Simulation of the impacting body as a rigid body

In this subsection, a variant of the impact problem is proposed using the rigid body algorithm. Note that for this geometry, the impact body represents a long wall (Figure 9.1a). In the previous example, the material was considered linear elastic with Young's modulus of 50,000 kPa. This stiffness is lower than the modulus of steel and concrete. The implementation of realistic elastic modulus of these materials is computationally expensive, because the critical time step is drastically reduced. To simulate rigid motion, Anura3D uses a simple but efficient algorithm to simulate rectilinear movement. A detailed explanation of how the algorithm works can be found in [7].

The model is based on the same geometry and boundary configurations as the previous example (section 9.2.12), hence the file created for the previous model can be used as a starting point.

9.3.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in subsection 3.2.1. Save the file as a new GID problem with a consistent name (e.g. Impact_v2.GID).

9.3.2 Defining geometry

The geometry can be defined following the instructions in subsection 9.2.2.

9.3.3 Defining material properties

It is important to follow the next steps to ensure the correct functionality of the rigid body:

(1) The user **must define the rigid body material last**, i.e. after all other materials are defined. This is because the last material index is internally reserved for the master contacting body.
(2) In case the impacting body material is not the last material, it is recommended to delete all materials and start from the beginning.
(3) To assign a material as a rigid body, select "rigid body" in the material model solid drop list shown in Figure 9.17.
(4) Specify the motion constraints. In this variation, the motion must be solely in the y-direction; hence, the x-direction must be constrained.
(5) Save the clicking in the "OK" button.
(6) Check that the materials have been assigned correctly by clicking on "draw" and then "draw values."

Figure 9.17: *Rigid body material assignation.*

9.3.4 Specifying material points

If this variation was created based on the previous example, there is no need to modify the material points configuration. Otherwise, follow the steps indicates in section 9.2.4.

9.3.5 Defining boundary conditions

The boundary conditions are the same as in section 9.2.6. If the previous model files were used as a base for this variation, there is no need to modify the fixities.

9.3.6 Defining initial velocity on material points

Similarly as in section 9.2.7, assign the velocities to the surface representing the wall. The velocity to be assigned is –5m/s on the y-direction.

9.3.7 Defining contact properties

It is important to highlight that the rigid body algorithm requires the activation of the contact algorithm. Hence, it has to be assigned and activated in the CPS file. In this variation, we will simulate a fully rough contact (to compare results with example showed in section 9.4.12). Therefore, assign the following properties to the contact: 2 for the friction and 10 kPa for the adhesion (Figure 9.18).

To learn more about the contact algorithm, refer to chapter 6.

Figure 9.18: *Contact properties.*

9.3.8 Defining a surface for computation of reaction forces

Follow the instructions explained in sections 8.2.8 and 9.2.8 to assign the reaction at surfaces. Select two (2) surfaces: the tip and the shaft.

9.3.9 Defining the moving mesh

The moving mesh is set in the same fashion as demonstrated in section 9.2.9. If this variation was constructed based on the previous model, there is no need to define the moving mesh again.

9.3.10 Creating the mesh

To generate the mesh follow the steps given in Section 6.1.5. The mesh is identical to the one created in section 9.2.10. Regenerate the mesh even if the previous model is used for this variation.

9.3.11 Calculation parameters

Follow the instructions in section 9.2.11 to assign the calculation parameters.

9.3.12 Performing calculation in Anura3D

The calculation is executed as explained in Section 3.3. The calculation should take around 5 to 6 minutes.

9.3.13 Visualisation of results in ParaView

The results can be visualized using ParaView. Figure 9.19 shows the evolution of the deviatoric strain when the impacting body is rigid in the right, and elastic (from previous model) in the left. Notice the similarities in the deviatoric strain contour plot.

9.3.14 Visualisation of text output files

A material point centered in the impacting body is tracked similarly as in subsection 9.2.14. The comparison between the elastic and rigid body displacement is shown in Figure 9.20. It can be observed that both solutions are similar. Only a small difference in the maximum displacement in the last part of the simulation is observed.

9.4 Variation: Simulation in 3D cylindrical axisymmetric conditions

In this section, a variation is proposed taking into account axisymmetric conditions. Note that in this case, the impacting body represents a free fall penetrometer (Figure 9.1b). In this case, it is recommended to start a new model from the beginning in order to avoid conflicting assignations between 2D and 3D models. The following instructions will guide you towards the construction of the axisymmetric case making use of the 3D cylindrical axisymmetric condition.

9.4.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "Impact 3D").

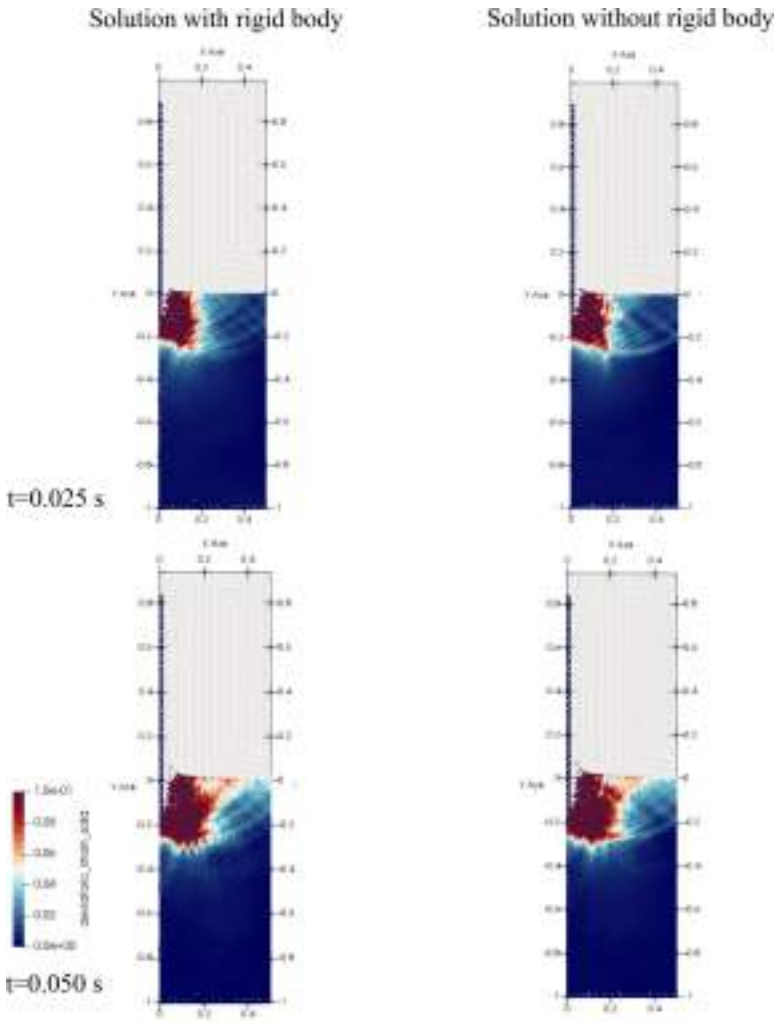


Figure 9.19: *Evolution of the deviatoric strain for the solution with rigid body and elastic material.*

9.4.2 Defining geometry

Start defining the same geometry defined in section (9.2.2). You can use table 9.3 to identify the corner coordinates of the 2D section. Then, a rotation of the 2D section is required to create a 3D axisymmetric geometry, representing a slice of the problem. Follow the next steps:

(1) Select the Utilities menu on the menu bar in GID;
(2) Click on the "Copy" option as shown in figure 9.21.
(3) The "Copy" dialog box will pop up (figure 9.21).
(4) Change "Entities types" to "Surfaces" (9.21).
(5) Change "Transformation" to "Rotation" (9.21).
(6) In the "Angle" text box write 20. This is the angle of the cylinder slice.
(7) In the "Second point" box, write y= 1 as shown in figure 9.21. This will indicate the axis of rotation along the y axis at x=0 and z=0. The code will require that **ALL** the 3D cylindrical models are rotated along this axis.
(8) Change the "Do extrude" option to "Volumes" as shown in figure 9.21.
(9) Click on the "select" buttom and chose all the surfaces of the actual geometry.
(10) Finally, click on "Finish".

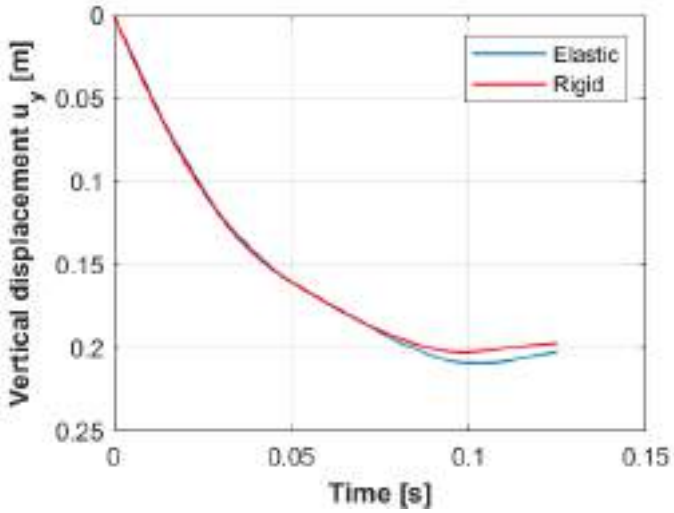


Figure 9.20: *Comparison between the vertical displacement vs. time for i) the elastic and ii) rigid impacting wall.*

A 20° slice of a 3D cylindrical model is generated as shown in figure 9.22.

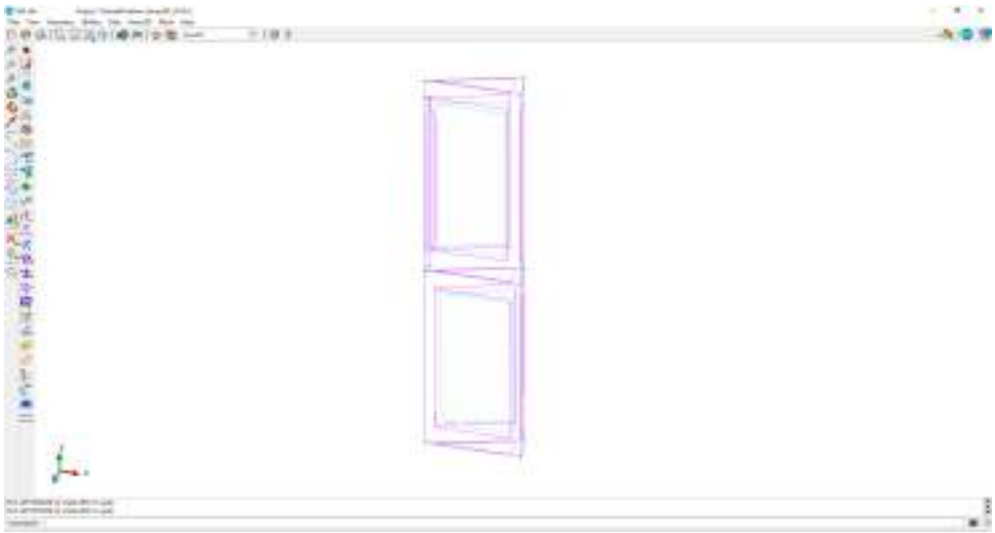


Figure 9.22: *20° slice of a cylinder to simulate an impact problem.*

9.4.3 Defining problem dimensions

Click on the "Units and dimensions" dropdown menu to change the problem dimensions to "3D Axisymmetric" as shown in Figure 9.23.

9.4.4 Defining material properties

Similarly as in section 9.2.3, assign the materials to the corresponding volumes.

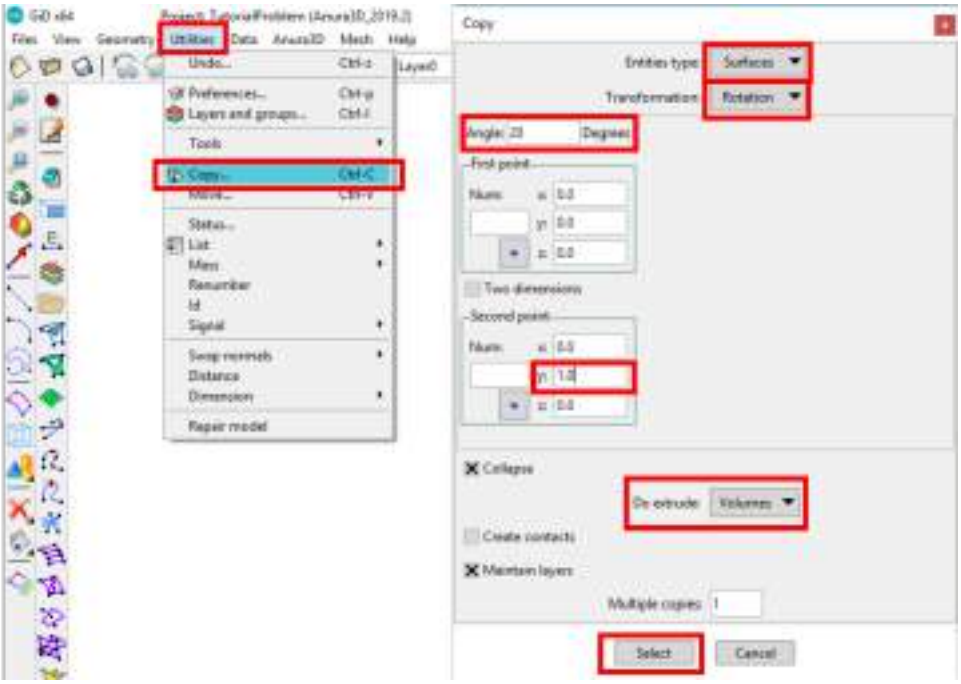


Figure 9.21: *Copy Dialog box setup to create a 20° cylindrical slice.*

9.4.5 Specifying material points

Assign one (1) material point to all the material volumes (i.e. impacting body and soil) (consider section 9.2.4 for reference). Note that in section 9.2.4 three (3) points were assigned to the soil instead. The reason for reducing the number of material points is to minimize the computational resources required to solve this tutorial exercise. Consider more material points and mesh refinement if more accurate results are required.

9.4.6 Defining boundary conditions

Note that for cylindrical coordinates, the x, y, and z directions are replaced by the r, y, and θ respectively.

Applying fixities

Similarly as in section 9.2.6, only the solid phase needs to be considered. In order to avoid inconsistent fixities on the intersections of the geometrical items, the fixities will be assigned to the surfaces, boundary lines, and corner points as indicated in figure 9.24.

◇ **Surface fixities**

(1) Lateral sides: normally fixed (θ direction only);
(2) Curved side: normally and tangentially fixed (r and θ direction);
(3) Top side: vertical fixity in y-direction on the top surface;
(4) Bottom side: vertical fixities in y-direction on the bottom surface;
(5) Check if the fixities are assigned correctly, see Figure 9.24a;

◇ **Line fixities**

(1) Vertical lines: normally and tangentially fixed (r and θ direction);
(2) Curved lines: fully fixed (all directions);
(3) Top lines: normally fixed (y and θ direction);

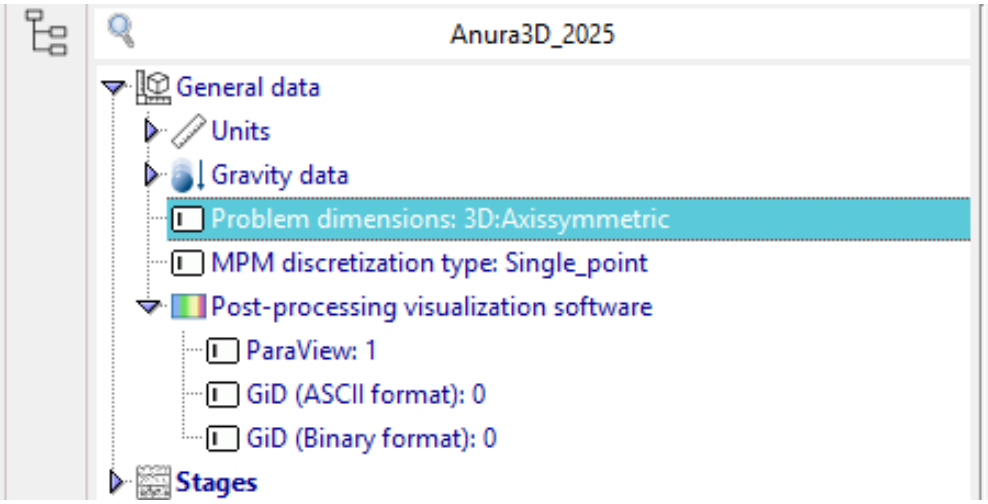


Figure 9.23: *Setting up the dimensions of the problem.*

(4) Bottom side: normally fixed (y and θ direction);
(5) Check if the fixities are assigned correctly, see Figure 9.24b;

◇ **Point fixities**

(1) Corner points: fully fixed (all directions);
(2) Check if the fixities are assigned correctly, see Figure 9.24c;

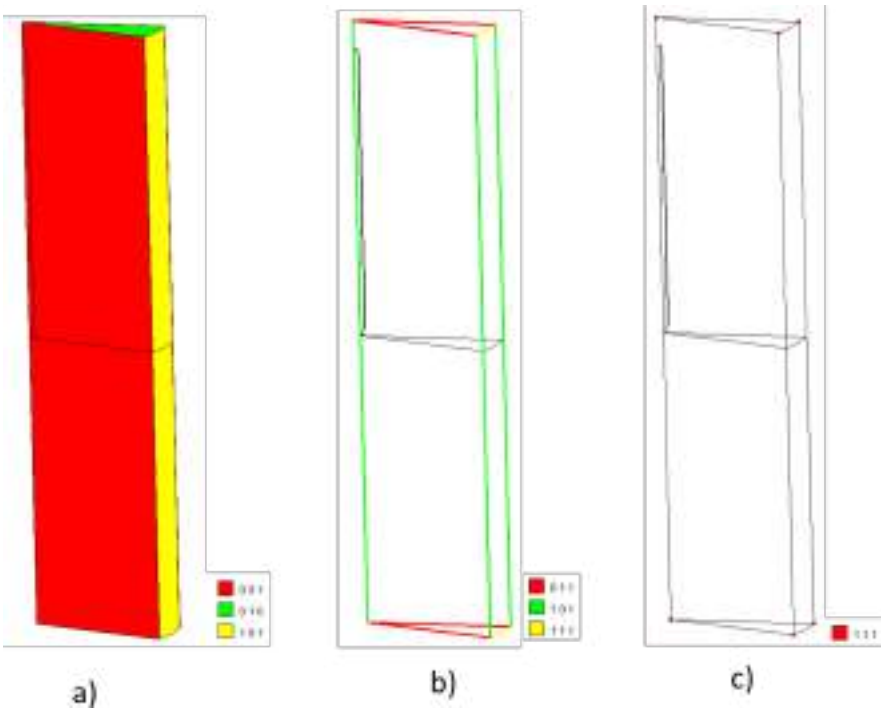


Figure 9.24: *Solid fixities of the impact problem in GiD for a 3D cylindrical problem, a) surface fixities, b) line fixities, c) point fixities.*

9.4.7 Defining initial velocity on material points

Similarly as in section 9.2.7, assign the velocities to the volume representing the cone. The velocity to be assigned is –5m/s on the y-direction.

9.4.8 Defining a surface for computation of reaction forces

Follow the instructions explained in sections 8.2.8 and 9.2.8 to assign the reaction at surfaces. Select two (2) surfaces: the tip and the shaft.

9.4.9 Defining the moving mesh

The moving mesh is set in the same fashion as demonstrated in section 9.2.9. Note that for the 3D cylindrical case, the compressive, and moving mesh areas are defined by 6 points each.

9.4.10 Creating mesh

An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example set the mean element size to "0.1". The mesh should look as shown in Figure 9.25. Note that the bigger mesh is selected only to reduce the computational resources required to follow this tutorial. Finer meshes are recommended to obtain more accurate results in the area of interest.

Figure 9.25: *Computational mesh of the impact problem in 3D cylindrical coordinates.*

9.4.11 Calculation parameters

The calculation parameters are identical to those specified in section 9.2.11.

9.4.12 Performing calculation in Anura3D

The calculation can be performed as explained in Section 3.3. This simulation should take around 5-6 min.

9.4.13 Visualisation of results in ParaView

The results can be visualized using ParaView. Figure 9.26 shows the evolution of the deviatoric strain. Note that the results are different from the plane strain condition because the mesh is coarser, and the problem is simulating a cone impacting body (Figure 9.1b) instead of an impacting wall (Figure 9.1a).

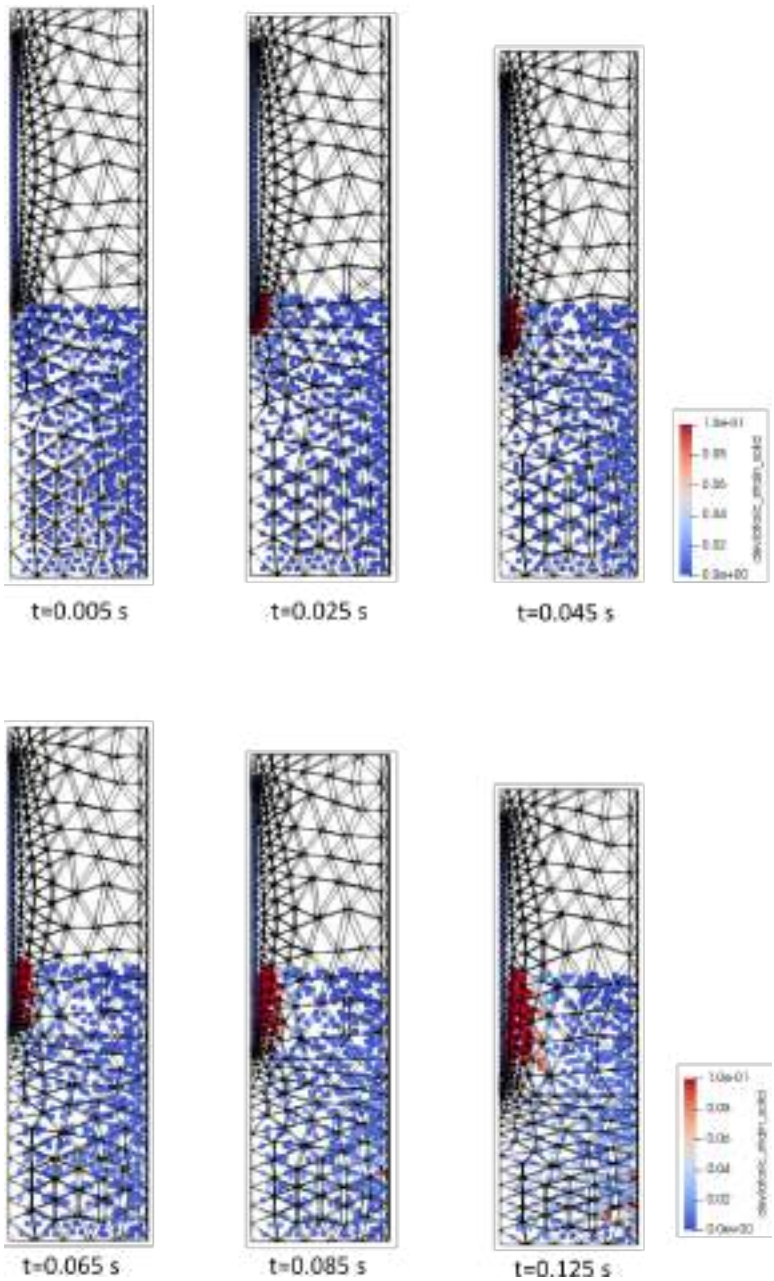


Figure 9.26: *Evolution of the deviatoric strain.*

9.5 Other variations

Other variations proposed for the impact problem are listed below:

(1) Examine the influence of the material properties on the penetration depth.
(2) Examine the importance of the impacting body weight on the penetration depth.

10 Excavation

Authors: Alexander Chmelnizkij; Angela Saade; Francesc Mirada; Alba Yerro; Gaia Di Carluccio; Nuria Pinyol; Bruno Zuada; James Fern; John Murphy; Francesca Ceccato

In the following tutorial, two examples of the excavation feature implemented in Anura3D are described. First, a 3D column of soil is considered and four volumes of soil are removed progressively from the top of the column. Two variations of the problem are proposed considering different material models (Linear elasticity and Mohr-Coulomb). The second example looks at the stability of 2D supported vertical excavations in elasto-plastic material. Two different approaches to simulate the failure mechanism are considered. The first approach continuously removes material points as they rise inside the excavation. In the second approach, after the removal of MPs, soil is allowed to move into and accumulate in the excavated zone.

10.1 3D Column excavation

10.1.1 Problem definition

The dimensions of the soil column are given in Figure 10.1. Five equal volumes of soil are considered and an empty space is added on top of the column to allow material points to move freely. The dimensions are given in Table 10.1. Note that each excavation depth is H/5.

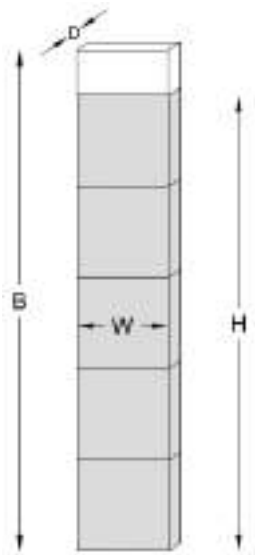


Figure 10.1: *Definition of the excavation problem including empty space at the top.*

Table 10.1: *Dimensions of the column for the excavation problem.*

Column height H(m)	Column width W(m)	Depth D(m)	Box Height (m)
5	1	0.1	5.5

10.1.2 Creating input data

The geometry, boundary conditions, material properties and mesh are defined using GiD pre-processor. The calculation options are defined in the *CPS* file. Both follow the same

Table 10.2: *Material parameters of the soil for the excavation problem.*

Material	Column
material type	dry material
initial porosity	0.25
density solid (kg/m^3)	2650
K0-value	0
material model solid	1-Linear Elasticity
	2-Mohr-Coulomb
Young modulus (kPa)	20000
Poisson ratio (-)	0.20
cohesion (kPa)	5*
effective friction angle (°)	30*
dilatancy angle (°)	0
tensile strength (kPa)	0

(*) specific to Mohr-Coulomb material model

procedure as described in Section 3.2. Only variations to previous examples are highlighted in the following sections.

10.1.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "columnexcavation").

10.1.2.2 Defining geometry

Create the geometry following the steps as described in Section 3.2.2 with dimensions as given in Table 10.1 of the problem description. Note that an entity containing empty elements should be defined at the top side of the column to allow for free movement of material points.

The geometry is created by defining 6 rectangles (in 2D) which will be discretised by a semi-structured mesh. The geometry should look like shown in Figure 10.2. Note the division into 6 prismatic volumes.

Figure 10.2: *Geometry of the column excavation problem including empty entity at the top.*

An index is automatically assigned to each volumes in GiD. This index cannot be changed and is needed for the excavation process. The labels can be obtained from the <View> <Label> <All on in> <Volumes> menu as shown Figure 10.3.

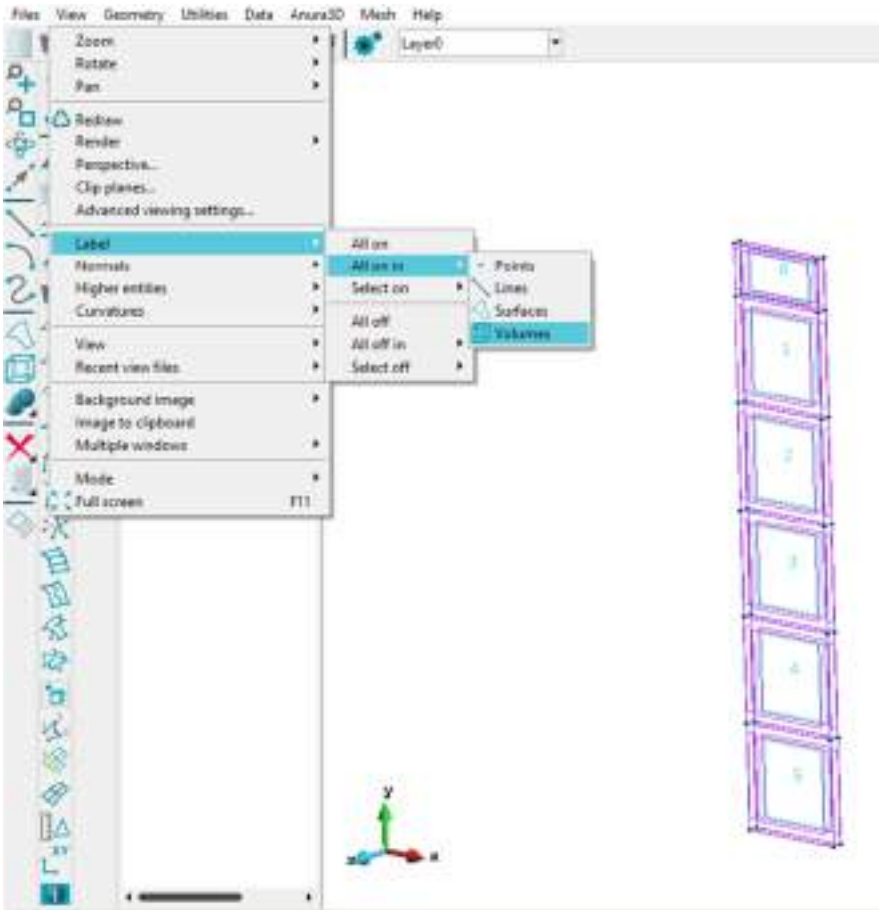


Figure 10.3: *Labels for the volumes in GiD.*

10.1.2.3 Defining material properties

Define the material properties following steps as described in Section 3.2.5. In this example, a linear elastic model is considered. Enter the material parameters according to Table 10.2 as shown in Figure 10.4.

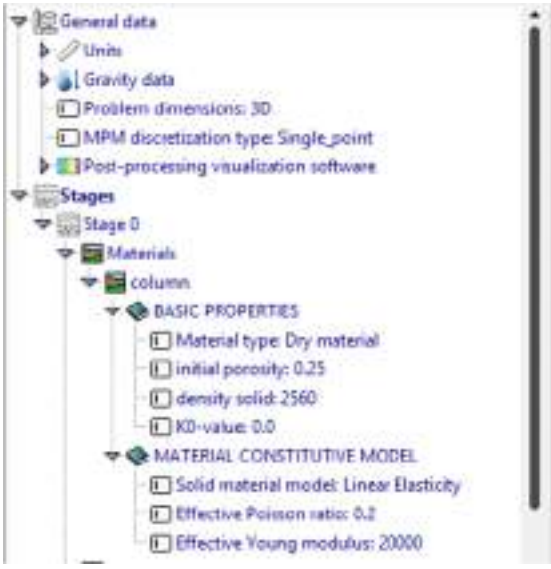


Figure 10.4: *Defining material properties for the column excavation problem, linear elastic model.*

10.1.2.4 Specifying material points

Specify the initial material point distribution following instructions in Section 3.2.6. Choose 4 material points per element for the column. Check if the material and the number of material points are assigned correctly (Figure 10.5).

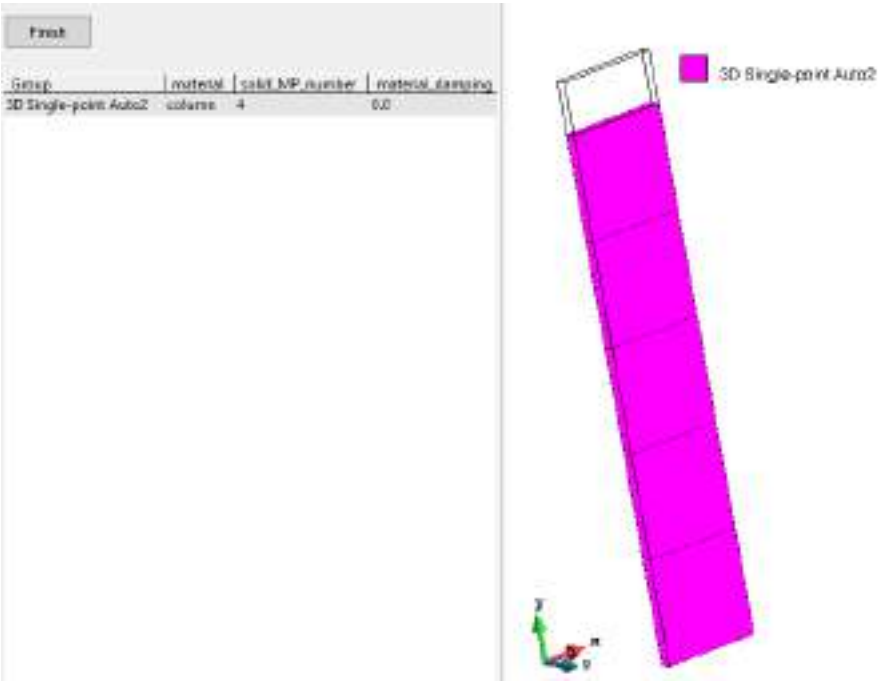


Figure 10.5: *Specification of the material and the number of material points per element for the column excavation problem.*

10.1.2.5 Defining boundary conditions

Define the boundary conditions following the steps given below:

(1) Left side at x=0.0: horizontal fixity in x-direction on the 6 surfaces
(2) Right side at x=1.0: horizontal fixity in x-direction on the 6 surfaces
(3) Top side at y=5.5: vertical fixity in y-direction
(4) Bottom side at y=0: fixities in x- and y- directions
(5) Front and back sides: horizontal fixities in z-direction

The assigned fixities of the column excavation problem should look like shown in Figure 10.6.

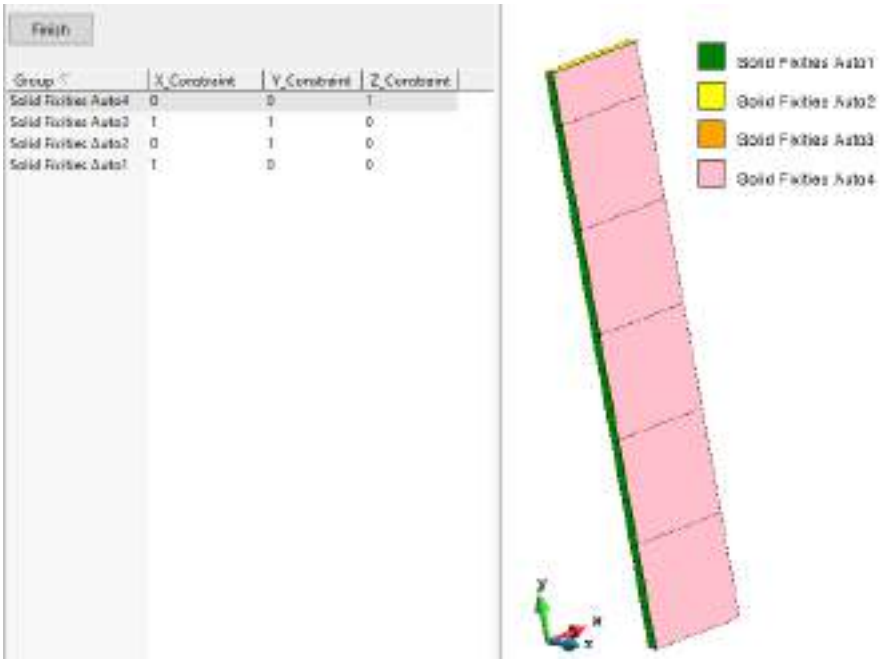


Figure 10.6: *Fixities of the column excavation problem in GiD.*

10.1.2.6 Creating mesh

For this example, a semi-structured mesh in z-direction is used with an element size of 0.1m. To generate the mesh follow the steps in Section 3.2.8. The number of divisions in the structured direction to assign to semi-structured volumes is 1.

10.1.2.7 Defining excavation steps

Anura3D provides an excavation tool which allows to simulate soil excavation in stages by defining the range of load steps in which the material points belonging to a certain volume have to be removed. In this example, the first load step consists of applying gravity load. The second load step includes the excavation of the first volume of soil, which is the volume no. 5 according to the labelling in Figure 10.2. Therefore, in order to define the excavation of volume no. 5, the first step of the excavation sequence should be set equal to 2. No soil is allowed to move into volume no. 5 during the entire process of excavation. Thus, the last step in the excavation sequence for volume no. 5 should be set equal to 6 (which corresponds to the final load step). The definition of the excavation sequence for volume no. 5 is shown in Figure 10.7.

Figure 10.7: *Definition of an excavation step.*

The assigned excavation step for volume no. 5 should look like shown in Figure 10.8

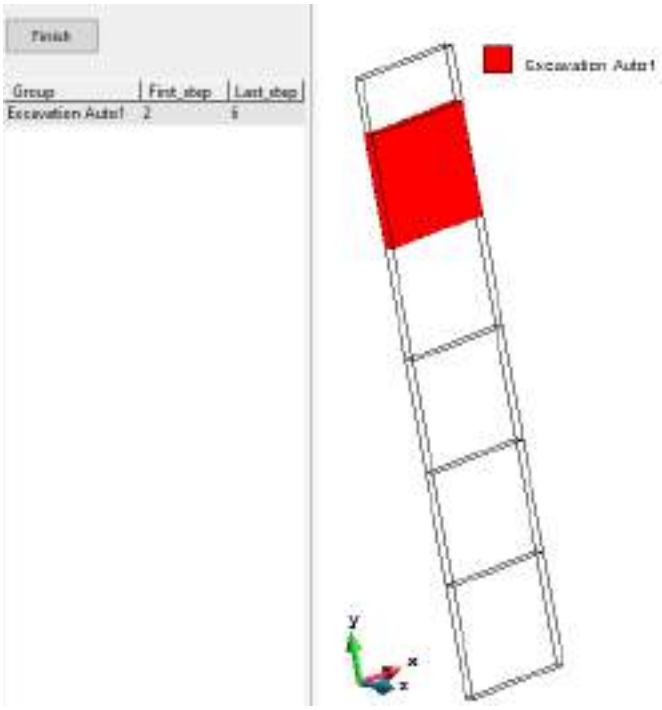


Figure 10.8: *Assigned excavation for volume no. 5.*

The next volume to be excavated is volume no. 4. The first step of the excavation sequence would be 3 whereas the last step would be 6. Proceed similarly to define and assign the excavation steps for volumes 4, 3 and 2. The final setup of the excavation process should look like shown in Figure 10.9. Once defined the excavation steps, the mesh should be regenerated.

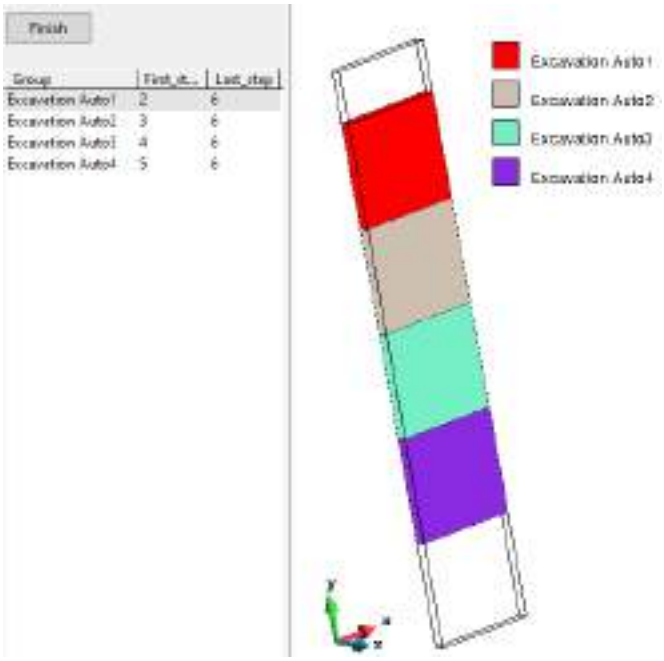


Figure 10.9: *Assigned excavation steps.*

10.1.2.8 Initialisation of calculation data

In the 2025 version, there is no need to separately define data in CPS files. Instead, all settings can be defined in GiD across different stages.

10.1.2.9 Stage0: Gravity loading

Only the calculation parameters required to initialize the first load step calculation are added in stage0. The calculation parameters for subsequent load steps will be adjusted in stage1.

Computational method
The computational method used in this example is the "MPM - mixed integration".

Calculation step data
The first calculation in stage0 runs a quasi-static calculation to initialize stresses in the soil. This initial stage occurs in one Anura3D load step. The end of each calculation step is determined by the convergence criteria. A Courant number of 0.98 is advised to ensure the stability of the solution.

Load multipliers - Gravity Load
The full gravity load is applied from the start of the calculation and held constant for the quasi-static load step. Therefore either applygravityload – linear or applygravityload – stepwise can be used. Set the gravity multiplier initial and final to 1.0.

Quasi-static convergence
The determination of the end of each calculation step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition.

Homogeneous local damping

In order to converge to the quasi-static solution an overall local damping coefficient for all active elements of 0.75 is applied.

Output data
One material point located at the bottom of the column will be selected for which an additional output file will be created. Section 3.4.7 explains how to determine the ID number of a material point.

To define the initial calculation parameters follow the instructions given in Section 3.2.9. For this example, enter the values as given below:

- Select "COMPUTATION METHOD" as "MPM - mixed integration" from the drop down menu
- Set "number of calculation steps [-]" as "1"
- Set "time per calculation step [s]" as "0.5". Note that any value except 0.0 can be entered since the quasi-static convergence criterion is used
- Set "GRAVITY LOADING" as "apply gravity load - stepwise" from the drop down menu
- Set "gravity multiplier initial [-]" to "1.0"
- Set "gravity multiplier final [-]" to "1.0"
- Set "QUASI-STATIC CONVERGENCE" as "apply convergence criteria"
- Set "HOMOGENEOUS LOCAL DAMPING" as "apply homogeneous local damping" from the drop down menu
- Set "Local damping coefficient" to "0.75"
- Set "Material Point Output" - "Select element for MPs output" - "Number of output MPs per element" to "1"
- Set "Time between output[s]" as "0.01"
- Select element which is closed to the coordinate (0.268; 0.1819; -0.01382)

All other calculation parameters can be kept at their default values.

10.1.2.10 Stage1: Excavation

By copying Stage0, define second stage for excavation. All settings remain the same; only the number of load steps is increased to 6 to include the 4 excavation steps and a final stage.

10.1.3 Performing calculation in Anura3D

Click "Calculate" on the top bar in Anura3D. Input errors and calculation errors are displayed in the window as well.

10.1.4 Visualisation of results in ParaView

The results for the dry soil column with linear elastic constitutive model are presented for the end of the gravity loading stage and at the end of each excavation stage. Effective stresses in y- and z-direction are shown for each stage in Figures 10.10 and 10.11.

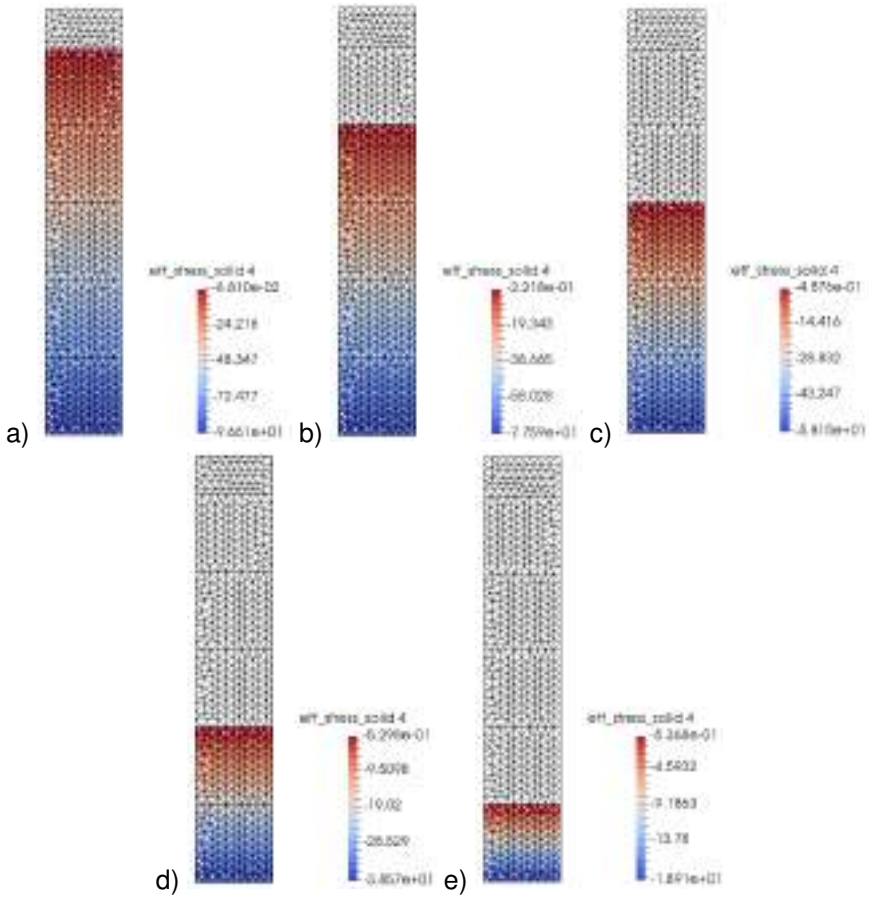


Figure 10.10: *Vertical effective stresses σ_{yy} in [kPa]: a) after gravity loading, b) after excavating volume 5, c) after excavating volume 4, d) after excavating volume 3, e) after excavating volume 2.*

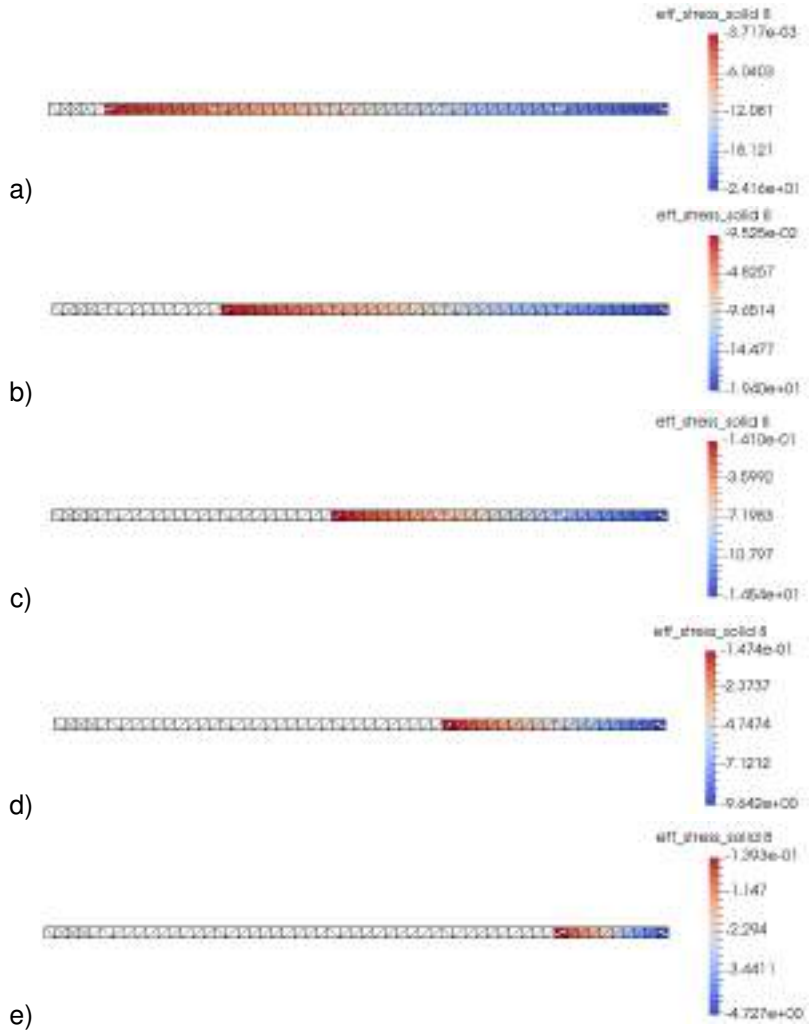


Figure 10.11: *Horizontal effective stresses σ_{zz} in [kPa]: a) after gravity loading, b) after excavating volume 5, c) after excavating volume 4, d) after excavating volume 3, e) after excavating volume 2.*

10.1.5 Visualisation of text output files

Following the instructions from Section 3.6, further results can be plotted for a material point located at the bottom of the column which is shown in Figure 10.12.

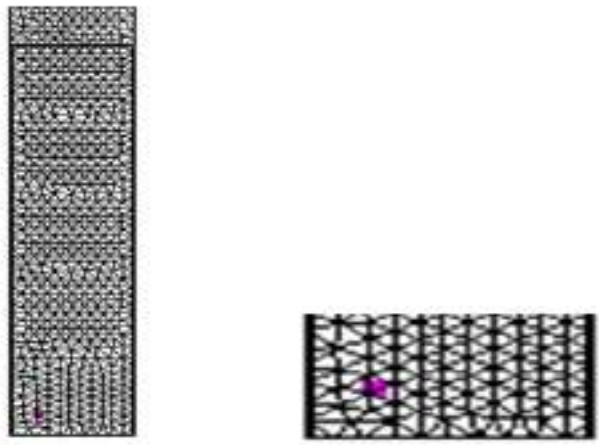


Figure 10.12: *Location considered for results which has a material point ID equal to 1228 and coordinates (0.268; 0.1819; -0.01382).*

Figure 10.13 presents the stress–strain curve and shows that the behaviour is completely linear elastic when loading and unloading. The theoretical oedometric modulus is obtained using the following equation:

$$E^0 = \frac{1-\nu}{(1+\nu)(1-2\nu)}E \tag{10.1}$$

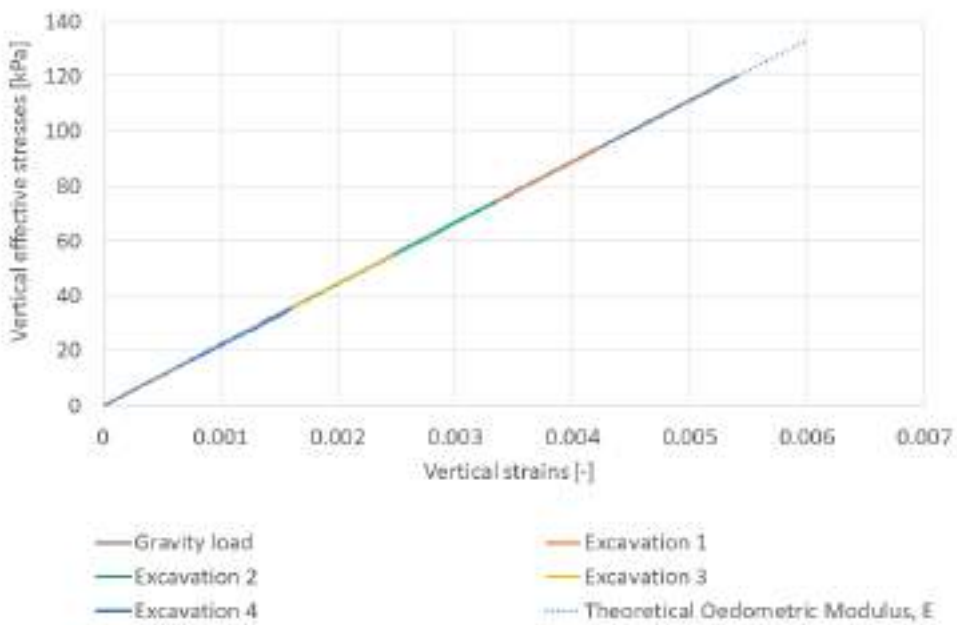


Figure 10.13: *Stress–strain curve for linear elastic material.*

The stress decrease in time is shown in Figure 10.14.

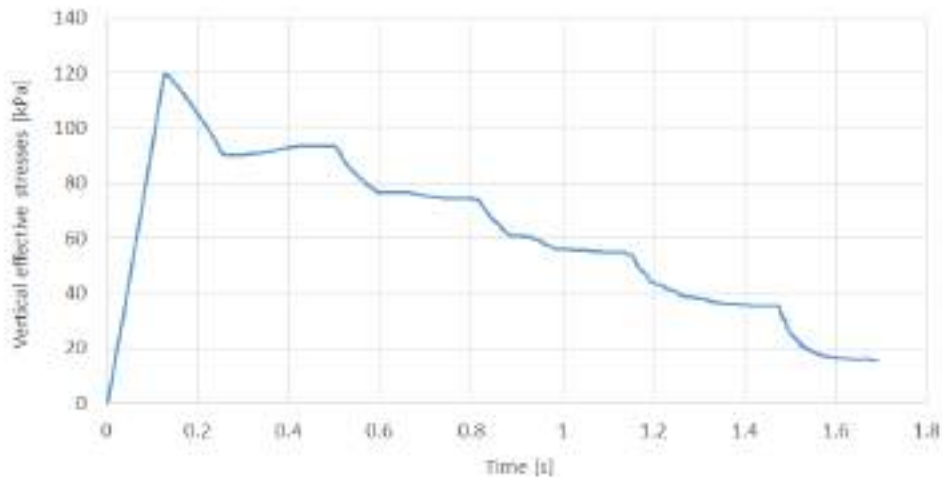


Figure 10.14: *Stress–time curve for linear elastic material*

10.1.6 Excavation with Mohr-Coulomb material

In this section, a similar analysis is carried out using the same geometry and same boundary conditions. Now, the Mohr-Coulomb model is adopted instead of the linear elastic model. Enter the soil parameters according to Table 10.2 and as shown in Figure 10.15.

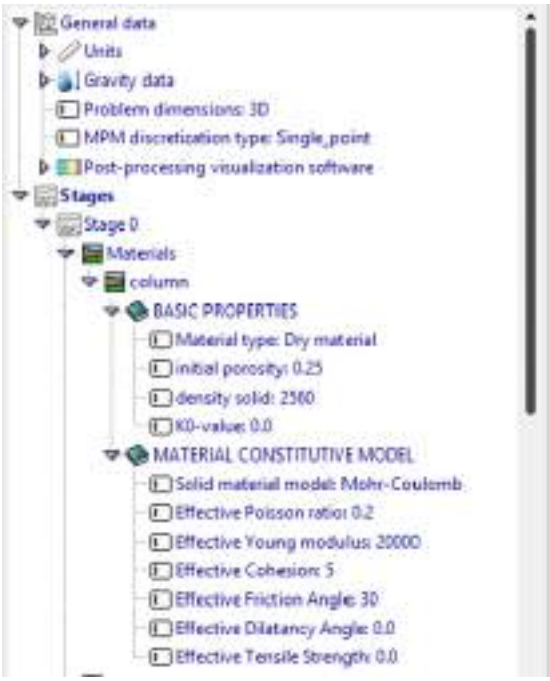


Figure 10.15: *Defining material properties for the column excavation problem, Mohr-Coulomb model*

The results for the dry soil column with Mohr-Coulomb constitutive model are presented for the end of the gravity loading and the end of each excavation stage in Figures 10.16 and 10.17.

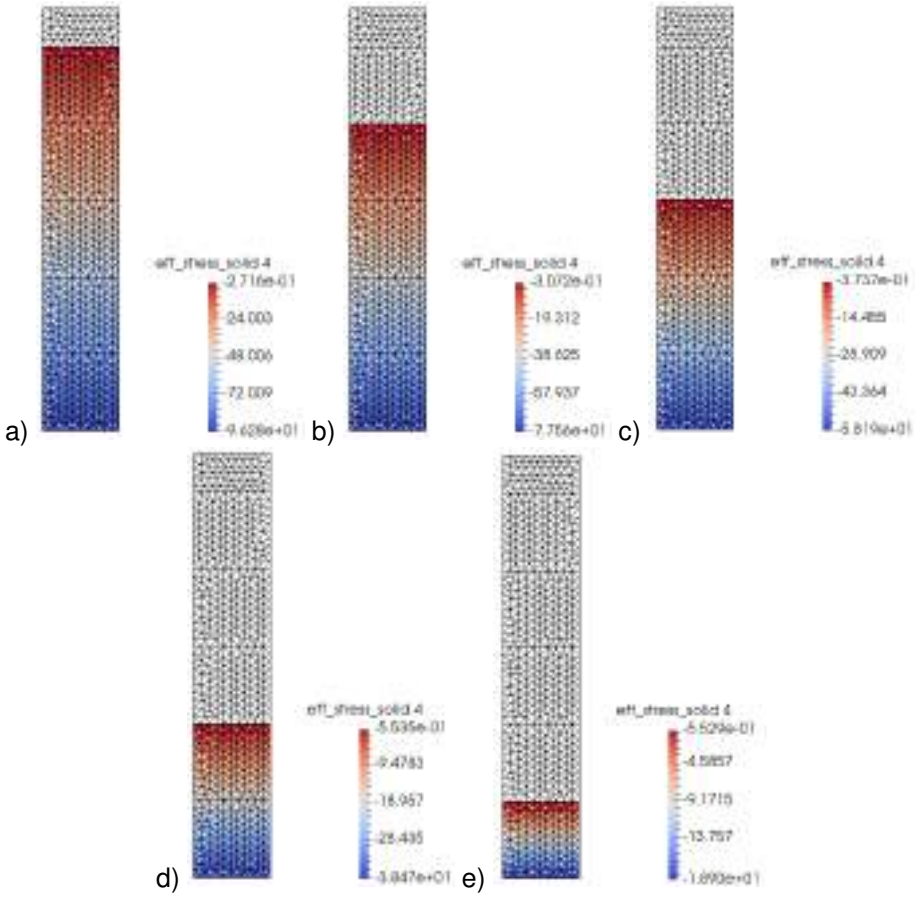


Figure 10.16: *Vertical effective stresses σ_{yy} in [kPa]: a) after gravity loading, b) after excavating volume 5, c) after excavating volume 4, d) after excavating volume 3, e) after excavating volume 2.*

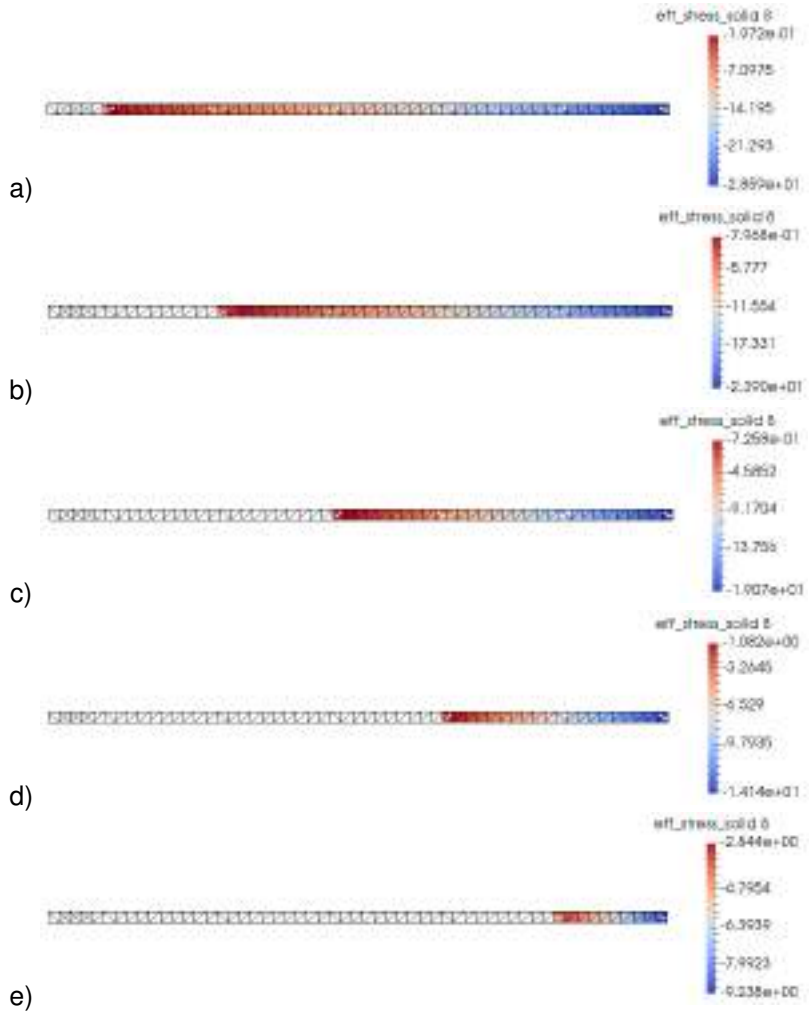


Figure 10.17: *Horizontal effective stresses σ_{zz} in [kPa]: a) after gravity loading, b) after excavating volume 5, c) after excavating volume 4, d) after excavating volume 3, e) after excavating volume 2.*

Figure 10.18 presents the stress–stain curve for the Mohr-Coulomb model. The theoretical oedometric modulus is also plotted. The yield criteria is reached and plastic deformations are accumulated.

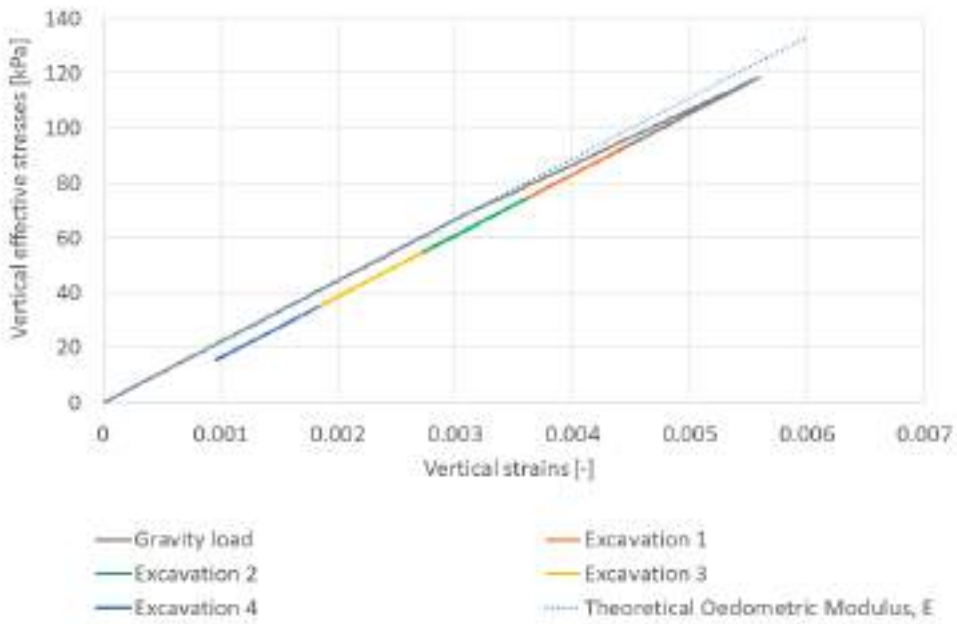


Figure 10.18: *Stress–strain curve for Mohr-Coulomb material.*

The stress decrease in time is shown in Figure 10.19.

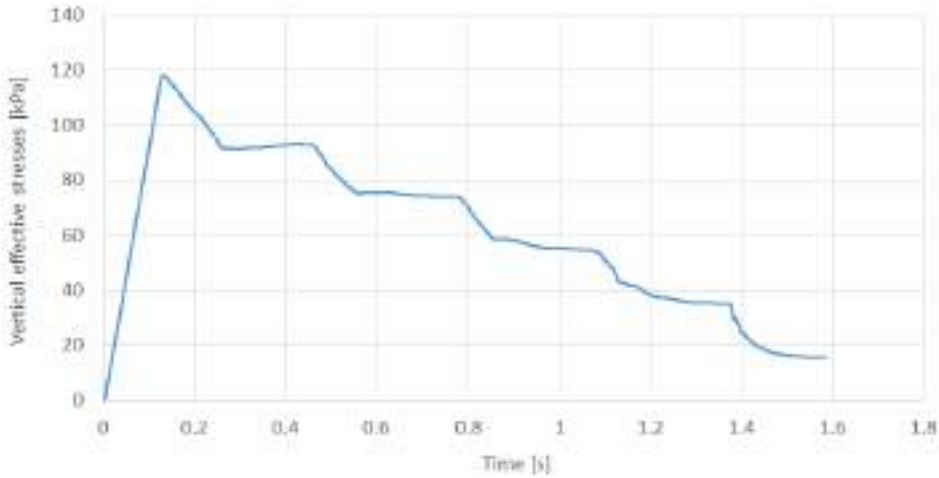


Figure 10.19: *Stress–time curve for Mohr-Coulomb material.*

10.2 2D Strutted excavation

10.2.1 Problem definition

The problem is simulated in 2D-plane strain conditions. The geometry of the problem after the excavation process is given in Figure 10.20. The dimensions of the model are given in Table 10.3. The analysis is conducted in undrained total stresses.

The calculation will be performed in two stages:
(1) stress initialisation with quasi-static gravity loading
(2) excavation process by consecutively removing soil surfaces of 1m height.

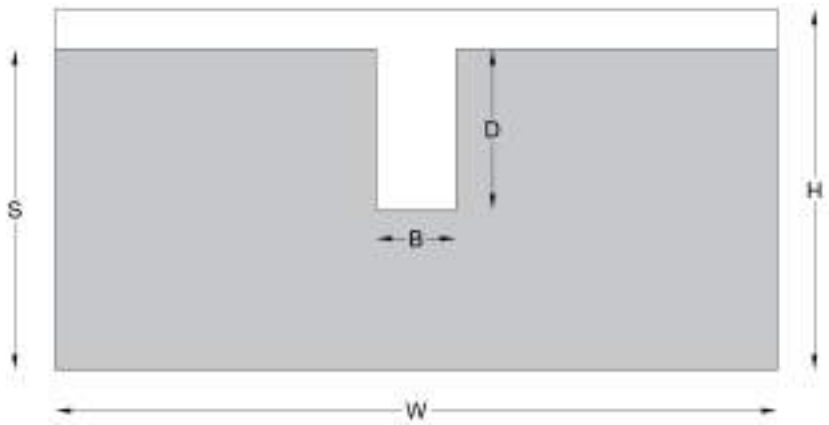


Figure 10.20: *Definition of strutted excavation problem.*

Table 10.3: *Dimensions of strutted excavation.*

D/B ratio	Box height H [m]	Box width W [m]	Soil height S [m]	Excavation width B [m]	Excavation depth D [m]
2:1	4.5	9	4	1	2

10.2.2 Creating input data

The geometry, boundary conditions, material properties and mesh are defined using GiD pre-processor. The calculation options are defined in the CPS file. Both follow the same procedure as described in Section 3.2. Only variations to previous examples are highlighted in the following sections.

10.2.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. “struttedexcavation").

10.2.2.2 Defining geometry

Create the geometry following the steps as described in Section 3.2.2 with dimensions as given in Table 10.3. As a reference the key coordinates indicated in Figure 10.21 and Table 10.4 can be used. As in the previous example, an entity containing empty elements should

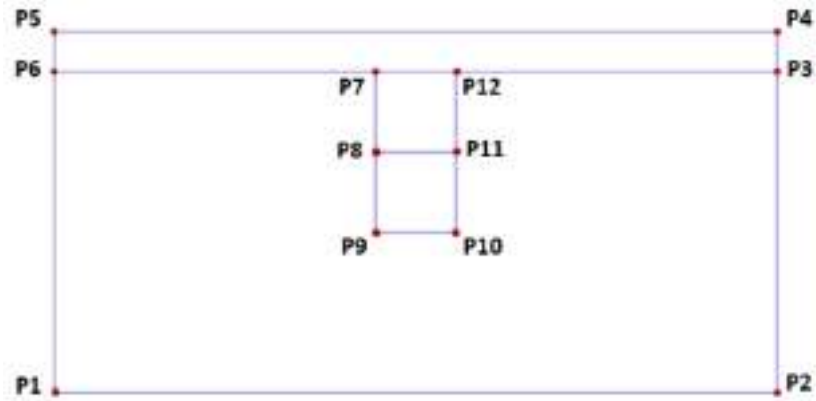


Figure 10.21: *Geometry of strutted excavation problem. Key points are indicated as given in Table 10.4*

Table 10.4: *Key coordinates of the strutted excavation problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	0	0
P2	9	0
P3	9	4
P4	9	4.5
P5	0	4.5
P6	0	4
P7	4	4
P8	4	3
P9	4	2
P10	5	2
P11	5	3
P12	5	4

be defined at the top side of the soil model to allow for free movement of material points. The geometry should look like shown in Figure 10.22. An index is automatically assigned to each surfaces in GiD. This index cannot be changed and is needed for the excavation process.

10.2.2.3 Defining material properties

Define the material properties following the steps as described in Section 3.2.5. In this example, the soil is assumed to be a purely cohesive material and modelled with a Tresca [8] failure criterion; that is a Mohr-Coulomb failure criterion in which friction (φ) and dilatancy (ψ) angles are 0 and the cohesion is equal to the shear strength. Enter the material parameters according to Table 10.5 as shown in Figure 10.23. Finally, assign the material to the soil surfaces.

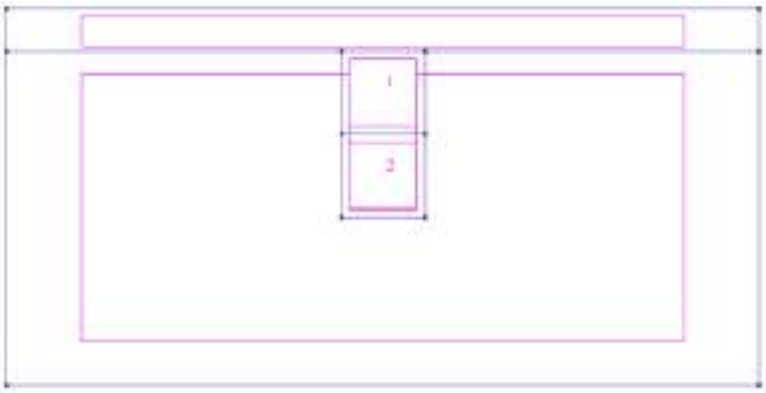


Figure 10.22: *Geometry of strutted excavation problem including labelling of excavated surfaces.*

Table 10.5: *Material parameters of the soil.*

Material	Soil
material type	saturated material-undrained total stress
initial porosity (-)	0.2
density solid (kg/m^3)	2650
density liquid (kg/m^3)	1000
K0-value	0
material model solid	Mohr-Coulomb
Poisson ratio (-)	0.49
Young modulus (kPa)	10000
cohesion (kPa)	5.5
friction angle(°)	0
dilatancy angle(°)	0

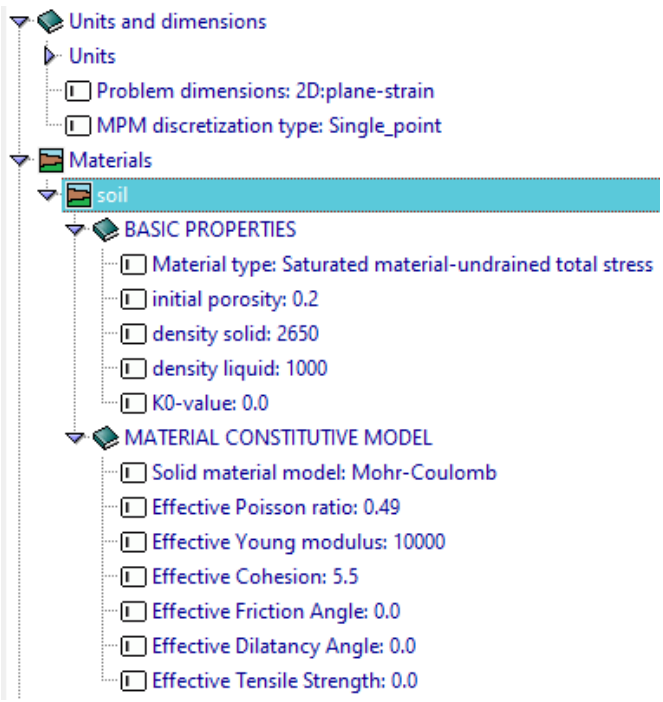


Figure 10.23: *Defining material properties for the strutted excavation problem, Mohr-Coulomb model.*

10.2.2.4 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. Choose 3 material points per element. Check if the material and the material points are assigned correctly (Figure 10.24).

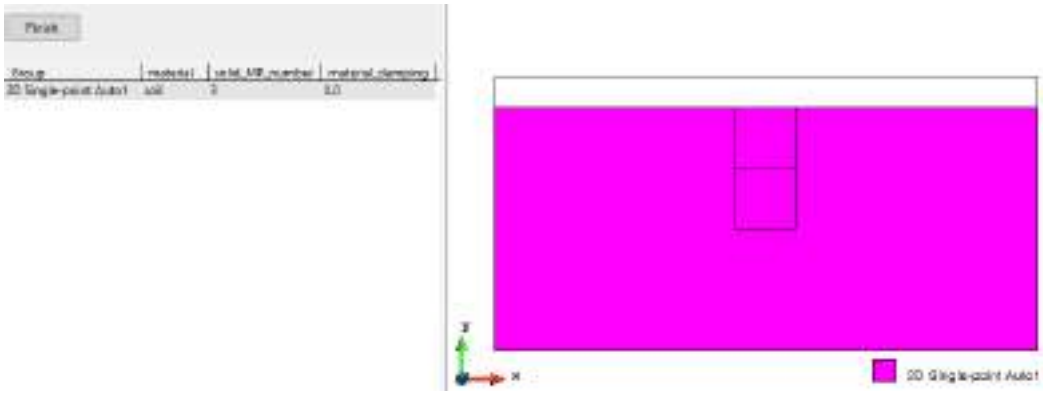


Figure 10.24: *Specification of the material and the number of material points per element for the strutted excavation problem.*

10.2.2.5 Defining boundary conditions

Define the boundary conditions for solid phase following the steps given below. Note that the hypothesis of strutted excavation is ensured by constraining horizontal displacements at the vertical contours of the excavation.

(1) Left side at x=0.0: horizontal fixity in x-direction
(2) Right side at x=9.0: horizontal fixity in x-direction
(3) Top side at y=4.5: vertical fixity in y-direction
(4) Bottom side at y=0: full fixities in x- and y- directions
(5) Left side at x=4.0 and right side at x=5.0 of excavation: horizontal fixity in x-direction

The assigned fixities of the column excavation problem should look like shown in Figure 10.25.

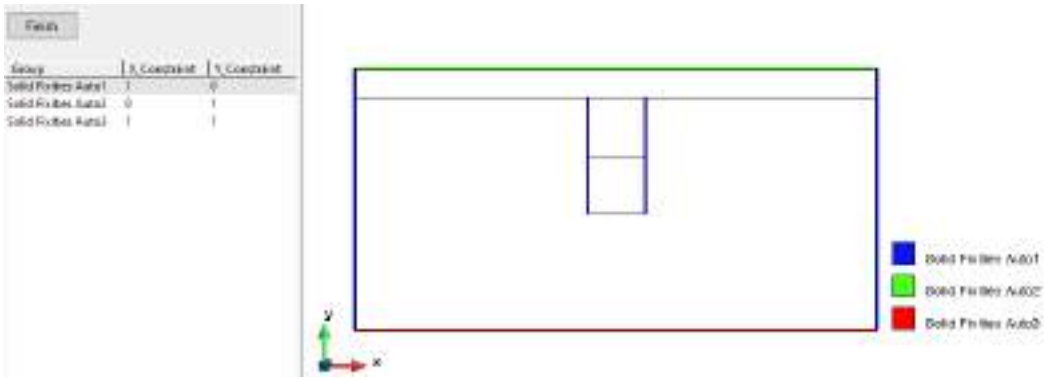


Figure 10.25: *Fixities of the strutted excavation problem in GiD.*

10.2.2.6 Creating mesh

For this example, an unstructured mesh with an element size of 0.25m is used. To generate the mesh follow the steps in Section 3.2.8.

10.2.2.7 Defining excavation steps

The first 5 load steps would consist of applying gravity load. Then the excavation process is simulated in two stages by removing the material points belonging to volumes of 1 m height. They are removed immediately at the beginning of each excavation stage and then enough time is left in order to reach the equilibrium.

The sixth load step would include the excavation of the first volume of soil, which is the volume no. 1 according to the labelling in Figure 10.22. Therefore, the first step of the excavation sequence of volume no. 1 should be set equal to 6.

The two approaches proposed differ just for the definition of the excavation steps.

Approach 1

In the first approach no soil is allowed to move into the excavation and the MPs are continuously removed as they rise inside it. The last step in the excavation sequence for volume no. 1 should be set equal to 75 (which corresponds to the final load step). The definition of the excavation sequence for volume no. 1 is shown in Figure 10.26. The next volume to be excavated is volume no. 2. Since the excavation of volume no. 1 is performed in 10 calculation steps, the first step of the excavation sequence of volume no. 2 is set equal to 16 whereas the last step would be 75. The final setup of the excavation process should look like shown in Figure 10.27(a).

Approach 2

In the second approach, once the excavation of volumes no. 1 and 2 ends, soil is allowed to move and accumulate into the excavated zone. Since the second excavation is carried out in the load step 16, the last step in the excavation sequence for volume no. 1 and 2 should be set equal to 16. The final setup of the excavation process should look like shown in Figure 10.27(b).

Once defined the excavation steps, the mesh should be re-generated.

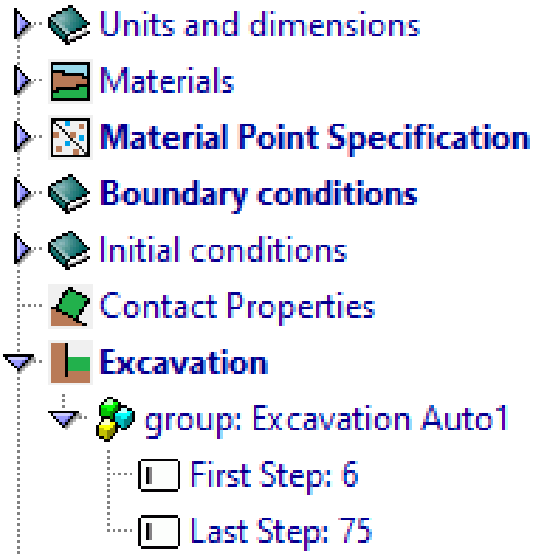


Figure 10.26: *Definition of the first excavation step for Approach 1.*

(a)

(b)

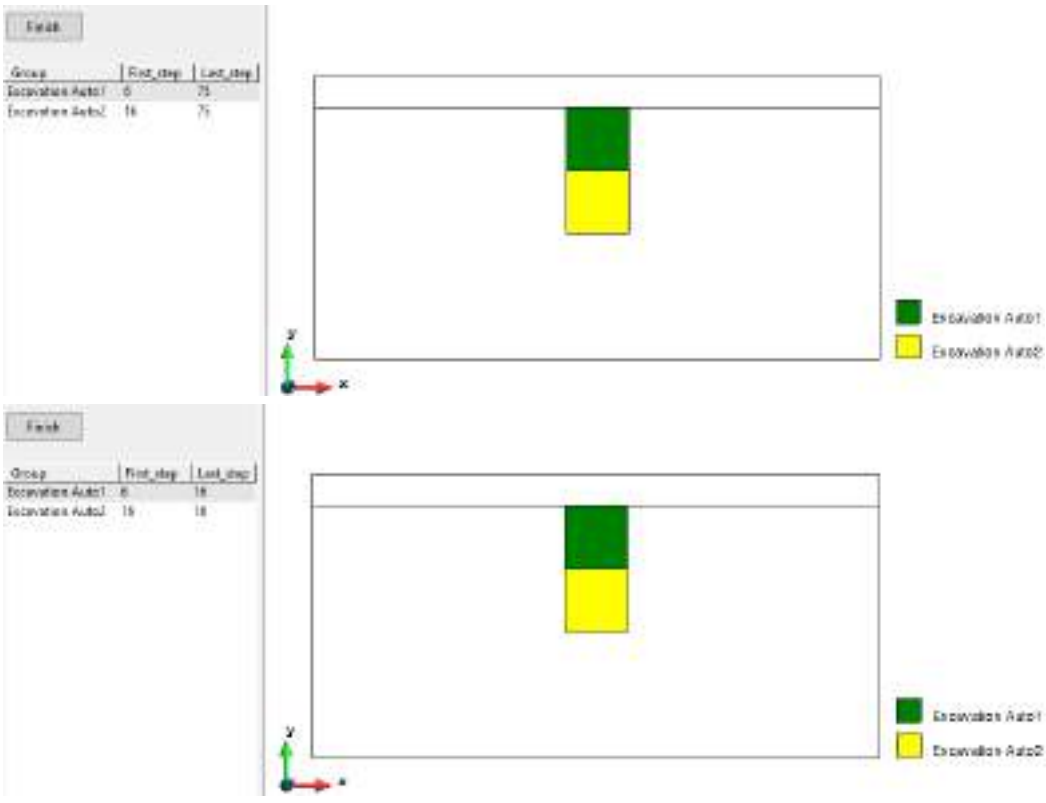


Figure 10.27: *Assigned excavation steps: (a) Approach 1, (b) Approach 2.*

10.2.2.8 Initialisation of calculation data

In the 2025 version, there is no need to separately define data in CPS files. Instead, all settings can be defined in GiD across different stages.

10.2.2.9 Stage0: Gravity loading

Only the calculation parameters required to initialize the first load step calculation are added in stage0. The calculation parameters for subsequent load steps will be adjusted in stage1.

Computational method
The computational method used in this example is the "MPM - mixed integration".

Calculation step data
The initial stage corresponding to gravity loading is carried out in five calculation steps. The end of each calculation step is determined by the convergence criteria. A Courant number of 0.98 is advised to ensure the stability of the solution.

Load multipliers
The gravity load is applied in five steps. Therefore set the multipliers for gravity loading as 0.0 (initial) and 1.0 (final value).

Quasi-static convergence
The determination of the end of each calculation step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition.

Homogeneous local damping
In order to converge to the quasi-static solution an overall local damping coefficient for all active elements of 0.75 is applied.

Strain smoothing
The strain smoothing algorithm will be used to minimise kinematic locking, which might occur for volumetric strains.

To define the Stage0 calculation parameters follow the instructions given in Section 3.2.9. For this example, enter the values as given below:

- Set "COMPUTATION METHOD" as "MPM - mixed integration" from the drop down menu
- Set "number of calculation steps [-]" as "5"
- Set "time per calculation step [s]" as "0.5". Note that any value except 0.0 can be entered since, in the first stage, the quasi-static convergence criterion is used
- Set "GRAVITY LOADING" as "apply gravity load - stepwise" from the drop down menu
- Set "gravity multiplier initial [-]" to "0.0"
- Set "gravity multiplier final [-]" to "1.0"
- Set "QUASI-STATIC CONVERGENCE" as "apply convergence criteria"
- Set "HOMOGENEOUS LOCAL DAMPING" as "apply homogeneous local damping" from the drop down menu
- Set "Local damping coefficient" to "0.75"
- Set "STRAIN SMOOTHING" as "apply strain smoothing" from the drop down menu

All other calculation parameters can be kept at their default values.

10.2.2.10 Stage1: Excavation

Copy Stage0 to retain all settings, but increase the number of load steps to 75 to include the two excavations and a final stage.The time per load step is set as 0.15 seconds. Therefore set:

- Set "number of calculation steps [-]" as "75"
- Set "time per calculation step [s]" as "0.15".
- Set "QUASI-STATIC CONVERGENCE" as " not apply convergence criteria"
- Set "HOMOGENEOUS LOCAL DAMPING" as "apply homogeneous local damping" from the drop down menu
- Set "Local damping coefficient" to "0.05"

10.2.3 Performing calculation in Anura3D

Once the geometry, material properties, boundary conditions, and calculation parameters are defined, and the mesh is generated. Click "Calculate" on the top bar in Anura3D. Input errors and calculation errors are displayed in the window as well.

10.2.4 Visualisation of results in ParaView

The results for the strutted excavation problem are presented for the end of the gravity loading stage and for the end of the simulation. Vertical effective stresses in y-direction at the end of gravity loading are shown in Figure 10.28.

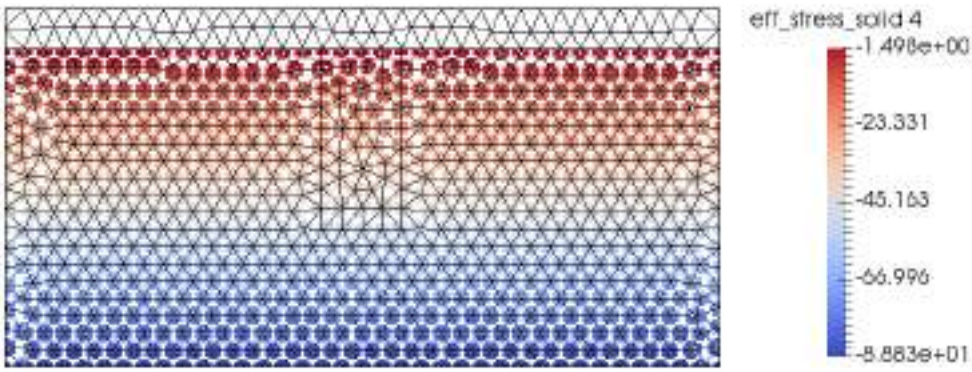


Figure 10.28: *Vertical effective stresses σ_{yy} [kPa] after gravity loading.*

Note: When the material is defined as "saturated material-undrained total stress", the effective stresses that are visualised in ParaView are equal to the total stresses.

Approach 1

Total displacements in x- and y-directions at the end of the simulation for Approach 1 are shown in Figure 10.29.

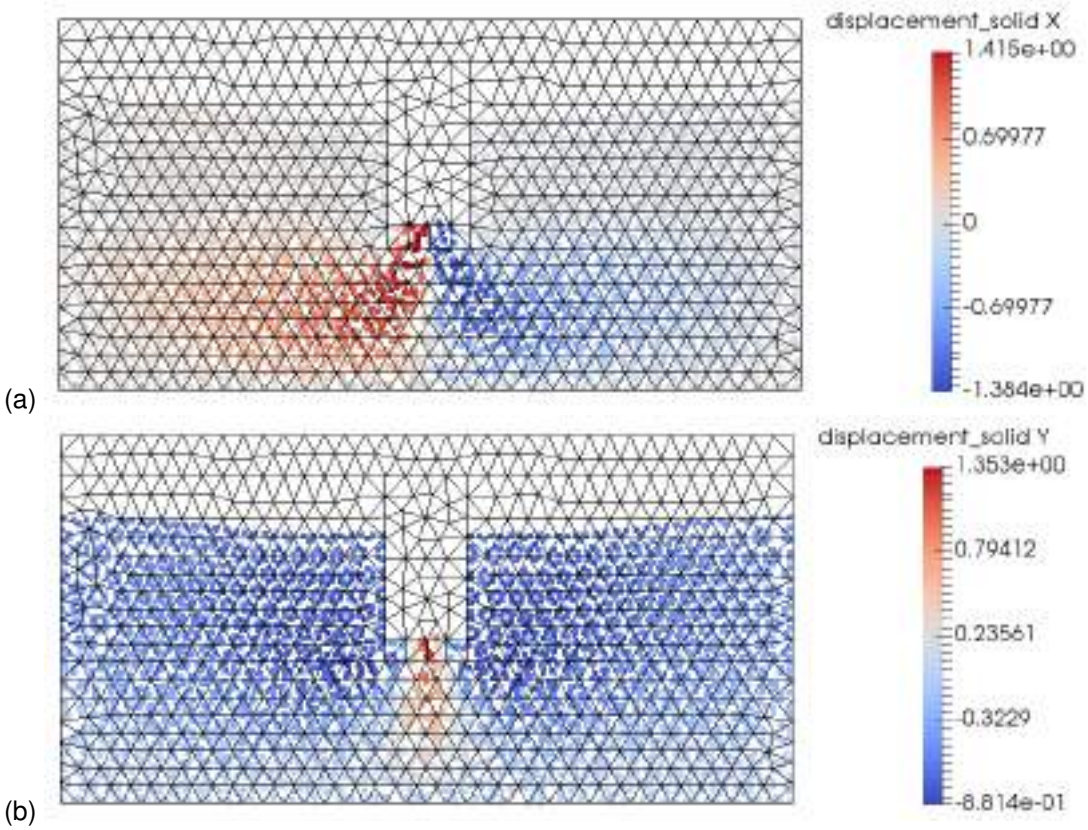


Figure 10.29: *(a) Horizontal displacements [m], (b) Vertical displacements [m] (Approach 1).*

Approach 2

Total displacements in x- and y-directions at the end of the simulation for Approach 2 are shown in Figure 10.30.

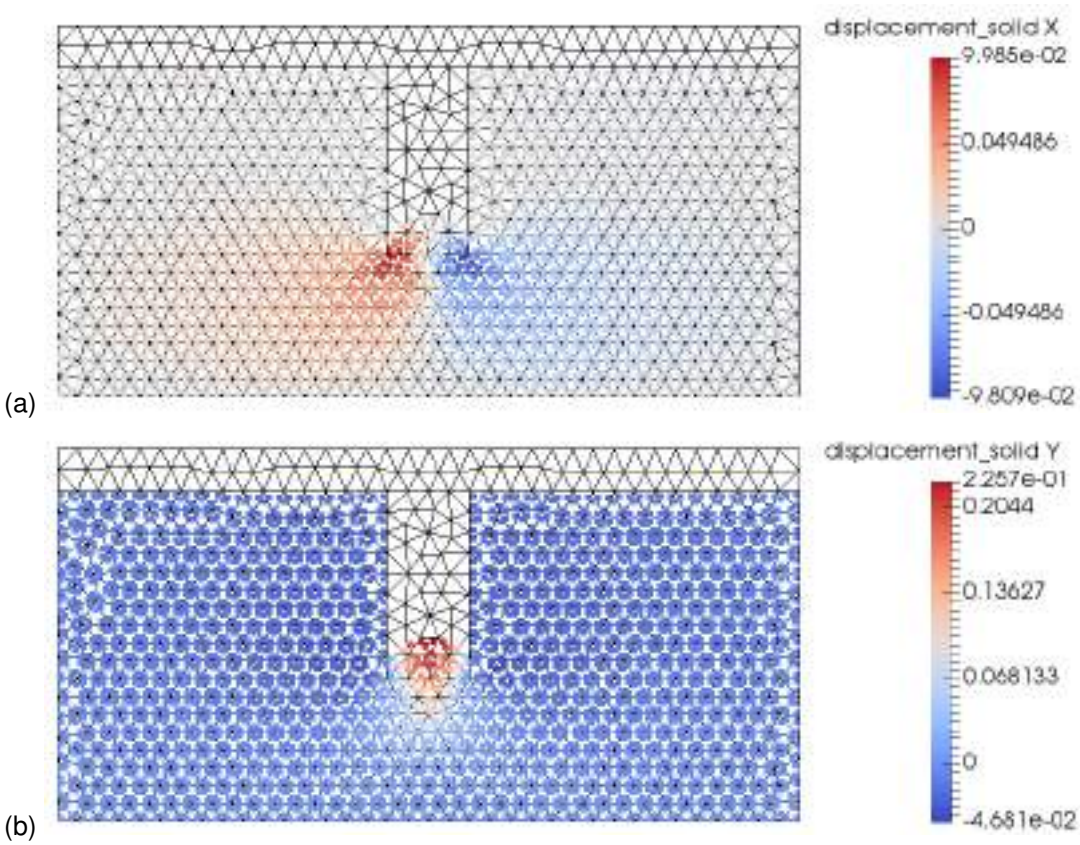


Figure 10.30: *(a) Horizontal displacements [m], (b) Vertical displacements [m] (Approach 2).*

The results of the simulation carried out with Approach 2, in which the material is allowed to accumulate into the excavation, show that total displacements are significantly lower compared with Approach 1. This behavior can be explained considering that the accumulation of soil changes the geometry of the problem, leading earlier to a new stable configuration.

More details about the MPM simulation of strutted excavation in clay and its comparison with solutions reported in the literature, can be found in Chapter 15 of "The Material Point Method for Geotechnical Engineering: A Practical Guide" by Fern et al. [1]

11 Construction

Authors: Luís Ángel Avilés; Gaia Di Carluccio; Mateu Maglia; Núria Pinyol

In the following tutorial, an example of the construction feature implemented in Anura3D is described. A 2D soil column under oedometric conditions is constructed in five layers. The column is modelled as linear elastic, and two variations are proposed considering different values of Young's modulus. In the first example a Young's modulus high enough to produce small deformation during the construction process is considered. In the second example, a lower Young's modulus is assigned, which would cause more significant deformations during the construction and prevent the desired final geometry from being achieved. In this latter case, the option to fill the elements that become empty due to deformations during the construction stages with additional material points is activated.

11.1 2D Column construction

11.1.1 Problem definition

The dimensions of the soil column are given in Figure 11.1. Five equal layers of soil are considered and an empty space is added on top of the column to allow material points to move freely. The dimensions are given in Table 11.1. Note that each construction layer depth is H/5.

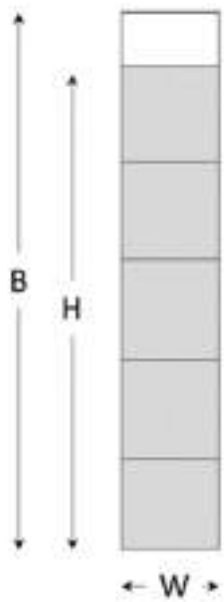


Figure 11.1: *Definition of the construction problem including empty space at the top.*

Table 11.1: *Dimensions of the column for the construction problem.*

Column height H(m)	Column width W(m)	Box Height B(m)
5	1	5.5

Table 11.2: *Material parameters of the soil for the construction problem.*

Material	Column
material type	dry material
initial porosity	0.35
density solid (kg/m^3)	2650
K0-value	0.0*
material model solid	Linear Elasticity
Young modulus (kPa)	2000
Poisson ratio (-)	0.25

(*) not used

11.1.2 Creating input data

The geometry, boundary conditions, material properties and mesh are defined using GiD pre-processor. Calculation options are defined in the *CPS* file. Both follow the same procedure as described in Section 3.2.

11.1.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "columnconstruction").

11.1.2.2 Defining geometry

Create the geometry following the steps as described in Section 3.2.2 with dimensions as given in Table 11.1 of the problem description. Note that an entity containing empty elements should be defined at the top of the column to allow for free movement of the material points. The geometry is created by defining 6 rectangles which will be discretised by a structured mesh. The geometry should look like that shown in Figure 11.2.

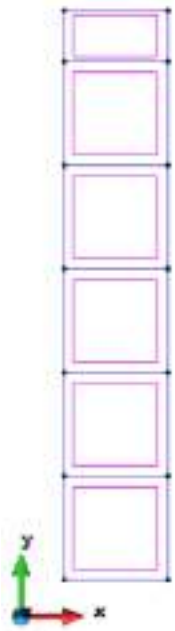


Figure 11.2: *Geometry of the column construction problem including empty entity at the top.*

11.1.2.3 Defining material properties

Define the material properties following the steps as described in Section 3.2.5. In this example, a linear elastic model is considered. Enter the material parameters according to Table 11.2 as shown in Figure 11.3.

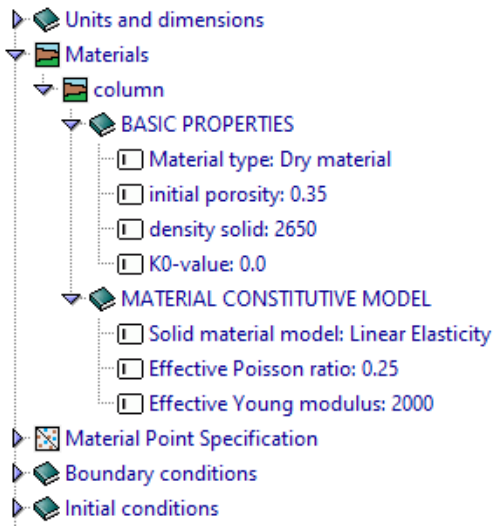


Figure 11.3: *Defining material properties for the column construction problem.*

11.1.2.4 Specifying material points

Specify the initial material point distribution following instructions in Section 3.2.6. Choose 3 material points per element for the column. Check if the material and the number of material points are assigned correctly (Figure 11.4).

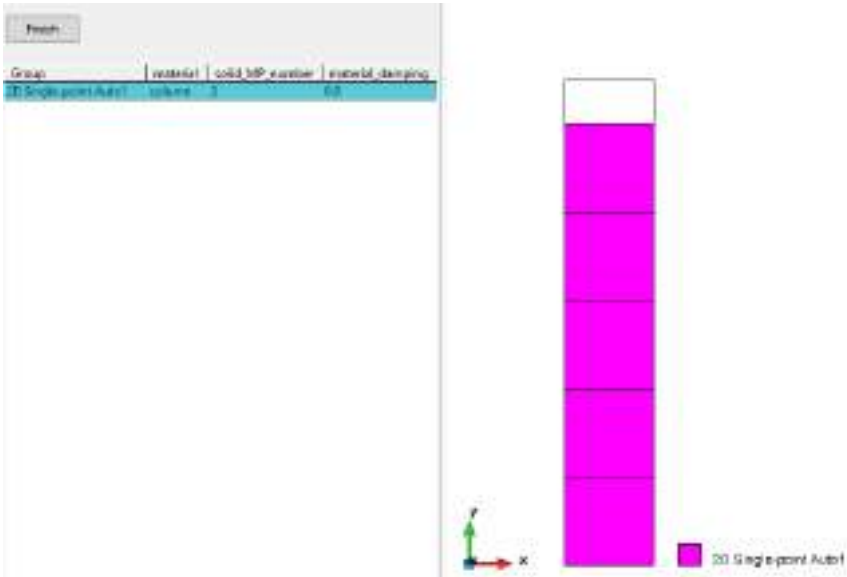


Figure 11.4: *Specification of the material and the number of material points per element for the column construction problem.*

11.1.2.5 Defining boundary conditions

Define the boundary conditions following the steps given below:

- ◇ Left side at x=0.0: horizontal fixity in x-direction on the 6 lines
- ◇ Right side at x=1.0: horizontal fixity in x-direction on the 6 lines
- ◇ Top side at y=5.5: vertical fixity in y-direction
- ◇ Bottom side at y=0: fixities in x- and y- directions

The assigned fixities of the column excavation problem should look like that shown in Figure 11.5.

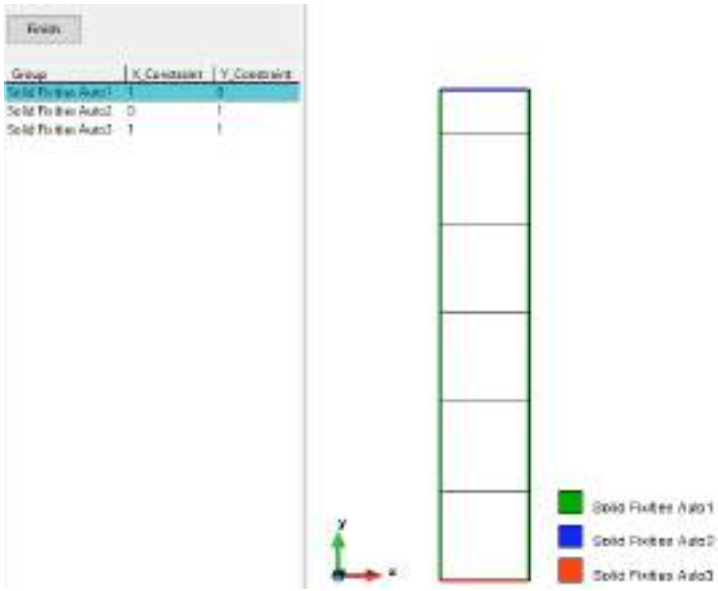


Figure 11.5: *Solid fixities of the column construction problem.*

11.1.2.6 Defining construction steps

Anura3D provides a construction tool which allows to simulate soil construction in stages by defining the range of load steps in which the material points have to be constructed in a certain area. In this example, the construction process is simulated in five stages by adding the material points to surfaces of 1 m height. Each construction stage is performed in two load steps. The material points are added during the first load step and then enough time is left in order to reach the equilibrium.

Therefore, in order to define the construction of the lowest layer, set the first step of the construction sequence as 1 and last step as 2. Then, assign the construction condition to the lowest surface. The following soil layers will be constructed sequentially from bottom to top. The final setup of the construction process and the correct assignment are shown in Figure 11.6.

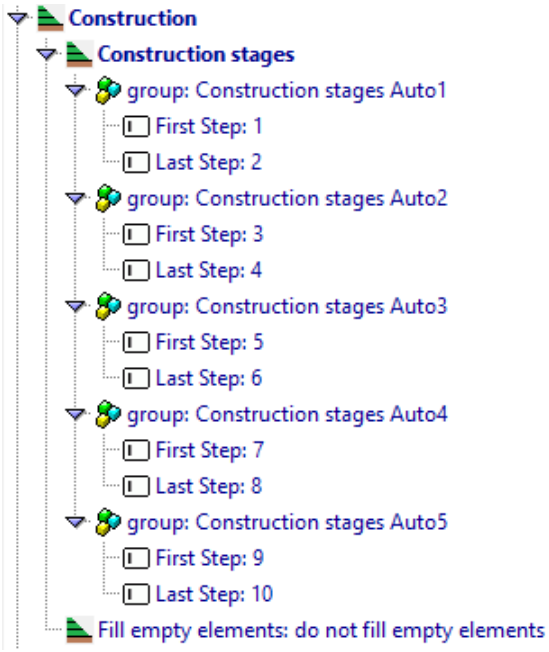


Figure 11.6: *Definition of construction stages.*

11.1.2.7 Creating mesh

For this example, a structured mesh is used with an element size of 0.1m. To generate the mesh follow the steps in Section 3.2.8.

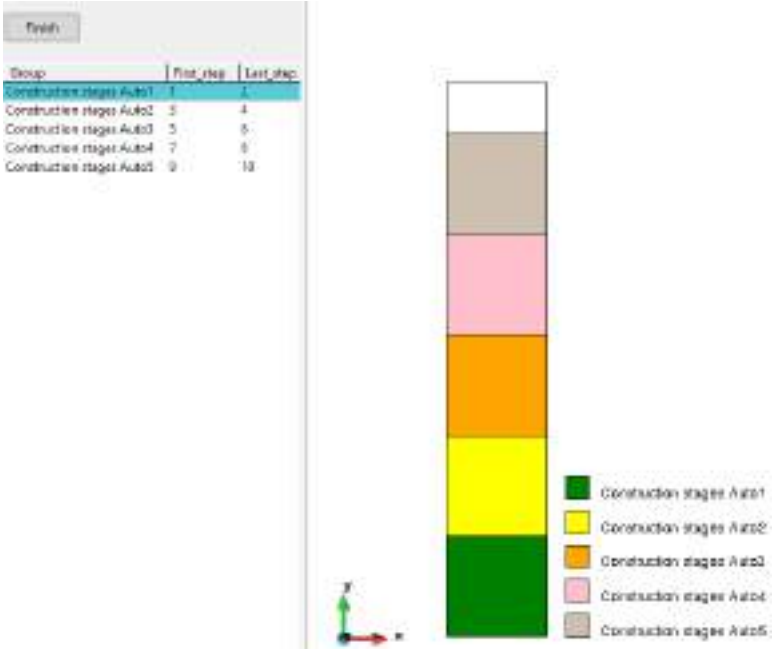


Figure 11.7: *Construction stages assignment to geometry.*

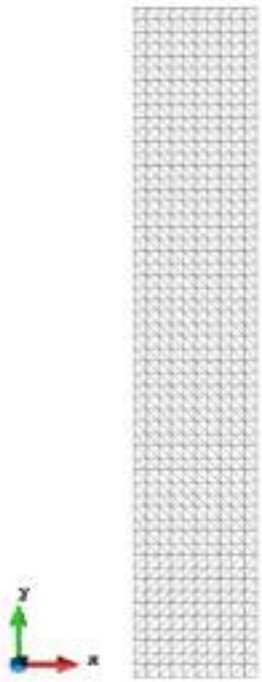


Figure 11.8: *Mesh of the column construction problem.*

11.1.2.8 Initialisation of calculation data

Computational method
The computational method used in this example is the "MPM - mixed integration".

Calculation step data
The computation will be carried out in 10 steps of 10s each.

Load multipliers - Gravity Load
The full gravity load is applied in the first load step of each construction stage. Therefore, select Applygravityload – step. Set the gravity multiplier initial and final to 1.0.

Homogeneous local damping
An overall local damping coefficient for all active elements of 0.05 is applied.

Output data
One material point located at the bottom of the column will be selected for which an additional output file will be created. Section 3.4.7 explains how to determine the ID number of a material point.

To define the initial calculation parameters follow the instructions given in Section 3.2.9. For this example, enter the values as given below:

- Select "COMPUTATION METHOD" as "MPM - mixed integration" from the drop down menu
- Set "number of calculation steps [-]" as "10"
- Set "time per calculation step [s]" as "10".
- Set "GRAVITY LOADING" as "apply gravity load - step" from the drop down menu
- Set "gravity multiplier initial [-]" to "1.0"
- Set "gravity multiplier final [-]" to "1.0"
- Set "HOMOGENEOUS LOCAL DAMPING" as "apply homogeneous local damping"
- Set "Local damping coefficient" to "0.05"
- Set "OUTPUT DATA" for "number of material points" to "1"
- Set "material point #1" to "35"

All other calculation parameters can be kept at their default values.

11.1.2.9 Generating input files

After the geometry, material properties, boundary conditions, construction steps, calculation parameters and the mesh are defined, the input files (i.e. *GOM* and CPS) have to be created as described in Section **??**.

The generated input files `columnconstruction.GOM` and `columnconstruction.CPS` can be found in the folder `columnconstruction.A3D` in the directory which was specified earlier as shown in Section 11.1.2.1.

11.1.3 Performing calculation in Anura3D

The Anura3D calculation can be started following instructions described in Section 3.3.

11.1.4 Visualisation of results in ParaView

The results for the dry soil column with linear elastic constitutive model are presented at the end of each construction stage. Vertical effective stresses are shown in Figure 11.9

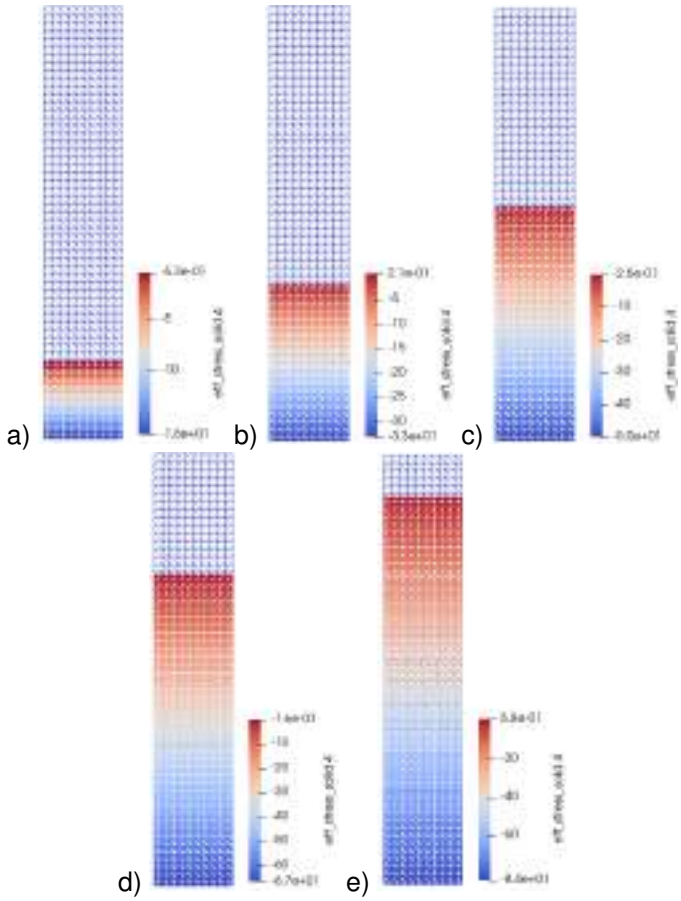


Figure 11.9: *Vertical effective stresses σ_{yy} in [kPa]: a) after construction of layer 1 , b) after construction of layer 2, c) after construction of layer 3, d) after construction of layer 4, e) after construction of layer 5.*

11.1.5 Visualisation of text output files

Following the instructions from Section 3.6, further results can be plotted for a material point located at the bottom of the column which is indicated in Figure 11.10.

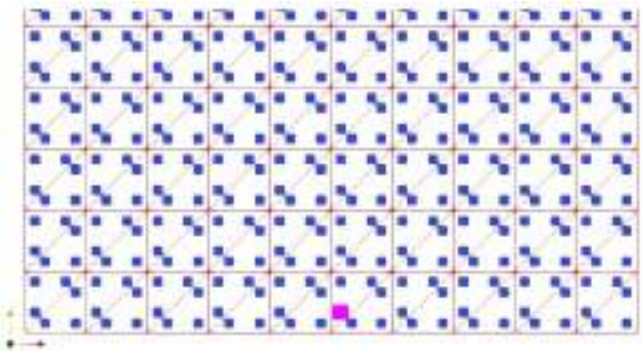


Figure 11.10: *Location considered for results which has a material point ID equal to 36.*

Figure 11.11 presents the stress–strain curve and shows that the behaviour is completely linear elastic when loading and unloading. The theoretical oedometric modulus is obtained using the following equation:

$$E^0 = \frac{1-\nu}{(1+\nu)(1-2\nu)}E \tag{11.1}$$

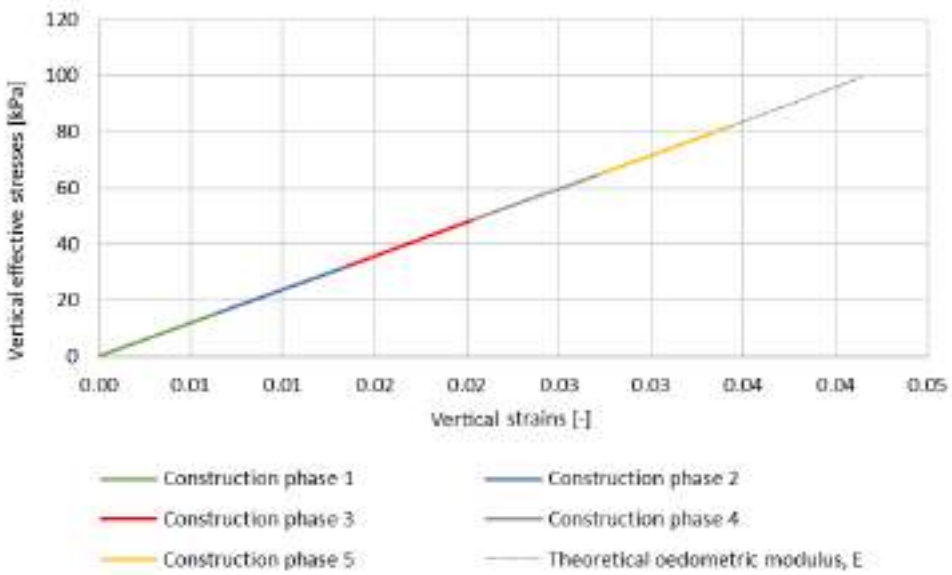


Figure 11.11: *Stress–strain curve for linear elastic material.*

11.1.6 Variation: filling empty elements during construction

A further simulation is performed using the same geometry and same boundary conditions but considering a lower value of Young's modulus equal to 500 kPa. Table 11.3 shows the new material properties of the constructed soil column. In this example, the elements that become empty due to deformation during the construction process are filled with additional material points to match the desired geometry. This option can be activated in GiD pre-processor as indicated in Figure 11.12. In order to apply this tool throughout the entire construction process, the last load step of each construction stage should be set equal to 10, which corresponds to the final load step of the simulation (Figure 11.12).

Table 11.3: *Material parameters of the soil for the construction problem with filling of empty elements.*

Material	Column
material type	dry material
initial porosity	0.35
density solid (kg/m^3)	2650
K0-value	0.5*
material model solid	Linear Elasticity
Young modulus (kPa)	500
Poisson ratio (-)	0.25

(*) not used

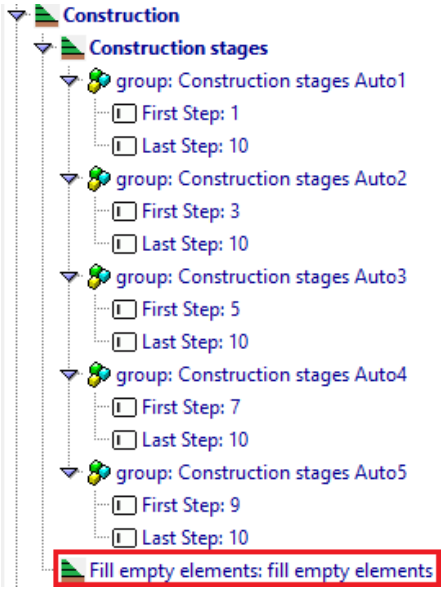


Figure 11.12: *Activate the option to fill empty elements.*

11.1.6.1 Visualisation of results in ParaView

Figure 11.13 show the results in terms of vertical effective stress distribution calculated at the end of the simulation. The results are also compared to those obtained without filling empty elements. The increment of weight due to the refill is observed in terms of vertical stresses.

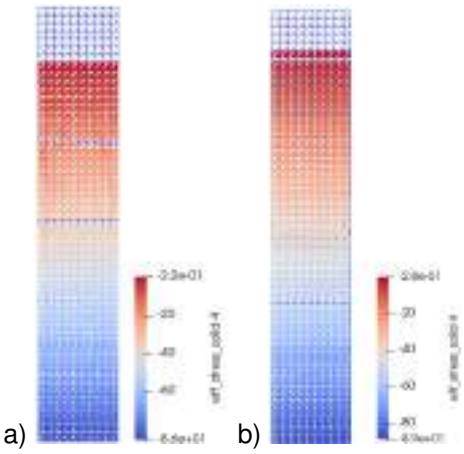


Figure 11.13: *Vertical effective stresses σ_{yy} in [kPa]: a) without refill , b) with refill*

12 Submerged slope collapse

Authors: Francesca Ceccato; Alba Yerro; Luis Angel Aviles; Mateu Maglia

This tutorial explains how to simulate the collapse of a submerged slope induced by an increase of pore pressure at the bottom in Anura3D. A small scale submerged slope is simulated in 2D-plane strain conditions applying the fully coupled two-phase formulation (see Chapter 1 of [1]).

The calculation is performed in three stages. In the first stage (Stage 0), stresses and pore water pressures are initialized by applying gravity loading. In the second stage (Stage 1) a liquid pressure is applied at the bottom of the slope, which decreases the effective stress, thus inducing the failure of the slope. Finally, in the last stage (Stage 2) the applied pressure is switched off and the slope stabilizes.

One particularity of this problem is that the *SUBMERGED CALCULATION* option is used, which means that the gravity load of the liquid phase is neglected and the submerged weight of the mixture is considered for the solid. As a result, the pore water pressures calculated in Anura3D are, in fact, the excess of pore pressures relative to the hydrostatic.

12.1 Problem definition

The geometry and the dimensions of the problem are given in Figure 12.1. The soil is modelled with the Mohr-Coulomb model with the properties given in Table 12.1.

Table 12.1: *Material parameters of the soil.*

material	soil
material type	saturated material - fully coupled
initial porosity [–]	0.45
density solid [kg/m^3]	2650
density liquid [kg/m^3]	1000
intrinsic permeability liquid [m^2/s]	5e-11
bulk modulus liquid [kPa]	45000
dynamic viscosity liquid [kPa $*$ s]	1e-6
material model solid	Mohr-Coulomb
effective Poisson ratio [–]	0.2
effective Young modulus [kPa]	5000
effective cohesion [kPa]	0
effective friction angle [$^\circ$]	32
dilatancy angle [$^\circ$]	0.0
tensile strength [kPa]	0.0

12.2 Creating input data

The geometry, discretization, material properties, and all other input parameters will be defined using the GiD pre-processor. This follows a similar procedure as described in Section 3.2. Here only the differences with respect to previous examples will be highlighted.

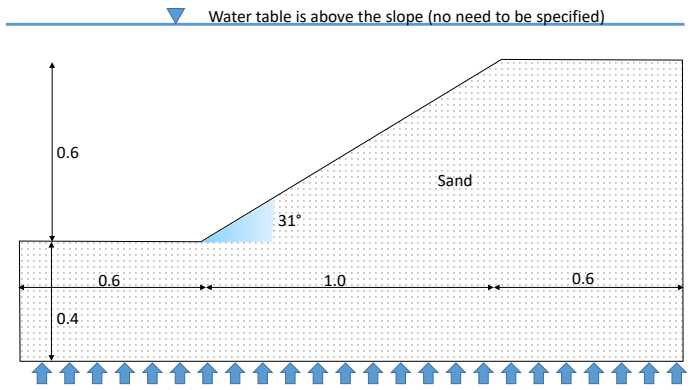


Figure 12.1: *Problem definition for failure of a submerged slope.*

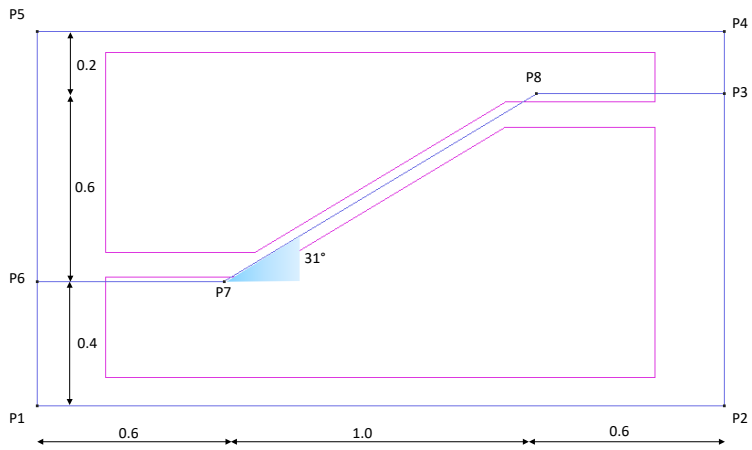


Figure 12.2: *Key points for definition of geometry od submerged slope.*

12.2.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Set the problemtype as described in Section 3.2.1 step (2). Save the project in the desired directory with an appropriate name (e.g. "SubmergedSlope").

12.2.2 Defining geometry

To generate the geometry follow the instructions given in Section 3.2.2. As a reference, the key coordinates indicated in Figure 12.2 and Table 12.2 can be used.

Table 12.2: *Key coordinates of the submerged slope problem.*

Point	X (m)	Y(m)
P1	0	0
P2	2.2	0
P3	2.2	1.0
P4	2.2	1.2
P5	0	1.2
P6	0	0.4
P7	0.6	0.4
P8	1.6	1.0

12.2.3 Defining general data

In this section, general data parameters common in all stages are defined. This are: units, gravity data, problem dimensions, MPM discretization and post-processing visualization. The non-default options are the problem dimension, which is set to "2D-plain strain", and the post-processing visualization. In this example, GiD post-processing visualization in Binary format is selected (12.3).

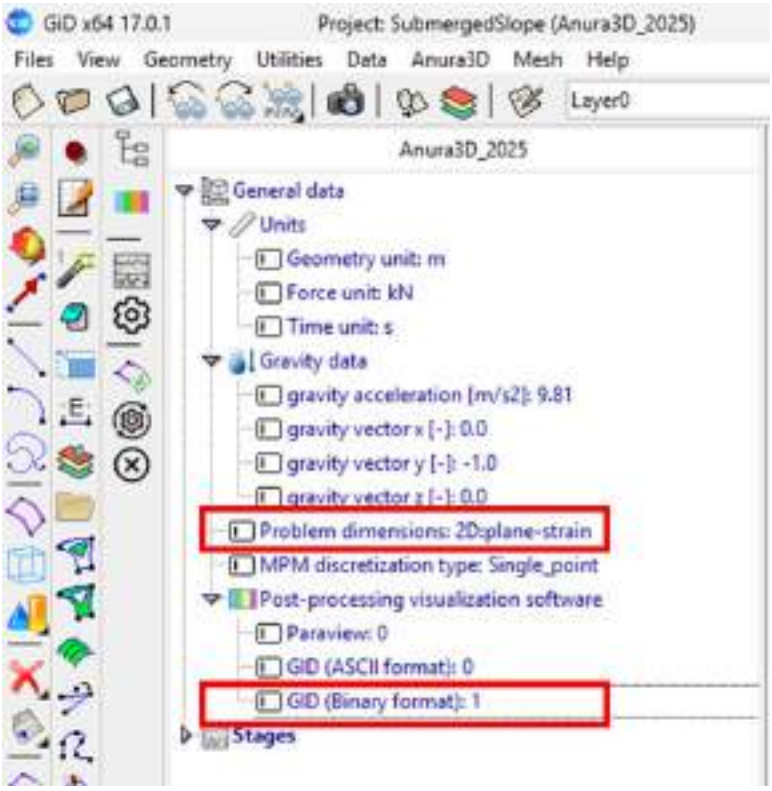


Figure 12.3: *General data for the submerged slope example.*

12.2.4 Defining calculation stages

As commented in the chapter introduction, this example consists of three stages. The first stage (Stage 0) initializes stresses and pore water pressure. In the second stage (Stage 1), an excess pore pressure is applied at the bottom of the model, inducing the slope failure. Finally, in the third stage (Stage 2), the excess pore pressure is deactivated, allowing the slope to stabilize and reach a new stable configuration.

12.2.4.1 Stage 0: Gravity loading

In this stage materials, material points and boundary conditions are defined to initialize stresses and pore pressures by applying a gravity loading.

12.2.4.1.1 Defining material properties

Define a new material with an appropriate name (e.g. "Sand") as explained in 3.2.5. Since we are going to perform a simulation in which solid-fluid interaction is of interest, the material type should be set to *saturated material - fully coupled*. Material parameters are given in Table 12.1. Note that the bulk modulus of the fluid is lower than the bulk modulus of the water, but it is significantly larger than the effective Young modulus in order to provide a realistic evolution of pore pressure without increasing the computational cost.

12.2.4.1.2 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. In this example choose 6 material points per element and no material viscous damping and assign it to the slope (Figure 12.4).

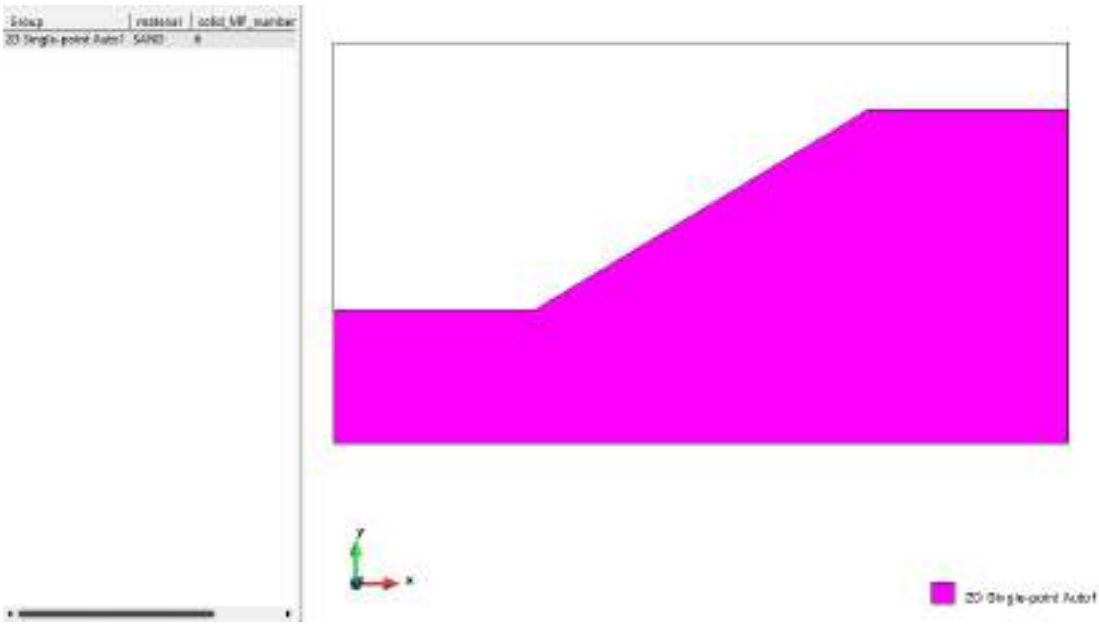


Figure 12.4: *Assignment of material for the submerged slope problem.*

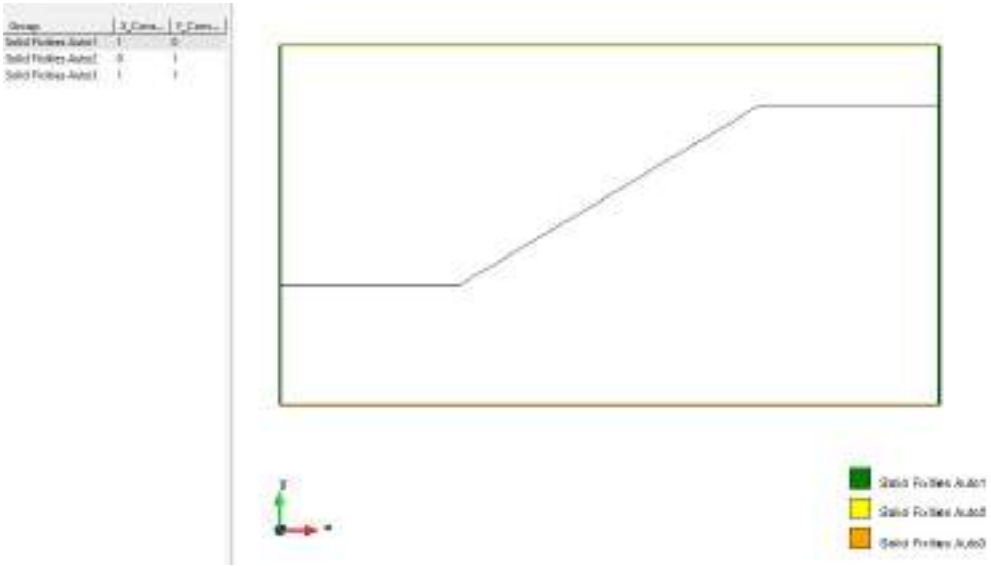


Figure 12.5: *Solid fixities of the submerged slope problem in GiD.*

12.2.4.1.3 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying loading fixities

Since in this example a fully coupled two-phase analysis is performed, solid and liquid fixities must be assigned. The solid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;
(4) Bottom side: full fixities in x- and y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 12.5;

The liquid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;
(4) Bottom side: vertical fixity in y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 12.6;

Notice that, liquid fixities at top and bottom side must be assigned separately to two distinct groups (*Liquid Fixities Auto 2* and *Liquid Fixities Auto 3* in Figure 12.6). This is necessary because, in the next stage (Stage 1), only the liquid fixities on the bottom side will be removed to apply the excess pore pressure.

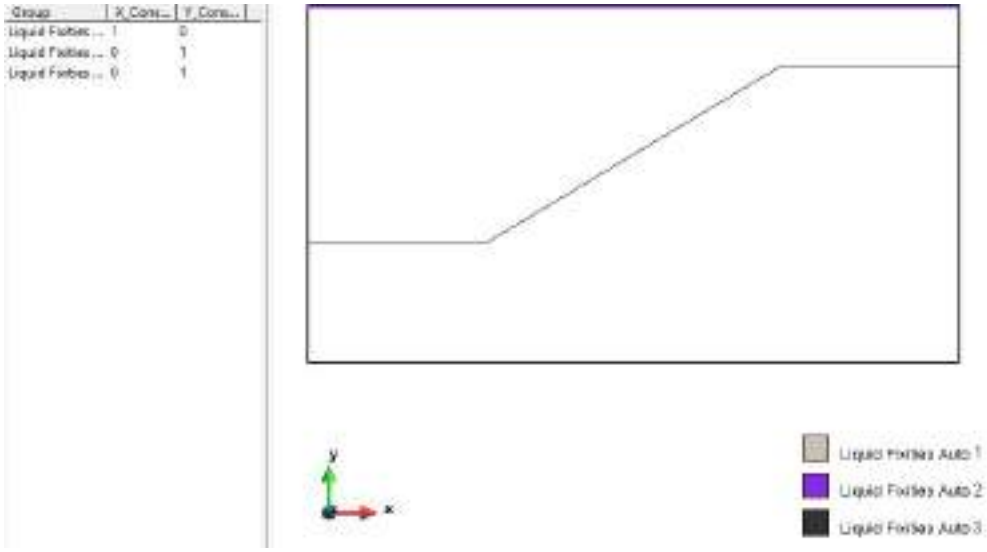


Figure 12.6: *Liquid fixities of the submerged slope problem in GiD.*

12.2.4.1.4 Defining calculation data

Figure **??** shows the calculation data of *Stage 0*.

Computation method
The option "MPM - mixed integration" should be selected.

Calculation step data
Gravity loading is performed in one calculation step. The end of the step is determined by the convergence criteria. A Courant number of 0.98 is advised to ensure the stability of the solution. Set "number of calculation steps [-]" to "1".

Gravity load
Gravity is applied from the beginning, therefore choose *Apply gravity load - stepwise* and set the multipliers for gravity loading as 1.0 (initial and final value). The gravity multipliers will be maintained constant throughout the calculation.

Quasi-static convergence
The end of the first load step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition. Set the QUASI-STATIC CONVERGENCE to "apply convergence criteria". The convergence criteria are the tolerated error of the kinetic energy, the tolerated error of the out-of-balance force, or the maximum number of time steps. In general, to obtain an accurate solution it is suggested to set the kinetic error and the force error to 0.01; however, in this example, in order to reduce the computational time, use a kinetic (energy) error and a force error of 0.04 for both the liquid and solid. Keep the default value of 10000 for the number of maximum time steps.

Homogeneous local damping
In order to damp oscillations to reach quasi-static convergence a local damping coefficient for all active elements of 0.7 is applied.

Strain smoothing
The strain smoothing algorithm will be switch on to minimise the kinematic locking.

Submerged Calculation

The slope is submerged, thus choose *apply submerged calculation*. The box *number of initialisation steps* will appear. In this example stresses are initialised in one calculation step, thus enter '1' in the input field. This option is useful to accelerate the convergence to quasi-static equilibrium by keeping the water pressure at zero and omitting the calculation of drag forces during the initialization step.

Default values

All other calculation options are not used in this example and the provided default values can be kept.

Figure 12.7: *Dialogue box to define calculation parameters in GiD for Stage 0 of the submerged slope problem.*

12.2.4.2 Stage 1: Slope failure

To create *Stage 1* by copying all information from *Stage 0*, right-click on *Stage 0* and select *copy (with assigned groups)* (Figure 12.8). In this stage a liquid pressure is applied at the bottom of the model to induce the slope failure. Thus, The only changes required with respect to the previous stage are the removal of the liquid fixity at the bottom line, the application of liquid pressure along the bottom line, and adjustments to the calculation parameters.

Figure 12.8: *Creation of Stage 1.*

12.2.4.2.1 Unassign fixities

To remove the liquid fixity at the bottom of the model go to <Boundary Conditions → Fixities → Liquid Fixities> and after right clicking on the group corresponding to the bottom side, select <Unassign group → Fully delete the group>. To check the correct unassignment right-click on the option *Liquid Fixities* → *Draw* → *Draw values* as visible in Figure 12.9.

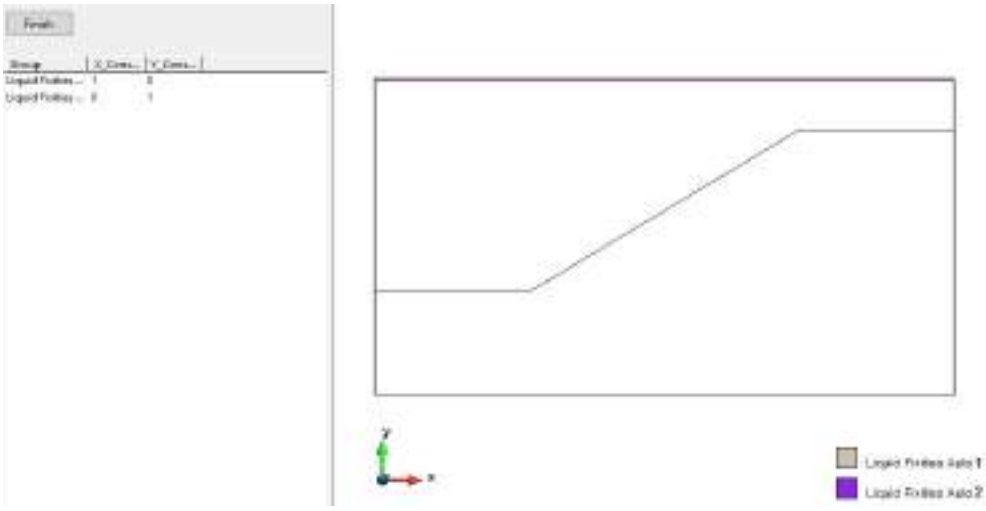


Figure 12.9: *Unassignment of liquid fixity at the bottom of the slope.*

12.2.4.2.2 Applying loading conditions

A pore pressure equal to –8kPa is applied at the bottom line. In this exercise the MPs adjacent to this boundary will not move significantly and therefore the pressure can be applied on the nodes. Choose <Boundary conditions → Loading Conditions → Liquid Pressure> and apply an uniform liquid pressure of –8kPa at the nodes and <Load system> A (Figure 12.10). Note that a negative value means compression in Anura3D .

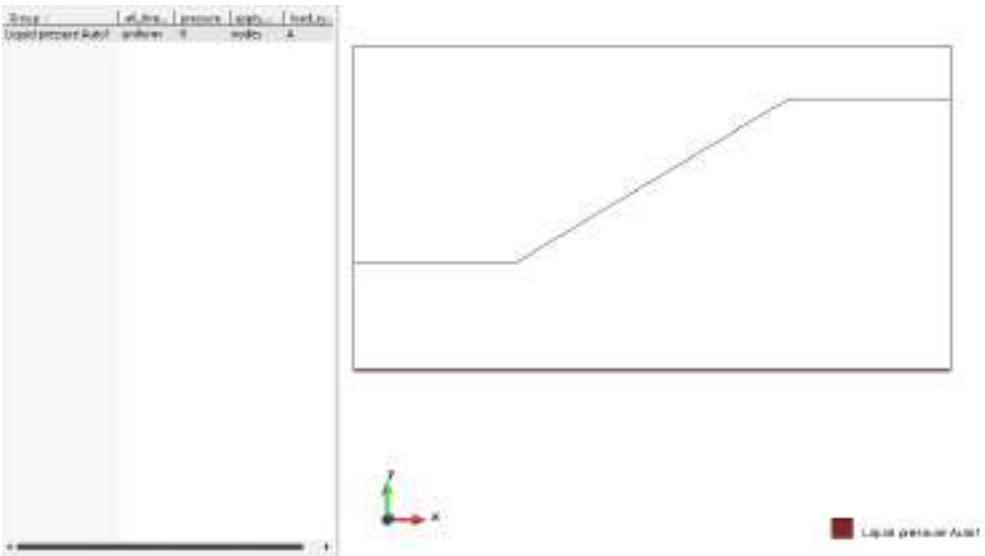


Figure 12.10: *Applied liquid pressure at the bottom of the slope.*

12.2.4.2.3 Defining calculation data

Figure 12.11 shows the calculation data of *Stage 1*.

Calculation step data
This stage of the analysis will last up to calculation step 2 that will last 0.5s. Set *number of calculation steps [-]* to 2 and *time per calculation step [s]* to 0.5.

Liquid pressure
The uniform distribution of liquid pressure is applied at the left of the model, thus choose *apply liquid pressure - linear* and set the multipliers for gravity loading as 1.0 (initial and final value).

Quasi-static convergence
This stage is performed trough a dynamic calculation. Thus, set the *QUASI-STATIC CONVERGENCE* to *do not apply convergence criteria*.

Homogeneous local damping
The local damping coefficient is reduced to 0.02.

Figure 12.11: *Dialogue box to define calculation parameters in GiD for Stage 1 of the submerged slope problem.*

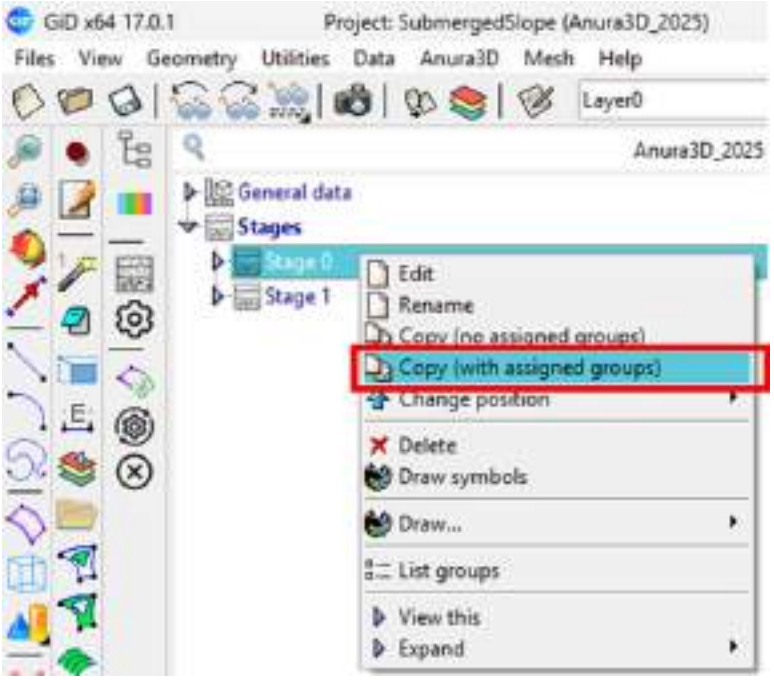


Figure 12.12: *Creation of Stage 2.*

12.2.4.3 Stage 2: Slope stabilization

In this stage, the liquid pressure is removed and liquid fixity at the bottom of the model is re-applied, thus, the boundary conditions are identical to those in textitStage 0. To duplicate all information from *Stage 0*, right-click on *Stage 0* and select *copy (with assigned groups)* (Figure 12.12).

12.2.4.3.1 Defining calculation data

The only difference compared to *Stage 0* is the calculation parameters. Figure 12.13 shows the calculation data of *Stage 2*.

Calculation step data
This stage of the analysis will last up to calculation step 35 and each calculation step will last 0.15s. Set *number of calculation steps [-]* to 35 and *time per calculation step [s]* to 0.15.

Homogeneous local damping
The local damping coefficient is reduced to 0.02.

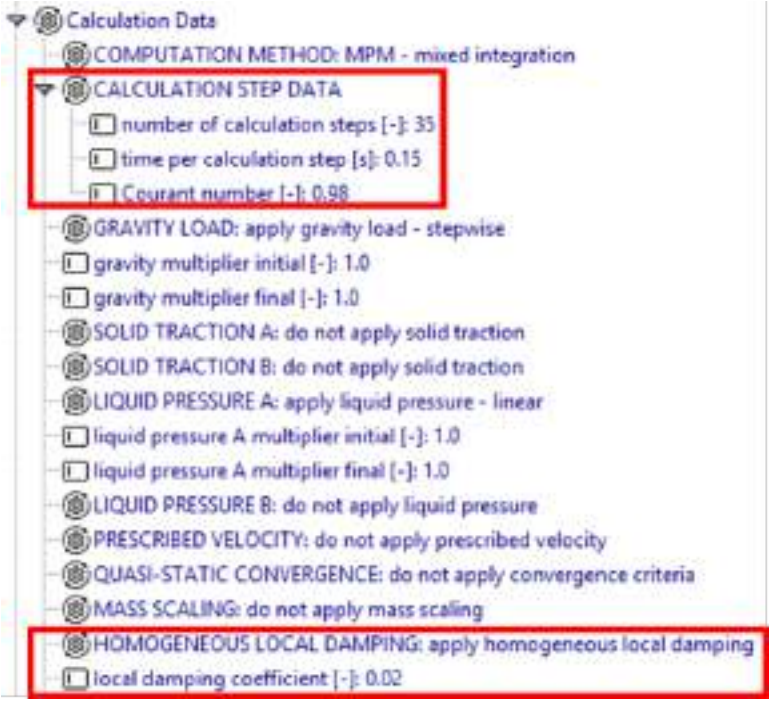


Figure 12.13: *Dialogue box to define calculation parameters in GiD for Stage 2 of the submerged slope problem.*

12.2.5 Creating mesh

In GiD it is possible to generate meshes in a very flexible way. An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example set the mean element size to "0.08". The mesh should count 936 triangular elements and 511 nodes as shown Figure 12.14.

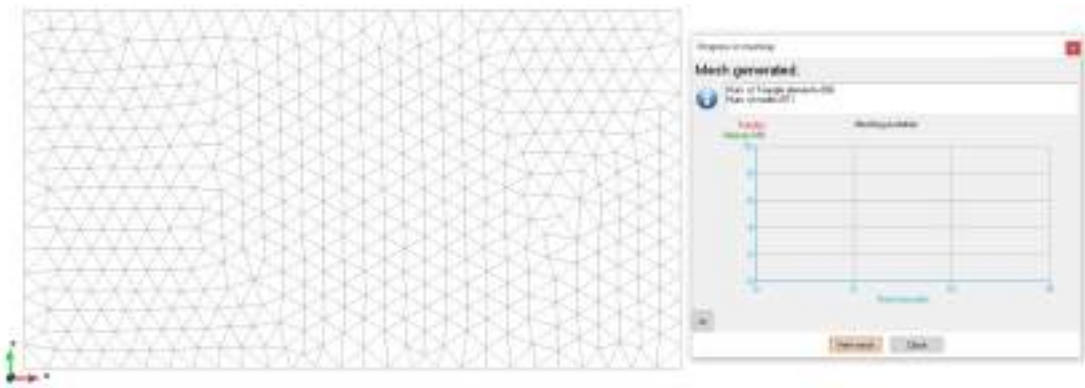


Figure 12.14: *Computational mesh of the submerged slope problem.*

12.3 Performing calculation in Anura3D

To start the calculation go to <Anura3D → Calculate>. The three stages will be computed automatically. Figure 12.15 .

Figure 12.15: *Calculate initiation.*

The generated input files for *Stage 0* of Anura3D simulation can be found in the newly created folder `RainfallInfiltration.A3D`, within the directory specified earlier by the user (Sec. 12.2.1):

D:\..\SubmergedSlope.A3D\SubmergedSlope.GOM_stage0

D:\..\SubmergedSlope.A3D\SubmergedSlope.OPD_stage0

D:\..\SubmergedSlope.A3D\SubmergedSlope.CPS_001

Once calculation of *Stage 0* is finished the necessary files for *Stage 1* are generated automatically:

D:\..\SubmergedSlope.A3D\SubmergedSlope.GOM_stage1

D:\..\SubmergedSlope.A3D\SubmergedSlope.OPD_stage1

D:\..\SubmergedSlope.A3D\SubmergedSlope.CPS_002

The same will occur for *Stage 2* after *Stage 1* is calculated:

D:\..\RainfallInfiltration.A3D\RainfallInfiltration.GOM_stage2

D:\..\RainfallInfiltration.A3D\RainfallInfiltration.OPD_stage2

D:\..\RainfallInfiltration.A3D\RainfallInfiltration.CPS_023

12.4 Results

12.4.1 Stage 0: Gravity Loading

At the end of the first calculation step, the effective stresses are initialized and are in equilibrium with the gravity. Note that in a submerged calculation the submerged weight of the soil-water mixture is used, i.e. $\gamma' = \gamma_{sat} - \gamma_{liquid}$, and the liquid weight is neglected. The recommended way to visualise Anura3D results is by using the ParaView software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results.

Figure 12.16 shows the vertical effective stress at the end of calculation step 1. The liquid pressure and the body force of the liquid are 0.

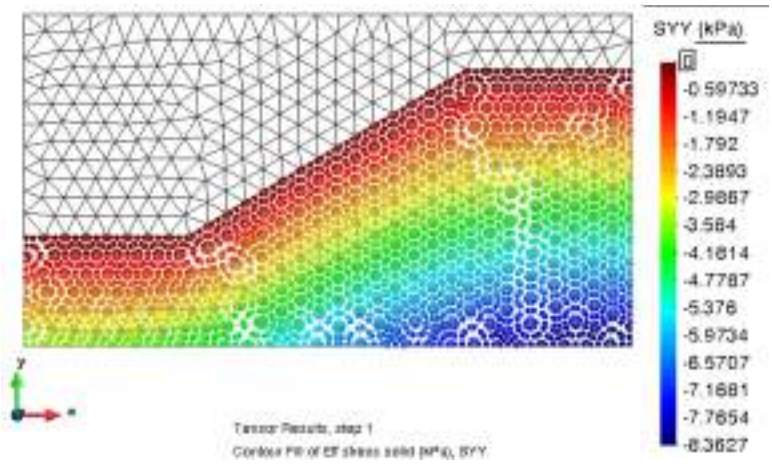
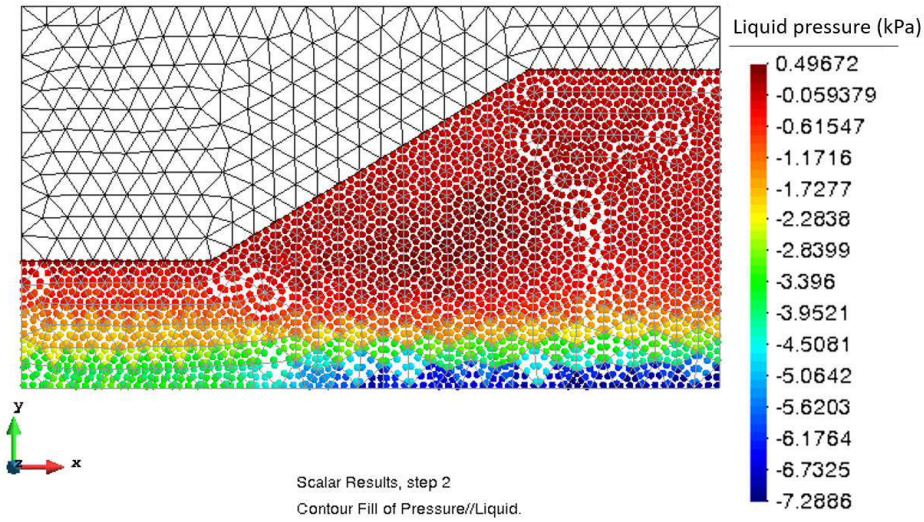


Figure 12.16: *Vertical effective stress at the end of calculation step 1.*

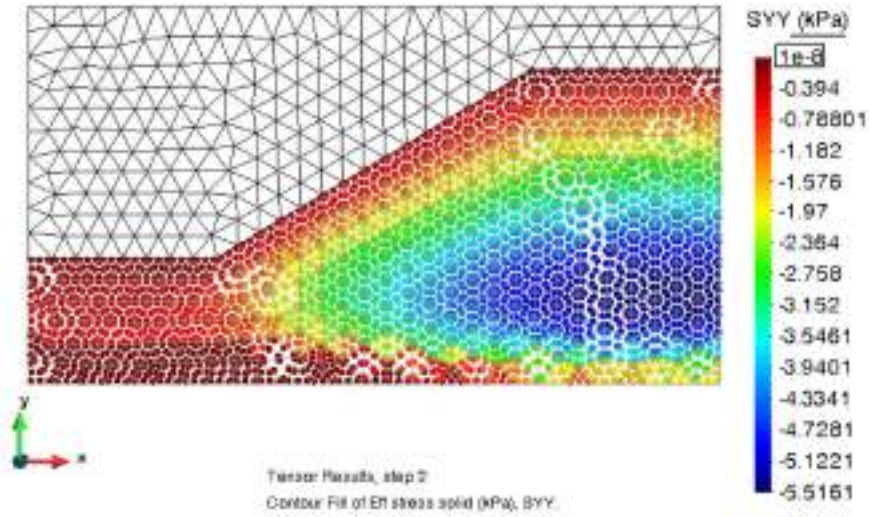
12.4.2 Stage 1: Slope Failure

In this calculation step the failure of the slope is induced by the application of a liquid pressure at the bottom of the slope for a duration of 0.05s.

The pore pressures (i.e. excess of pore pressure with respect to the hydrostatic pressure) increase at the bottom of the slope, while the vertical effective stresses decrease (Figure 12.17), thus inducing the failure of the slope.



(a)



(b)

Figure 12.17: *(a) Pore pressure and (b) vertical effective stress at the end of calculation step 2*

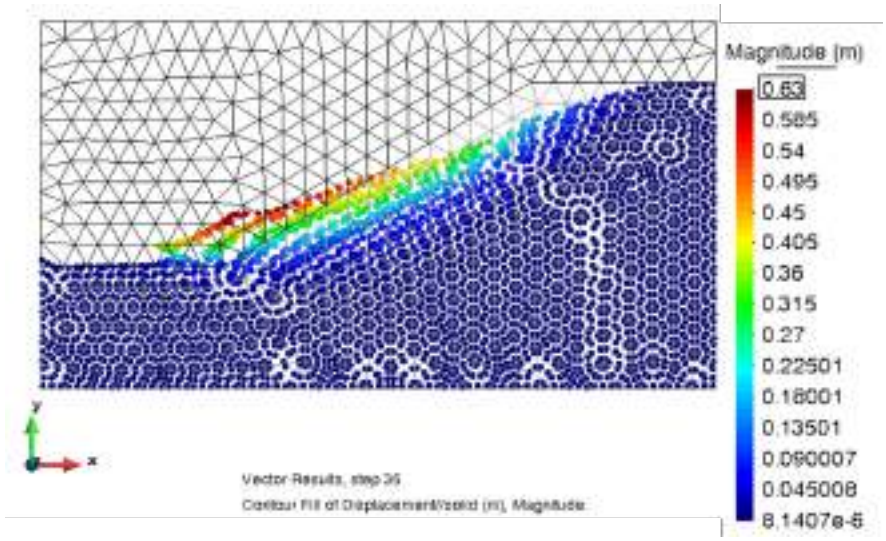
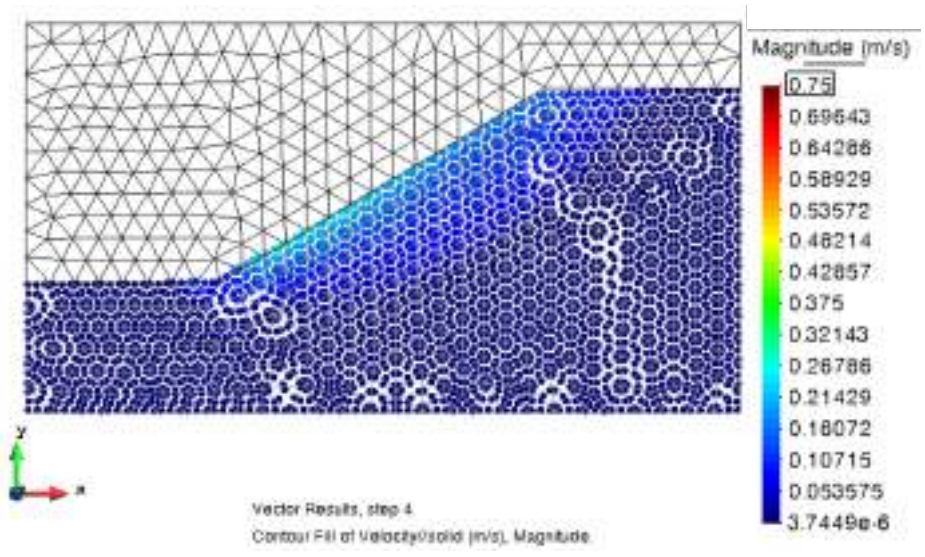


Figure 12.18: *Displacement of the MPs at the end of the simulation.*

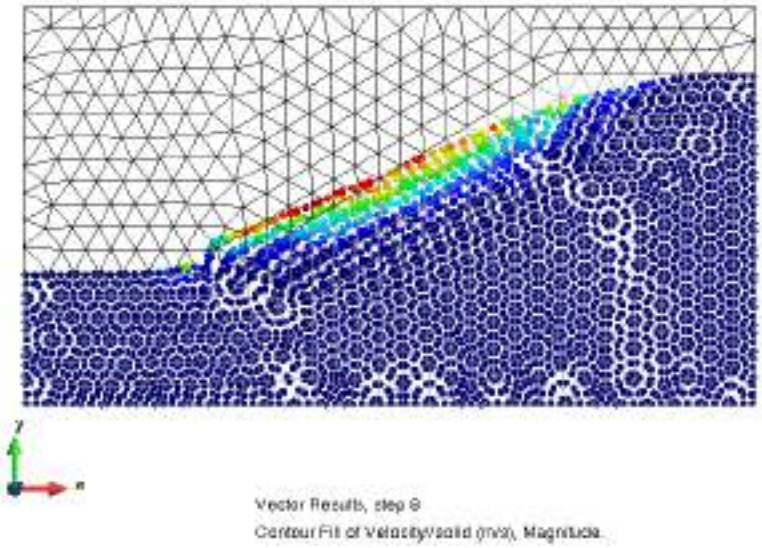
12.4.3 Step 2: Slope Stabilization

At this stage, the liquid pressure at the bottom of the slope is removed and the liquid fixity is re-established.

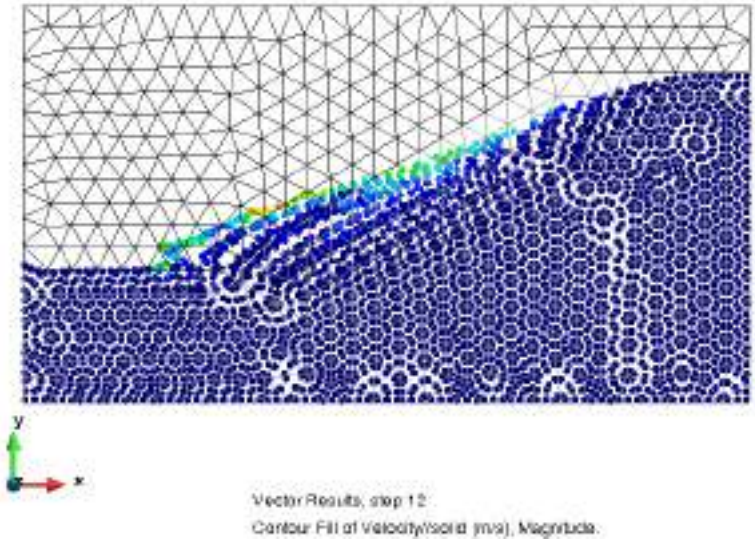
Since at this stage no pore pressure is applied at the bottom, the effective stress increases again, thus increasing the soil shear strength. The mobilized material initially accelerates, then it slows down (Figure 12.19), and finally, the slope comes to rest and a new deformed configuration (Figure 12.18).



(a)



(b)



(c)

Figure 12.19: *Velocity at MPs for different calculation steps. (a) Step 4; (b) step 8 and (c) step 12.*

13 Slope failure under transient hydraulic loads

Authors: Veronica Girardi; Gaia Di Carluccio; Francesca Ceccato; Alba Yerro

In this tutorial, two typical macro-instabilities of water retaining geostructures are simulated with the two-phase formulation for unsaturated soils of Anura3D. The first mechanism, reported in Figure 13.1(a), is induced by a rapid draw-down, controlled by a transient total hydraulic head condition applied on the slope boundary. The second mechanism, reported in Figure 13.1(b), is triggered by a combination of high saturation of the slope body with an intense rainfall. In this case, rainfall and potential seepage face boundary conditions are applied on the slope.

Both cases are simulated in 2D-plane strain conditions. The K_0-procedure is used to expedite convergence during the first load step, where gravity is applied and quasi-static conditions considered. The K_0-procedure, when working with a problem in which the two-phase formulation for unsaturated soils is adopted, requires the user to specify:

(1) an initial soil surface
(2) an initial phreatic surface

The initial soil surface is selected by the user, after creating the slope geometry in GID. Whereas, the initial phreatic surface can be specified with one of two available methodologies: sketched by the user while creating the geometry in GID or assigned from an external file, which specifies the coordinates of two or more points associated to the phreatic surface location.

After initialization, the two problems under analysis are modelled in fully dynamic conditions, applying respective hydraulic boundary conditions. Hence, failure and post-failure behaviors can be captured.

This tutorial demonstrates the use of

(1) two phase formulation for unsaturated soil
(2) transient hydraulic boundary conditions (total head, rainfall and seepage face)
(3) K_0-procedure with the user-specified initial soil and phreatic surfaces. The latter can be either specified in the GID preprocessor or assigned considering an external file

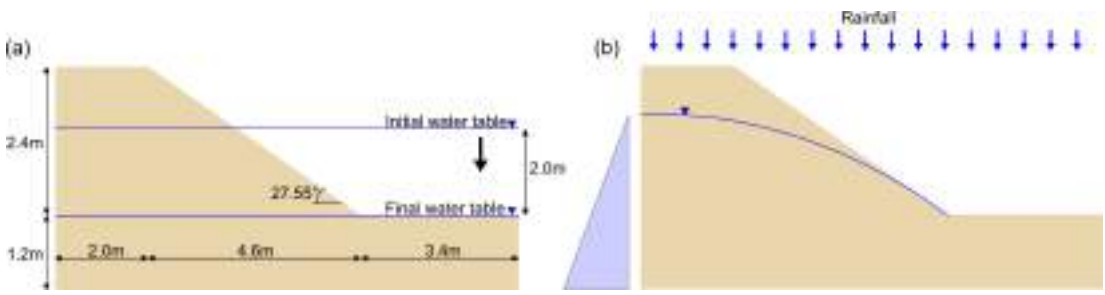


Figure 13.1: *Problem definition for failure of unsaturated slopes. Two mechanisms are analysed: an instability induced by rapid draw-down (a) and one triggered by the combination of high saturation and rainfall (b).*

13.1 Problem definition

The slope geometry and the dimensions of the problem are given in Figure 13.1. The soil is modelled with the Mohr-Coulomb model and a linear SWRC with the properties given in Table 13.1.

Table 13.1: *Material parameters of the soil.*

material	soil
material type	unsaturated material - 2-phase with suction effects
initial porosity [–]	0.4
density solid [kg/m^3]	2650
density liquid [kg/m^3]	1000
intrinsic permeability liquid [m^2]	6e-12
Bulk modulus liquid [kPa]	60000
dynamic viscosity liquid [kPa · s]	1e-6
K_0- value [-]	0.5
material model solid	Mohr-Coulomb
effective Poisson ratio [–]	0.3
effective Young modulus [kPa]	30000
effective cohesion [kPa]	1
effective friction angle [°]	27
dilatancy angle [°]	0.0
tensile strength [kPa]	0.0
liquid retention curve	linear
linear coefficient [1/kPa]	4e-4
hydraulic conductivity curve	constant

13.2 Rapid Drawdown

13.2.1 Creating input data

The geometry, discretization, material properties, and all other input parameters will be defined using the GiD pre-pocessor. This follows a similar procedure as described in Section 3.2. Here only the differences with respect to previous examples will be highlighted.

13.2.1.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Set the problemtype as described in Section 3.2.1 step (2). Save the project in the desired directory with an appropriate name (e.g. "RapidDrawDown").

13.2.1.2 Defining general data

The problem is modelled under 2D-plane strain conditions; therefore, under the *General data* menu go to *Problem dimensions* and set it to *2D:plane-strain*. Next, go to *MPM discretization type* and choose *single_point*, as the single point formulation will be used. Finally, for result visualization, click on *Post-process visualization software* and ensure only the option *ParaView* is selected.

13.2.1.3 Defining geometry

To generate the geometry follow the instructions given in Section 3.2.2. Figure 13.1 reports the slope dimensions. The computational domain is extended vertically above the crest of the slope, the entire domain is bounded by four points of coordinates P1 = (0; –0.6); P2 = (10; –0.6); P3 = (10; 3.6); P4 = (0; 3.6). The initial water table location corresponds to an horizontal line at y=2m. After drawing the line, the command *Geometry create IntersectLine* is used to cut the lines intersected by the last line drawn. All these lines need to be selected (as displayed in Figure 13.2) then press "esc" on your keyboard to obtain the final modification.

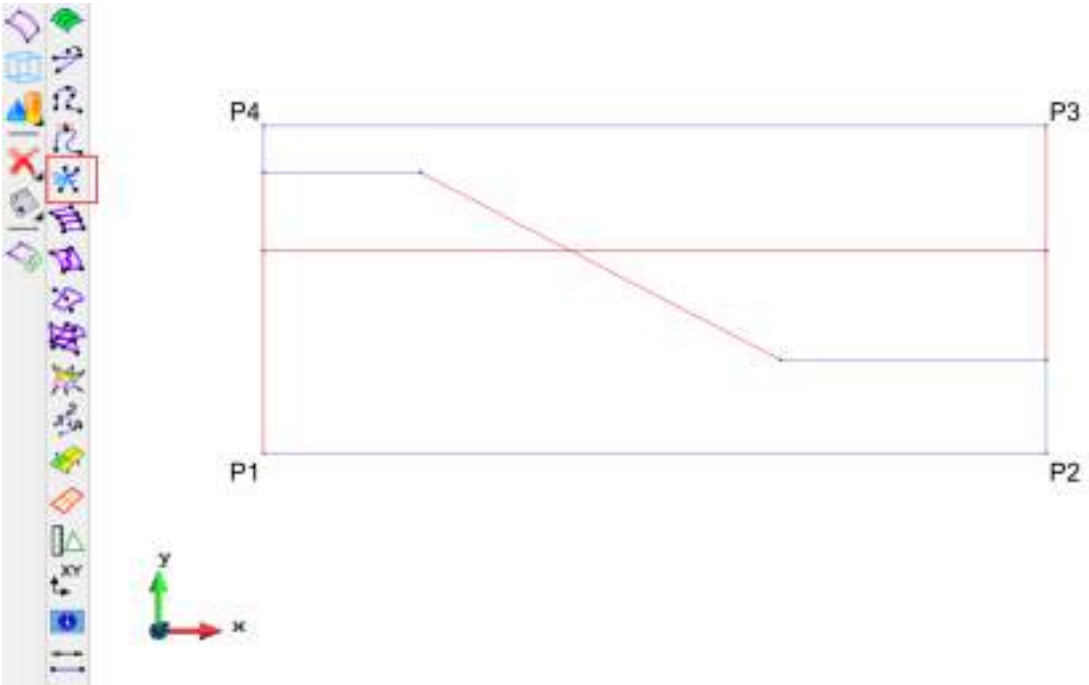


Figure 13.2: *Lines definition with initial water table as a geometrical entity (line). Red lines are selected after using the command Geometry create IntersectLine.*

Now it is possible to create the surfaces and complete the geometry definition, as visible in Figure 13.3.

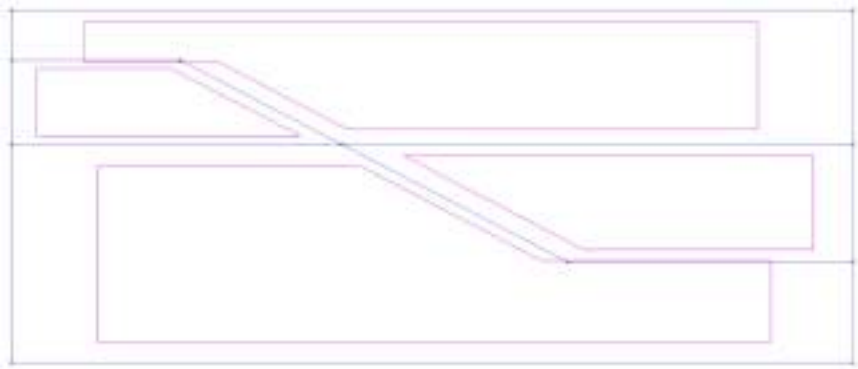


Figure 13.3: *Geometry definition of the rapid draw down problem.*

13.2.1.4 Defining stages

This example consists of two stages of calculation. In the first stage (Stage 0), stresses and pore water pressures are initialized by applying gravity loading. In the second stage (Stage 1), the slope failure is induced by a rapid draw-down.

13.2.1.4.1 Stage 0 - Gravity loading

13.2.1.4.1.1 Defining material properties

Define a new material with an appropriate name (e.g. "Slope") as explained in 3.2.5. Material parameters are given in Table 13.1 and shown in Figure 13.4, however during the initialization phase two parameters are different: permeability should be assigned a value of $6 \cdot 10^{-11} m^2$, to speed up the consolidation process, and effective cohesion c = 200kPa to prevent slope movements during this phase. Subsequently, in Stage 1, these values will be modified to those reported in Table 13.1.

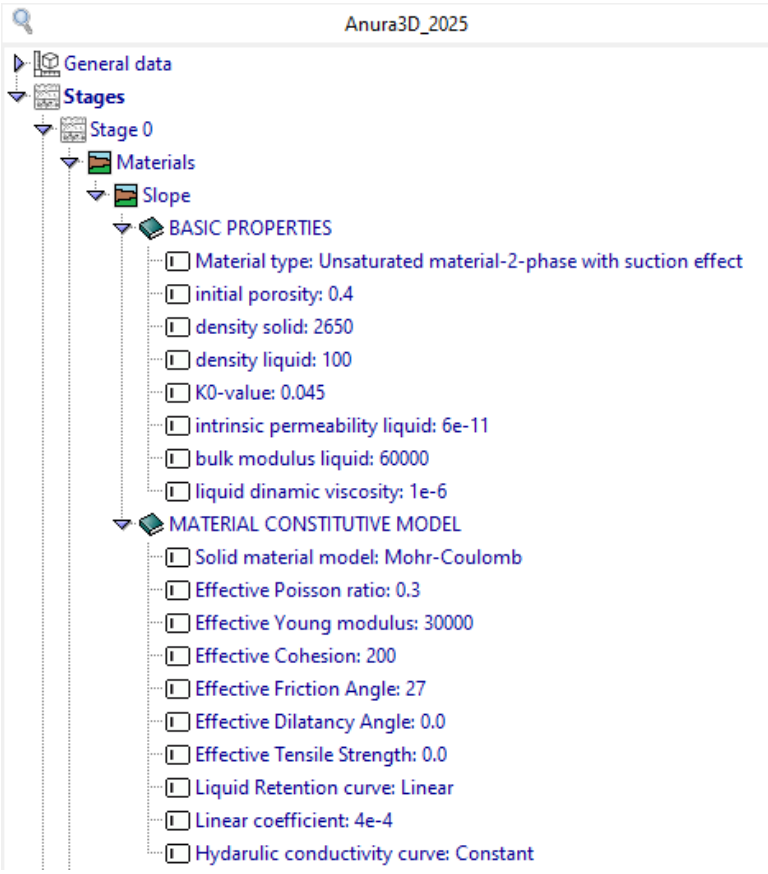


Figure 13.4: *Defining material properties for Stage 0 of the rapid draw down problem in GiD.*

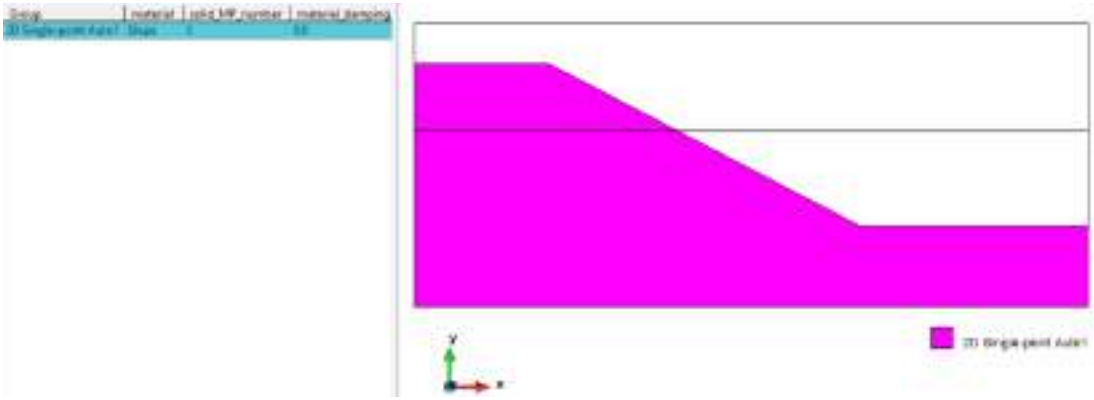


Figure 13.5: *Specification of the material and number of material points per element in the rapid draw down problem.*

13.2.1.4.1.2 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. In this example choose 3 material points per element for the slope. Check if the material and the number of material points are assigned correctly (Figure13.5).

13.2.1.4.1.3 Defining initial conditions

The stresses in the soil will be initialised by gravity loading, but to facilitate the convergence, the K_0 procedure is used, thus select <apply> on the <K0-procedure> option. This option can be found in the Anura3D tree menu, under the section <Initial conditions → Stress initialization>.

The K_0-procedure requires the specification of the soil and the phreatic initial surfaces (in this case lines since the model is in 2D). These options are found in the Anura3D tree menu under the section <Initial conditions → Stress initialization → General conditions>. Double click on <Soil Surface> and <Select> the lines corresponding to the slope profile, as reported in Figure 13.6, then, click on <End> and <OK>.

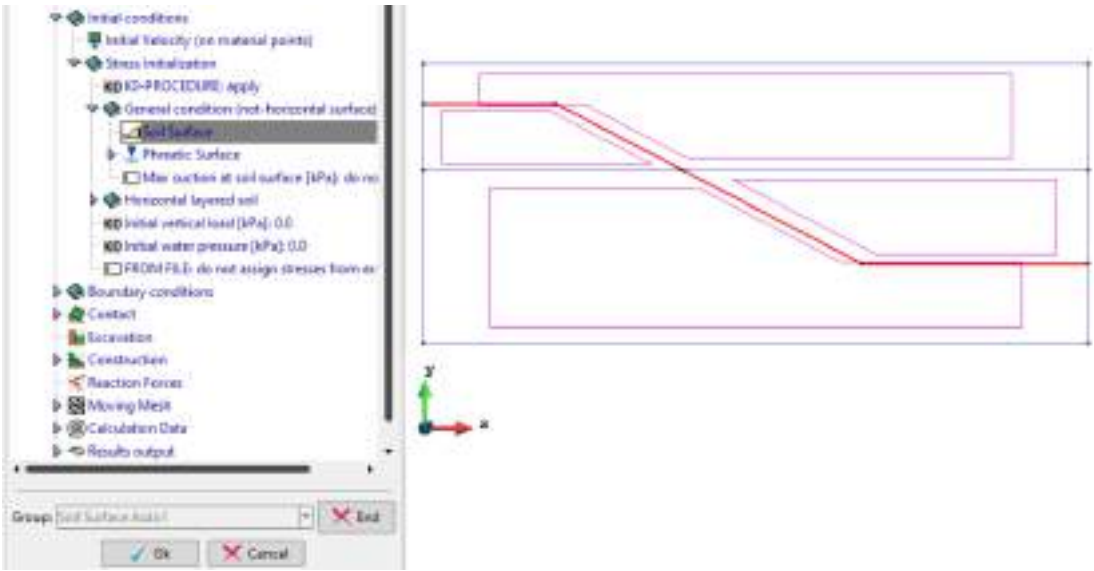


Figure 13.6: *Assign soil surface specification to lines for the rapid draw down problem.*

The correct assignment can be verified by right-clicking on the option <*Soil Surface* → *Draw* → *Draw values*>, as visible in Figure 13.7

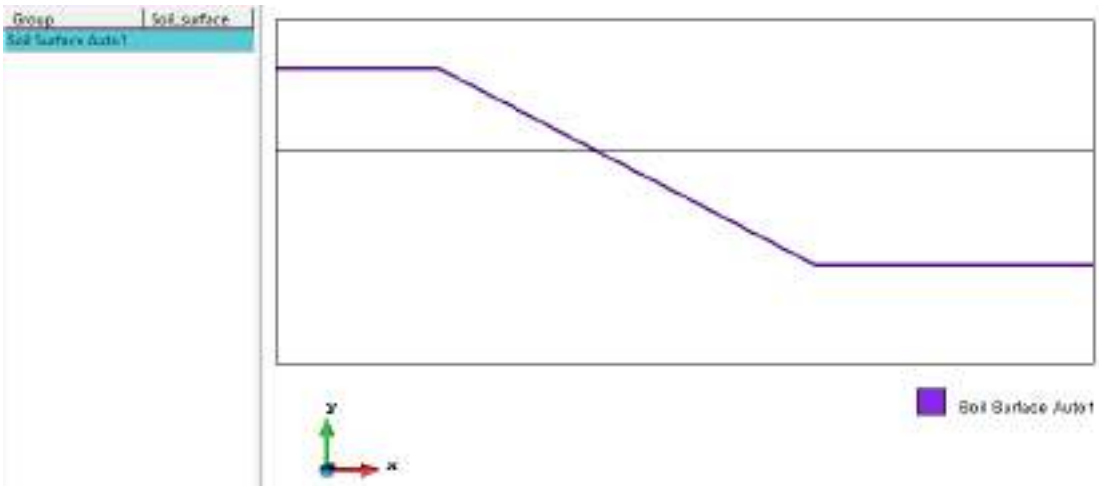


Figure 13.7: *Checking the initial soil surface correct assignment for the rapid draw down model.*

After specifying the soil surface, once more in the Anura3D tree menu, click on <Phreatic Surface → Select water table profile>. Once clicked on <Select>, the lines corresponding to the initial water table location need to be selected, as reported in Figure 13.8. If the initial phreatic surface is not specified by the user, it is automatically located along the horizontal lower edge of the model. The max suction at soil surface is not specified, thus the suction values above the initial river level will follow a linear approximation.

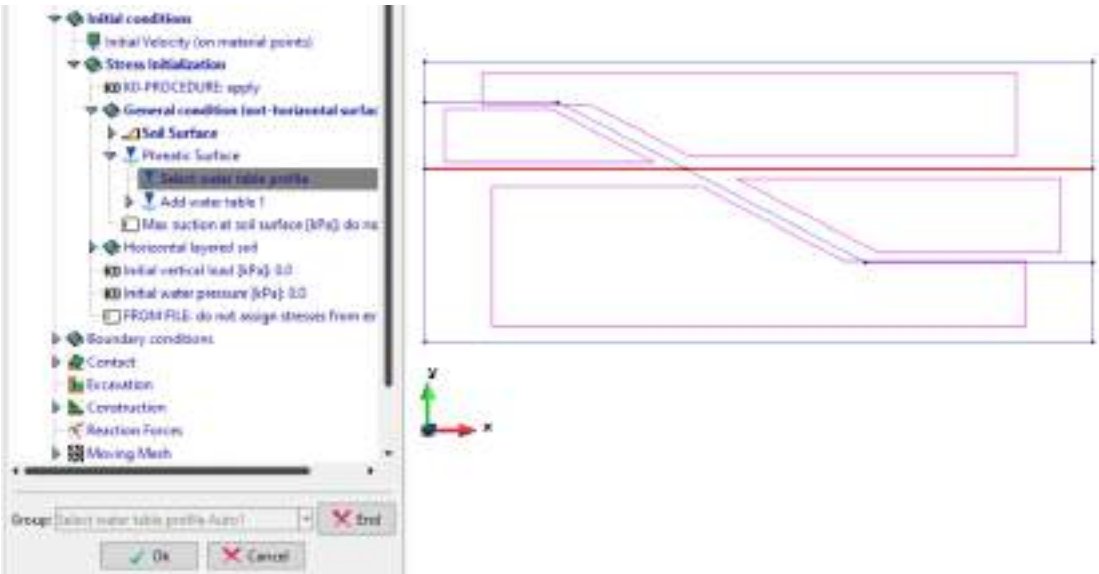


Figure 13.8: *Assign phreatic surface specification to lines for the rapid draw down problem.*

The correct assignment can be verified by right-clicking on the option <Phreatic Surface → Draw → Draw values>, as visible in Figure 13.9

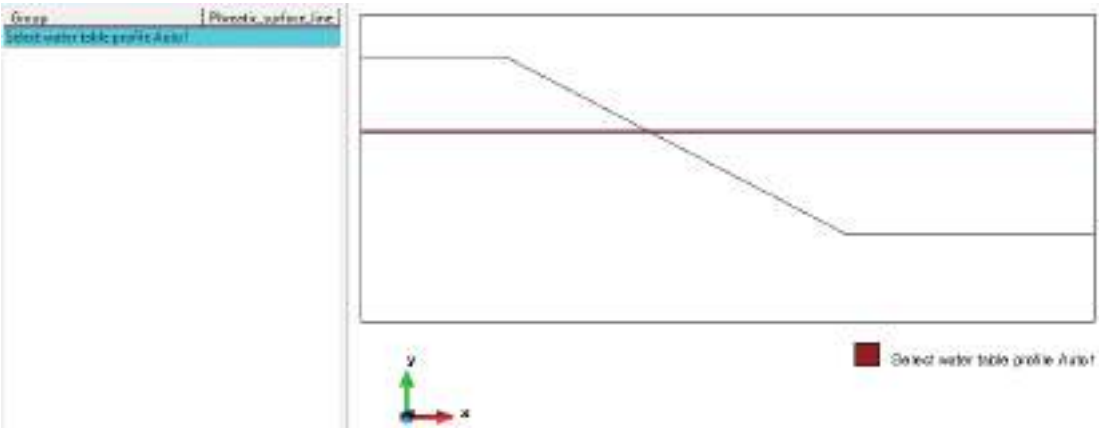


Figure 13.9: *Checking the initial phreatic surface correct assignment for the rapid draw down problem.*

13.2.1.4.1.4 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying fixities

Since in this example a fully coupled two-phase analysis is performed, solid and liquid fixities must be assigned. The solid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;
(4) Bottom side: full fixities in x- and y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 13.10;

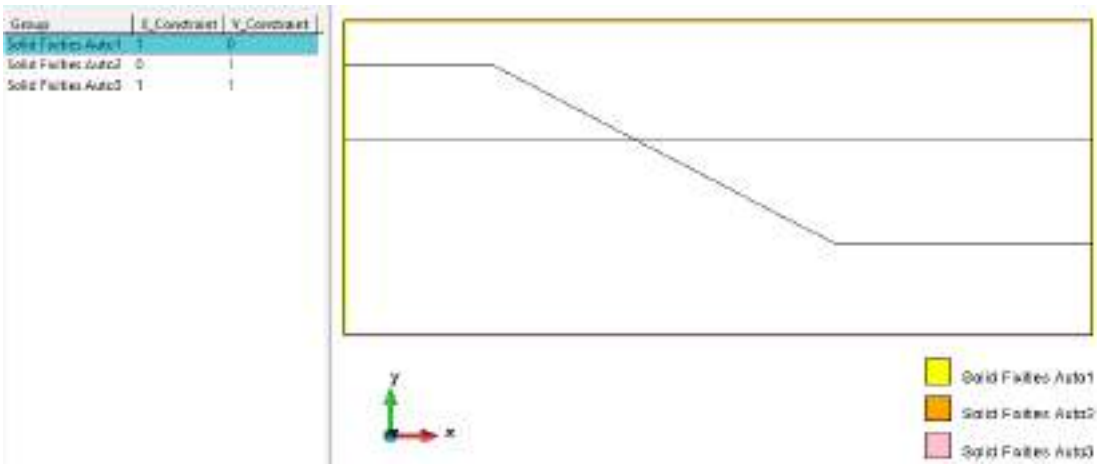


Figure 13.10: *Solid fixities for the rapid draw down problem.*

The liquid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;

(4) Bottom side: vertical fixity in y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 13.11;

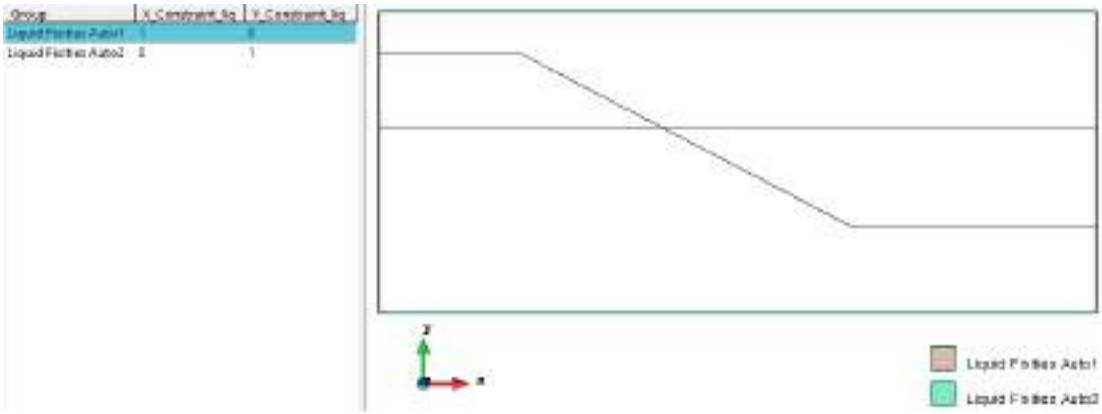


Figure 13.11: *Liquid fixities for the rapid draw down problem.*

Applying hydraulic boundary conditions

The hydraulic boundary conditions, i.e. total hydraulic head, potential seepage face and rainfall, are assigned to the boundary nodes of the body under analysis. Since the body is expected to experience large displacements, the initial boundary nodes may differ from those at later stages of the simulation, thus it is necessary to define a potential area/volume for searching of boundary nodes, as explained in [9]. This area is specified in GID, within the Anura3D menu, under the section <Boundary Conditions → Hydraulic Conditions>.

Here, it is possible to define for each hydraulic boundary condition, the coordinates of minimum and maximum points that delineate the rectangular area/ box volume where the condition is supposed to act throughout the simulation. In the rapid draw down problem it is necessary to assign an area to the total hydraulic head condition, and an equal area to the potential seepage face. In this manner, above the current river level (at the beginning of this simulation located at y = 2m) the boundary condition applied to the slope is the potential seepage face.

First, under the section <Hydraulic Conditions → Hydraulic head>, specify the coordinates of the minimum (*x- and y- min*) and maximum (*x- and y-max*) points defining the rectangular area where the condition applies. These correspond to the point P1 and P3 in the geometry shown in Figure 13.2). The final assignment should look as reported in Figure 13.12. Next, double-click on <Hydraulic head in time> and input the values shown in Figure **??**. These values represent the initial river level maintained throughout the initialization phase. If the simulation time is bigger then the last time instant specified in this file, Anura3D will automatically consider this last value of river height for the following part of the simulation.

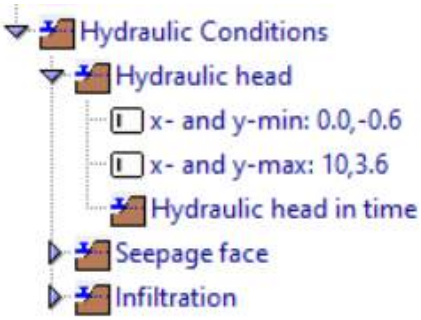


Figure 13.12: *Specify the coordinates defining the total hydraulic head potential area for the rapid draw down problem.*

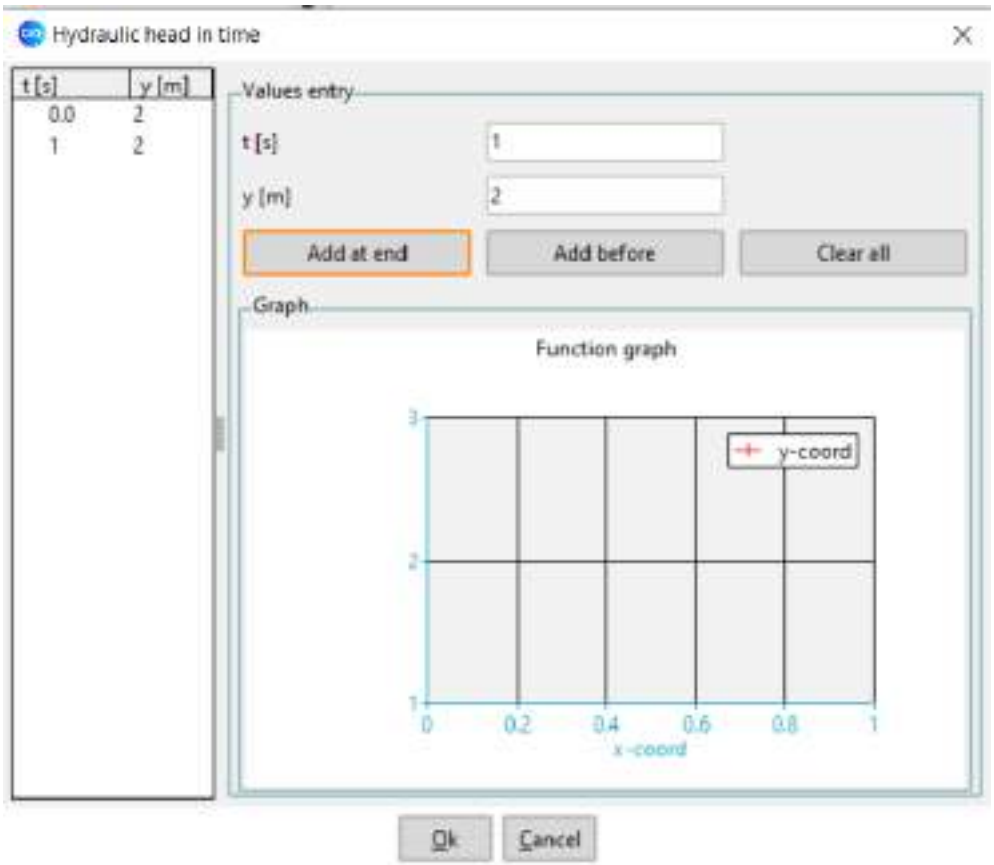


Figure 13.13: *Assign the hydraulic head in time for Stage 0 of the rapid draw down problem.*

To apply the seepage face condition, select <Seepage face> in the menu and specify the same coordinates as the hydraulic head condition. The final assignment is shown in Figure 13.14.

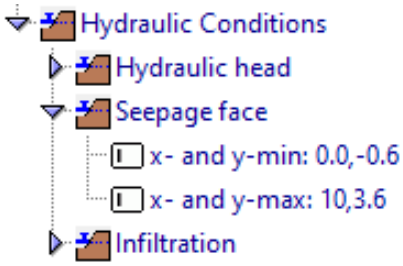


Figure 13.14: *Specify the coordinates defining the seepage face potential area for the rapid draw down problem.*

13.2.1.4.1.5 Defining calculation data

Figure 13.15 shows the calculation data of Stage 0.

Computation method
The option *MPM - mixed integration* should be selected.

Calculation step data
Gravity loading is performed in one calculation step. The end of the step is determined by the convergence criteria. A Courant number of 0.98 is advised to ensure the stability of the solution. Set *number of calculation steps [-]* to 1.

Gravity load
Gravity is applied from the beginning, therefore choose *apply gravity load - stepwise* and set the multipliers for gravity loading as 1.0 (initial and final value). The gravity multipliers will be maintained constant throughout the calculation.

Quasi-static convergence
The end of the first load step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition. Set the *QUASI-STATIC CONVERGENCE* to *apply convergence criteria*. The convergence criteria are the tolerated error of the kinetic energy, the tolerated error of the out-of-balance force, or the maximum number of time steps. In this example, in order to reduce the computational time, use a maximum number of time steps equal to 2500.

Mass scaling
In this example the mass scaling is used to speed up the computation and a value of 80 is assigned to the mass scaling factor.

Homogeneous local damping
A local damping coefficient for all active elements of 0.05 is applied.

Strain smoothing
The strain smoothing algorithm will be switch on to minimise the kinematic locking.

Liquid pressure increment smoothing
The liquid pressure smoothing will be switch on to minimise pressure oscillations during the post-failure phase.

Default values

All other calculation options are not used in this example and the provided default values can be kept.

Figure 13.15: *Dialogue box to define calculation parameters in GiD for Stage 0 of the rapid draw down problem.*

13.2.1.4.2 Stage 1 - Slope failure

To create *Stage 1*, right-click on *Stage 0* and select *copy (with assigned groups)*. This will copy all the conditions from the previous stage to the current one, allowing to specify only the differences required at this stage of calculation.

13.2.1.4.2.1 *Updating material properties*

To update the material properties, first set the option *Update Materials* to *apply*. Then, modify the intrinsic permeability of the liquid to 6-e-12 m^2 and the effective cohesion to 1 kPa (Figure 13.16)

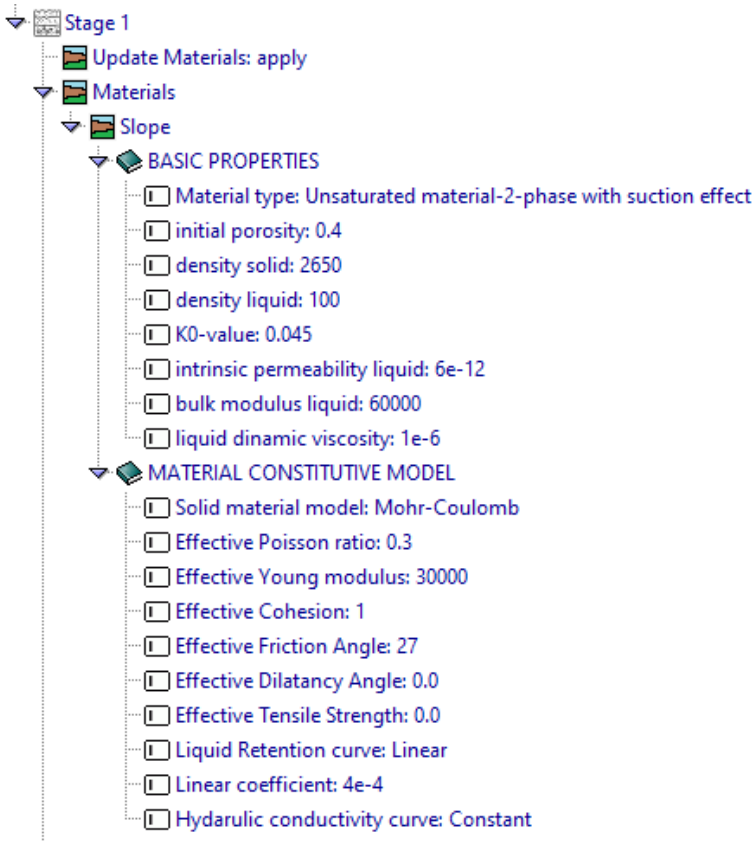


Figure 13.16: *Defining material properties in GiD for Stage 1 of the rainfall infiltration problem.*

13.2.1.4.2.2 *Defining hydraulic boundary conditions*

In this stage the failure of the slope is induced by the rapid draw down, so, the hydraulic head in time have to be modified as shown in Figure. It has not be noted that the speed of the drawdown is set as extremely rapid. This is due to the large values of hydraulic conductivity used in this problem, with the aim of speeding up computational time.

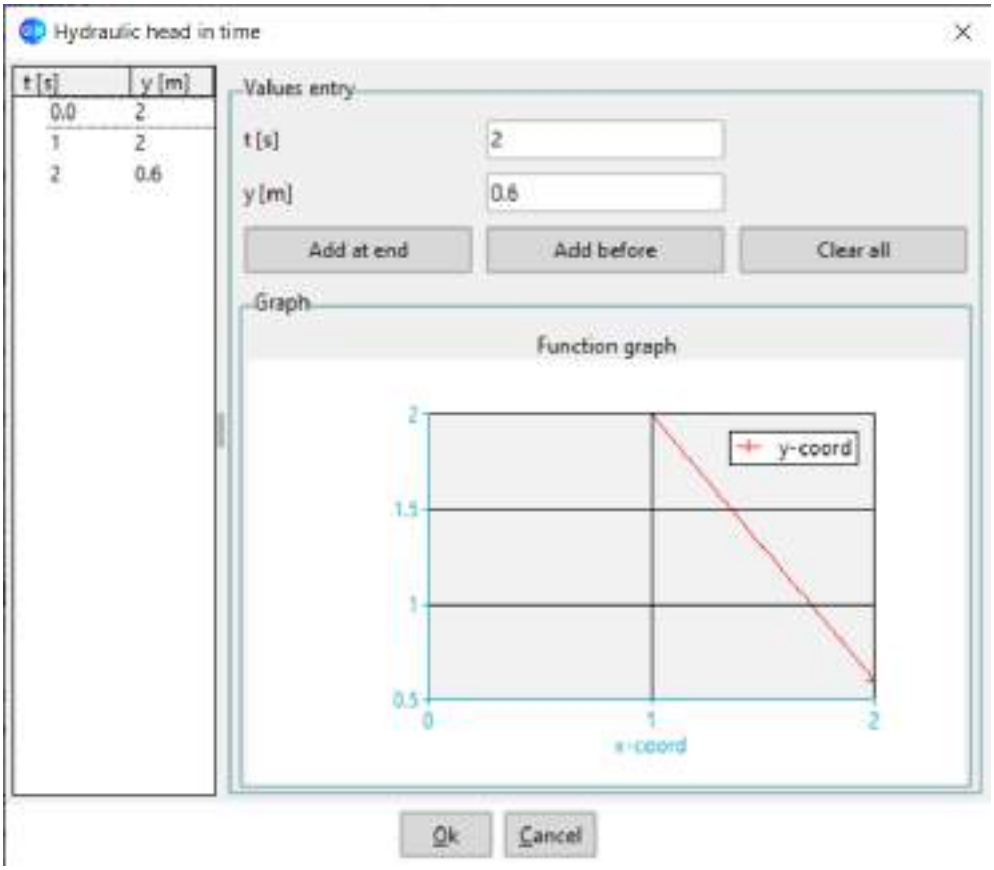


Figure 13.17: *Assign the hydraulic head in time for Stage 1 of the rapid draw down problem.*

13.2.1.4.2.3 Defining Calculation data

Figure 13.18 shows the calculation data of Stage 1.

Calculation step data
This stage of the analysis will last up to calculation step 19 and each load step will last 1s. Set *number of calculation steps [-]* to 19 and *time per calculation step [s]* to 1.

Quasi-static convergence
This stage is performed trough a dynamic calculation. Thus, set the *QUASI-STATIC CONVERGENCE* to *do not apply convergence criteria*.

Figure 13.18: *Dialogue box to define calculation parameters in GiD for Stage 1 of the rapid draw down problem.*

13.2.1.5 Creating mesh

An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example the size assigned to all the surfaces is "0.4", and after click on *Generate mesh*, the size of elements to be generated should be set equal to 1. The mesh should count 554 triangular elements and 313 nodes as shown Figure 13.19.

Figure 13.19: *Computational mesh of the rapid draw down problem.*

13.2.2 Performing calculation in Anura3D

To start the calculation go to <Anura3D> → <Calculate>. The two stages will be computed automatically.

The generated input files for Stage 0 of Anura3D simulation can be found in the newly created folder `RapidDrawDown.A3D`, within the directory specified earlier by the user (Sec. 13.2.1.1):

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.GOM_stage0
```

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.OPD_stage0
```

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.CPS_001
```

Once calculation of Stage 0 is finished the necessary files for Stage 1 are generated authomatically:

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.GOM_stage1
```

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.OPD_stage1
```

```
D:\..\RapidDrawDown.A3D\RapidDrawDown.CPS_002
```

Alternatively, to calculate only the first stage initially, click on the *Stage Manager* icon. Adjust the status of each stage by setting them to 'Calculate' or 'Not to Calculate' as needed, and then click on *Calculate* to proceed (Figure 13.20).

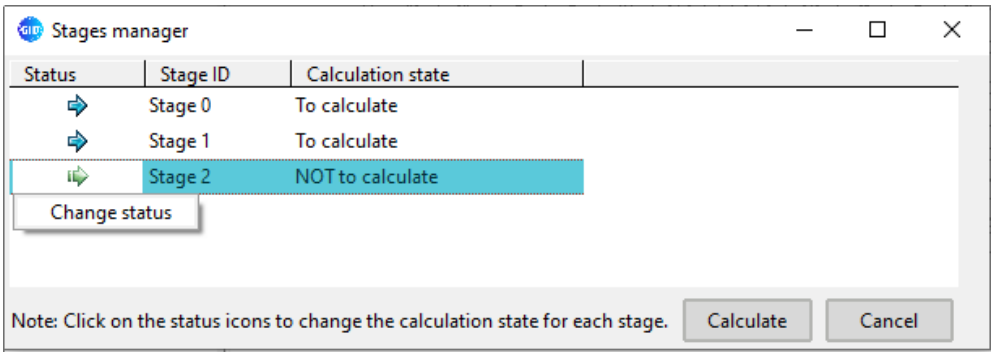


Figure 13.20: *'Stage Manager' window for the rainfall infiltration problem.*

13.2.3 Visualisation of results in ParaView

The recommended way to visualise Anura3D results is by using the Paraview software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results.

At the end of the first load step (Stage 0), the effective stresses are initialized and are in equilibrium with the gravity. Figure 13.21 shows the contour of liquid pressure (a) and total vertical stress (b) at the end of load step 1. The liquid pressure are positive for suction. Total stress account for slope material weight plus the water weight above the soil surface.

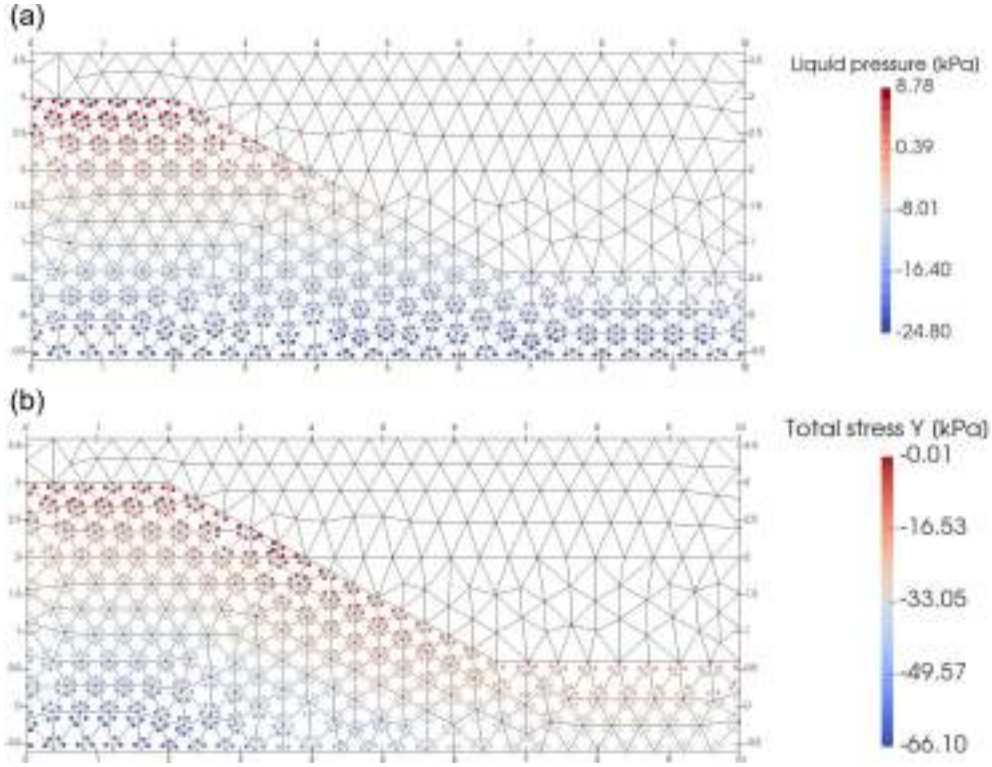


Figure 13.21: *Liquid pressure (a) and total vertical stress (b) at the end of load step 1.*

During Stage 1 the failure of the slope is induced by the rapid draw down. The river level is suddenly reduced, while the pore pressure in the slope are still high, triggering the failure mechanism, with displacements (norm) at t=18s reported in Figure 13.22.

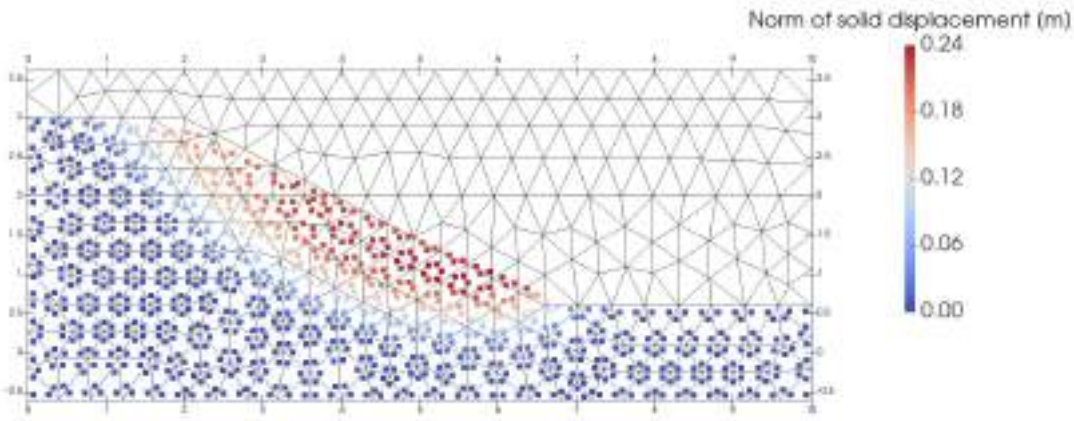


Figure 13.22: *Norm of solid displacements at the end of the post-failure phase, at 18s.*

It is possible to select MPs at the crest and at the toe of the slope, extract the selected data in terms of scalar or vector quantities and save in `.CSV` format, at one or more load steps. In Figure 13.23 norm of displacement and liquid pressure of MP1 (toe) and MP2 (crest) are plot vs time. Liquid pressures allow to visualize how the rapid river level draw down impacts on the toe values, while the crest MP (MP2) experience minimum variations from the initial value. In terms of displacements, both MP2 and MP1 show displacement progressively increasing from 3s, until reaching a stable value at t=16s, with the crest recording slightly higher displacements

compared to the toe of the slope.

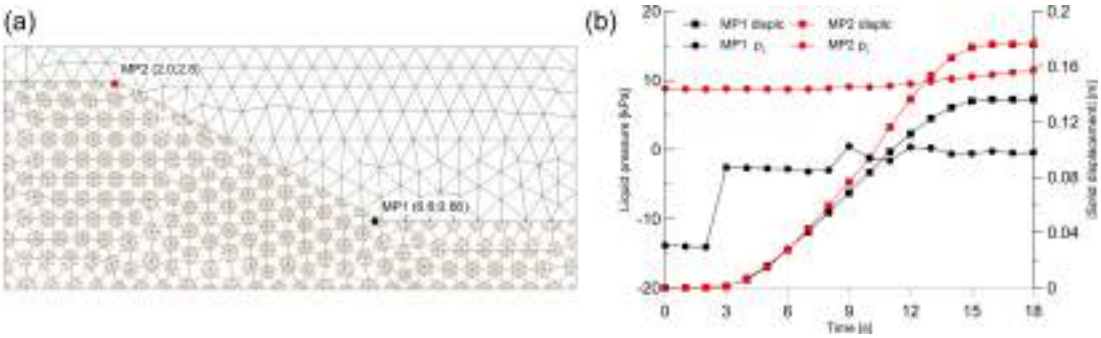


Figure 13.23: *Norm of displacement and liquid pressure (b) for MP1 and MP2. In (a) the initial location of the MPs.*

13.2.4 Variation on initial conditions: maximum suction threshold

An option to limit initial maximum suction at the slope crest during K_0 procedure is available in Anura3D. The maximum suction can be reasonably assigned by the user when for instance monitoring data are available. In this section, a variation of the rapid drawdown problem is presented by introducing a maximum suction threshold of 2 kPa. To apply this, go to <Initial conditions → Stress initialization → General conditions → Max suction at soil surface [kPa]>, set it to *specify* and then the value of 2 kPa to <Suction [KPa]>. The definition of Stage 1 remains unchanged from the reference rapid drawdown example.

The final displacements contour is reported in Figure 13.24, clearly showing the impact of the lower suction on the overall state of stress, with consequent larger displacements at the end of the post-failure phase.

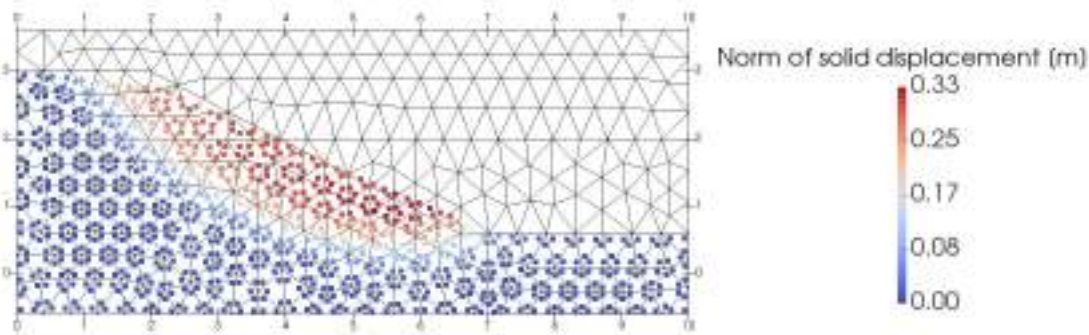


Figure 13.24: *Norm of solid displacements at the end of the post-failure phase, at 18s for the case with maximum suction threshold during K_0 procedure.*

13.3 Rainfall infiltration

13.3.1 Creating input data

The geometry, boundary conditions, material properties, calculation data and mesh are defined using GiD pre-processor. This follows a similar procedure as described in Section 3.2. Only variations to previous examples are highlighted in the following sections.

13.3.1.1 Starting pre-processor

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "RainfallInfiltration").

13.3.1.2 Defining general data

The problem is modelled under 2D-plane strain conditions; therefore, under the *General data* menu go to *Problem dimensions* and set it to *2D:plane-strain*. Next, go to *MPM discretization type* and choose *single_point*, as the single point formulation will be used. Finally, for result visualization, click on *Post-process visualization software* and ensure only the option *ParaView* is selected.

13.3.1.3 Defining geometry

To create the geometry follow the instructions given in Section 3.2.2. The slope dimensions are shown in Figure 13.1. As a reference, the key coordinates indicated in Figure 13.25 and Table 13.2 can be used. Unlike the previous example, the initial water table location is specified through its coordinates as explained in 13.3.1.4.1.3, thus no additional lines need to be created within the slope geometry.

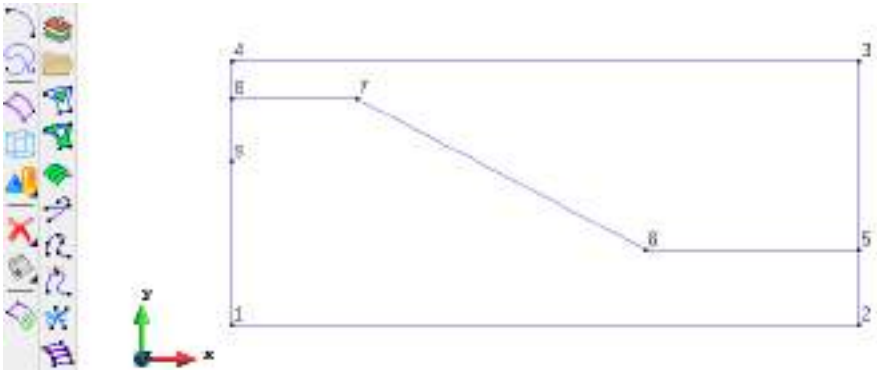


Figure 13.25: *Lines definition of the rainfall infiltration problem. Key points are indicated as given in Table 13.2*

Table 13.2: *Key coordinates of the rainfall infiltration problem.*

Point	X coordinate [m]	Y coordinate [m]
P1	0	-0.6
P2	10	-0.6
P3	10	3.6
P4	0	3.6
P5	10	0.6
P6	6.6	0.6
P7	2	3
P8	0	3
P9	0	2

Once created the surfaces, the geometry definition should look like shown in Figure 13.26.

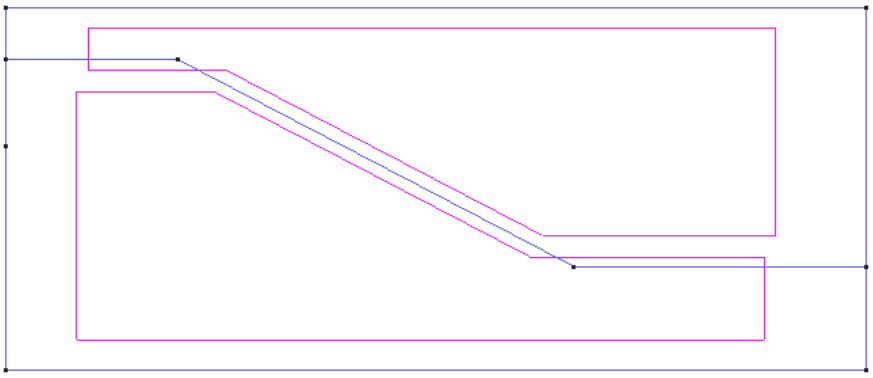


Figure 13.26: *Geometry definition of the rainfall infiltration problem.*

13.3.1.4 Defining calculation stages

This example consists of three stages of calculation. In the first stage (Stage 0), stresses and pore water pressures are initialized by applying gravity loading. In the second stage (Stage 1), the slope failure is induced by rainfall infiltration. In the third stage (Stage 2), the rainfall infiltration is deactivated, allowing the slope to reach its final stable configuration.

13.3.1.4.1 Stage 0 - Gravity loading

13.3.1.4.1.1 Defining material properties

Material parameters are the same adopted in previous example (Table 12.1). To define the material of the slope during the initialization stage (Stage 0), follow what reported in 13.2.1.4.1.1.

13.3.1.4.1.2 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. As in the previous example, choose 3 material points per element for the slope. Check if the material and the number of material points are assigned correctly (Fig. 13.27)

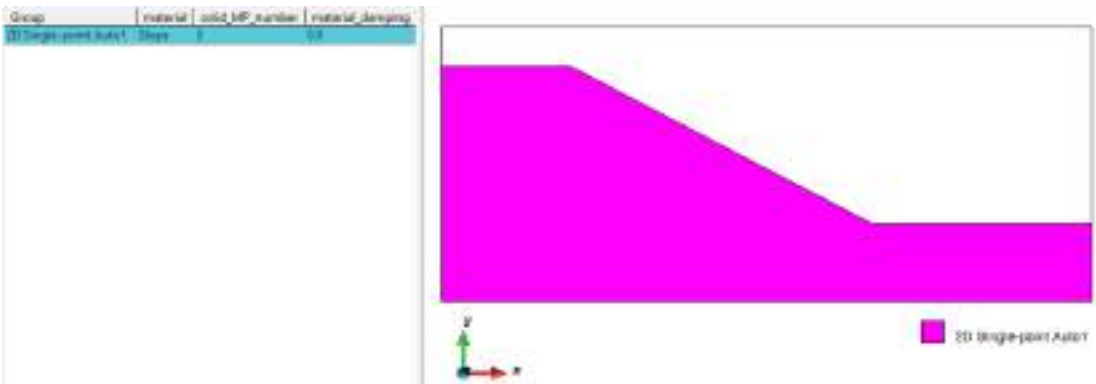


Figure 13.27: *Specification of the material and the number of material points per element in the rainfall infiltration problem.*

13.3.1.4.1.3 Defining initial conditions

The K_0-procedure requires the specification of the soil and the phreatic initial surfaces (in this case lines since the model is in 2D). As explained in 13.2.1.4.1.3 assign the Soil Surface condition to the lines corresponding to the slope profile. Figure 13.28 shows the correct assignment.

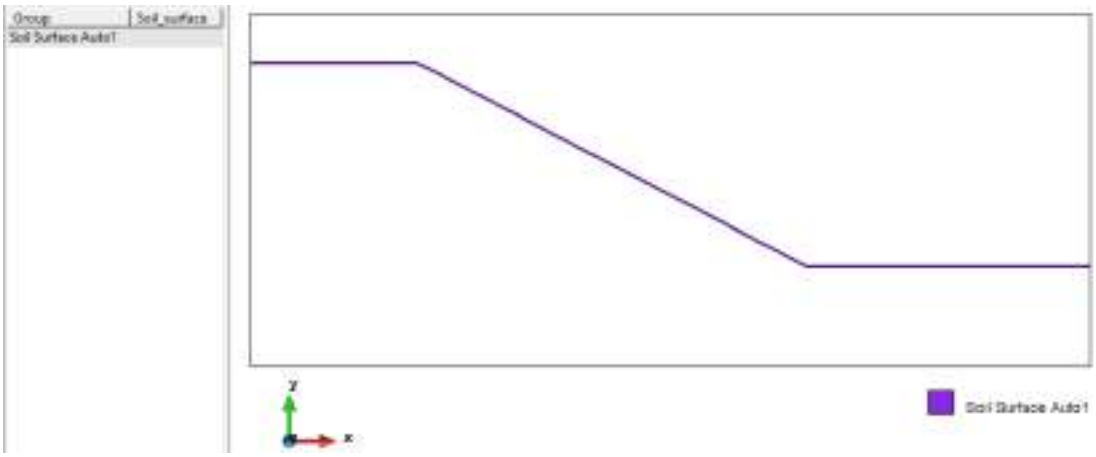


Figure 13.28: *Checking the initial soil surface correct assignment for the rainfall infiltration model.*

In this case the initial phreatic level is defined by specifying the coordinates of points corresponding to the location of the phreatic surface. This option can be found in the Anura3D tree menu under <Initial conditions → Stress initialization → General conditions → Phreatic Surface>. Double click on *Add water table 1*, insert the x and -y coordinates as indicated in Figure 13.29 and click on "OK". This feature allows the assignment of an initial phreatic level to one or more materials. In this example the initial phreatic level is assigned to the slope, thus, fill in the options as shown in Figure 13.30.

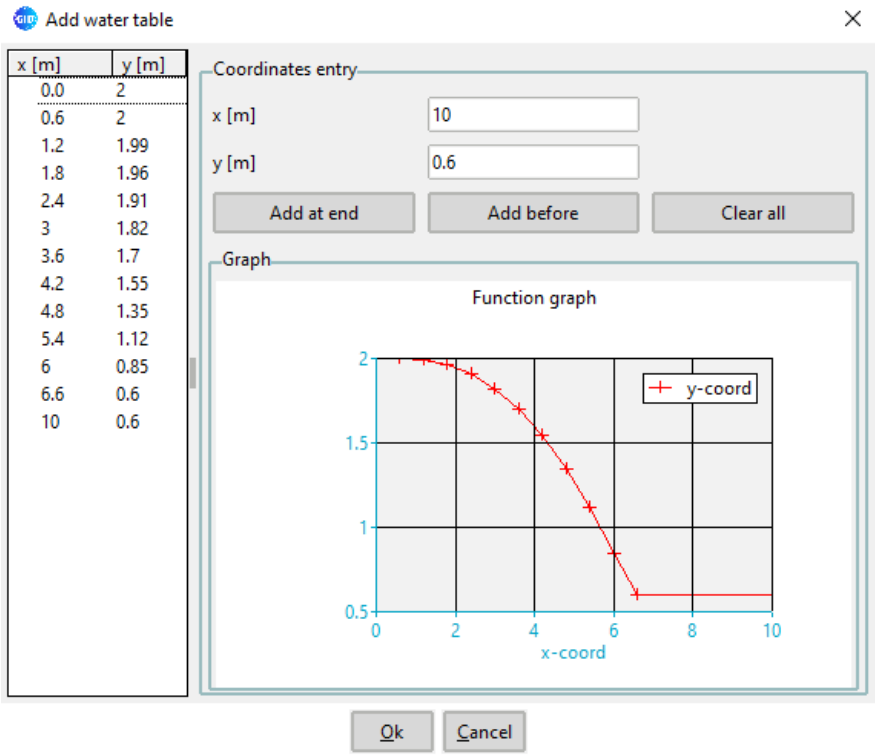


Figure 13.29: *Specify the coordinates of the initial phreatic surface.*

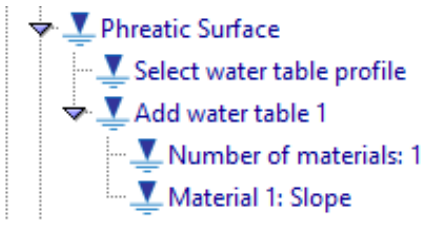


Figure 13.30: *Assign phreatic surface to the slope material for the rainfall infiltration problem.*

13.3.1.4.1.4 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying fixities

Since in this example a fully coupled two-phase analysis is performed, solid and liquid fixities must be assigned.

The solid displacement at the boundaries will be fixed as follows:

(1) Left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;
(4) Bottom side: full fixities in x- and y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 13.31;

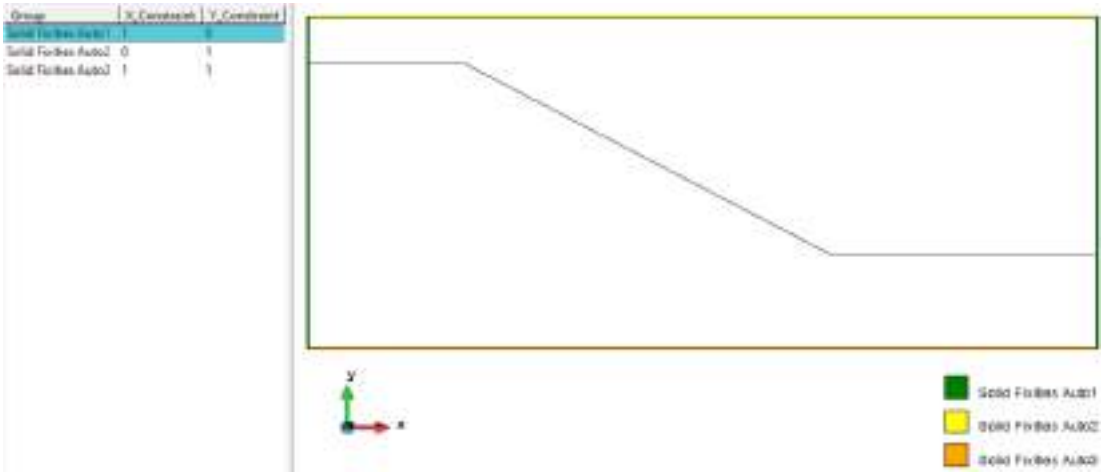


Figure 13.31: *Solid fixities for the rainfall infiltration problem.*

The liquid displacement at the boundaries will be fixed as follows:

(1) Upper lines on left side: horizontal fixity in x-direction;
(2) Right side: horizontal fixity in x-direction;
(3) Top side: vertical fixity in y-direction;
(4) Bottom side: vertical fixity in y-direction;
(5) Check if the x- and y- fixities are assigned correctly, see Figure 13.32;

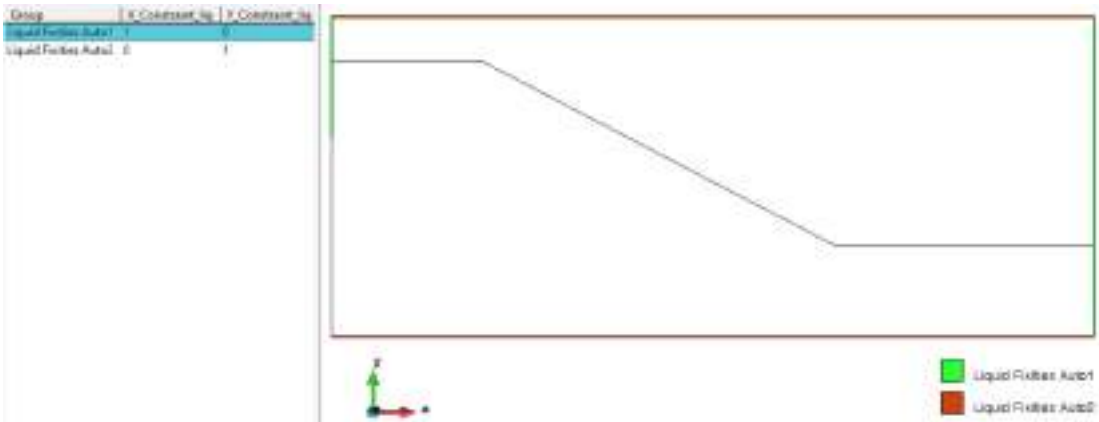


Figure 13.32: *Liquid fixities for the rainfall infiltration problem.*

Applying loading conditions

In order to reproduce a water level at the left-end, a liquid pressure linearly increasing with depth (hydrostatic pressure distribution) is applied on the left-bottom line of the model.

Choose *Loading Conditions*, double click on *Liquid Pressure* and introduce the values shown in Figure 13.33).

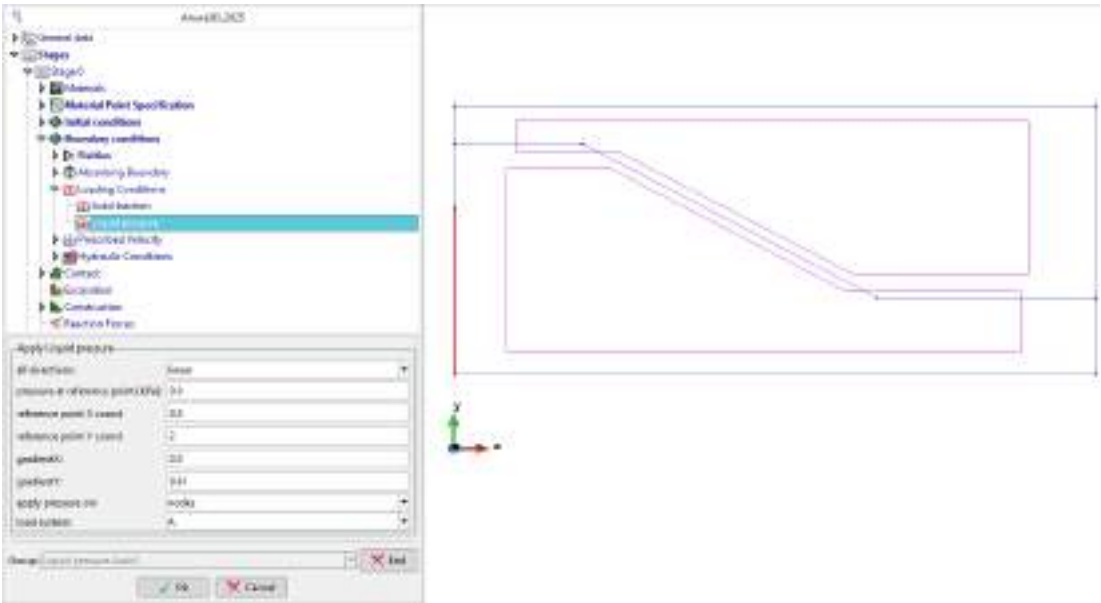


Figure 13.33: *Applied liquid pressure at the left-bottom line of the slope.*

Applying hydraulic boundary conditions

The hydraulic boundary conditions can be assigned in GiD as explained in 13.2.1.4.1.4. During the initialization phase, the slope surface acts as a potential seepage face. Therefore, specify the coordinates of minimum and maximum points of the rectangular area where the conditions are supposed to act, as shown in Figure 13.34.

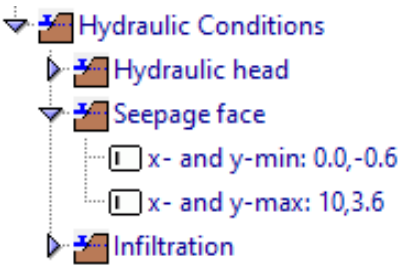


Figure 13.34: *Specify the coordinates defining the seepage face potential area for the rainfall infiltration problem.*

13.3.1.4.1.5 Defining calculation data

Figure 13.35 shows the calculation data of *Stage 0*.

Computation method

The option *MPM - mixed integration* should be selected.

Calculation step data

Gravity loading is performed in one calculation step. The end of the step is determined by the convergence criteria. A Courant number of 0.98 is advised to ensure the stability of the solution. Set *number of calculation steps [-]* to 1.

Gravity load

Gravity is applied from the beginning, therefore choose *apply gravity load - stepwise* and set the multipliers for gravity loading as 1.0 (initial and final value). The gravity multipliers will be maintained constant throughout the calculation.

Liquid pressure
The linear distribution of liquid pressure is applied at the left of the model, thus choose *apply liquid pressure - linear* and set the multipliers for gravity loading as 1.0 (initial and final value).

Quasi-static convergence
The end of the first load step is determined by convergence criteria, which corresponds to a quasi-static equilibrium condition. Set the *QUASI-STATIC CONVERGENCE* to *apply convergence criteria*. The convergence criteria are the tolerated error of the kinetic energy, the tolerated error of the out-of-balance force, or the maximum number of time steps. In this example, in order to reduce the computational time, use a maximum number of time steps equal to 2500.

Mass scaling
In this example the mass scaling is used to speed up the computation and a value of 80 is assigned to the mass scaling factor.

Homogeneous local damping
A local damping coefficient for all active elements of 0.05 is applied.

Strain smoothing
The strain smoothing algorithm will be switch on to minimise the kinematic locking.

Liquid pressure increment smoothing
The liquid pressure smoothing will be switch on to minimise pressure oscillations during the post-failure phase.

Default values
All other calculation options are not used in this example and the provided default values can be kept.

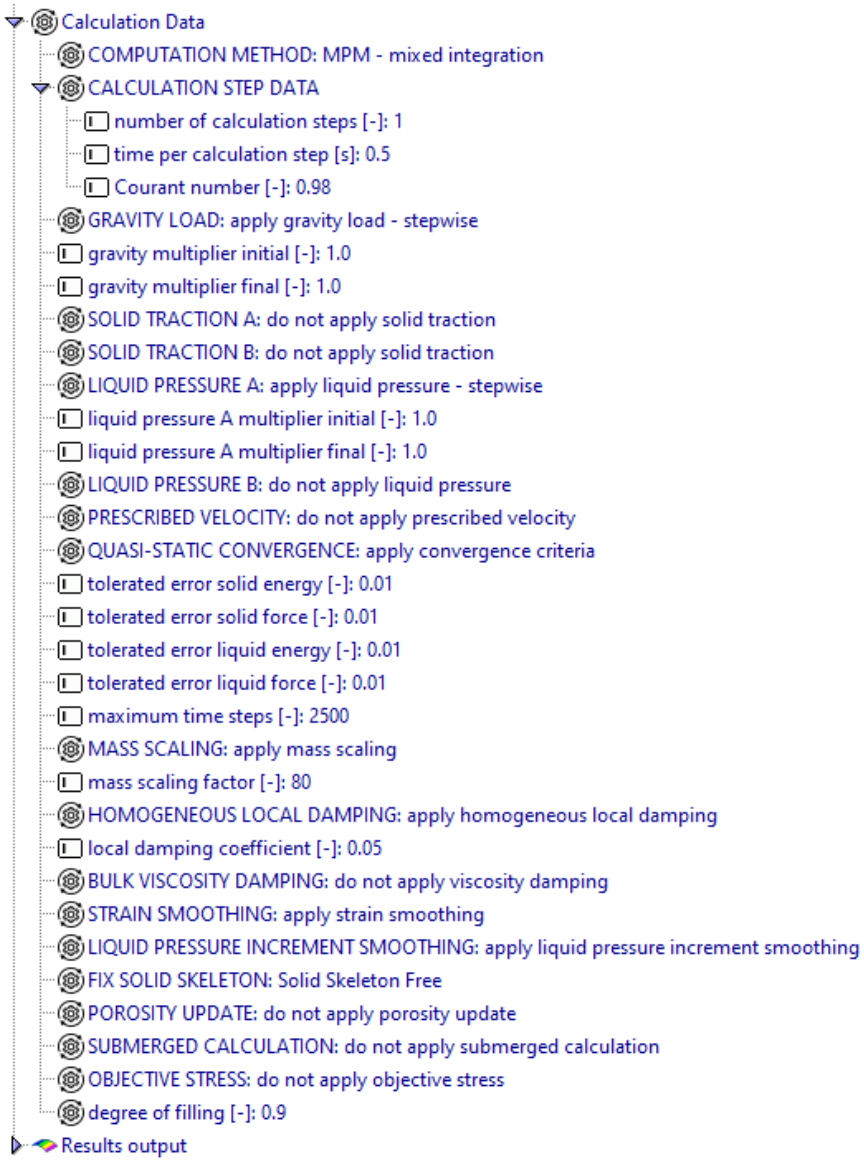


Figure 13.35: *Dialogue box to define calculation parameters in GiD for Stage 0 of the rainfall infiltration problem.*

13.3.1.4.2 Stage 1 - Slope failure

To create *Stage 1*, right-click on *Stage 0* and select *copy (with assigned groups)*. This will copy all the conditions from the previous stage to the current one, allowing to specify only the differences required at this stage of calculation.

13.3.1.4.2.1 *Updating material properties*

To update the material properties, first set the option *Update Materials* to *apply*. Then, modify the intrinsic permeability of the liquid to 6-e-12 m^2 and the effective cohesion to 1 kPa (Figure 13.36)

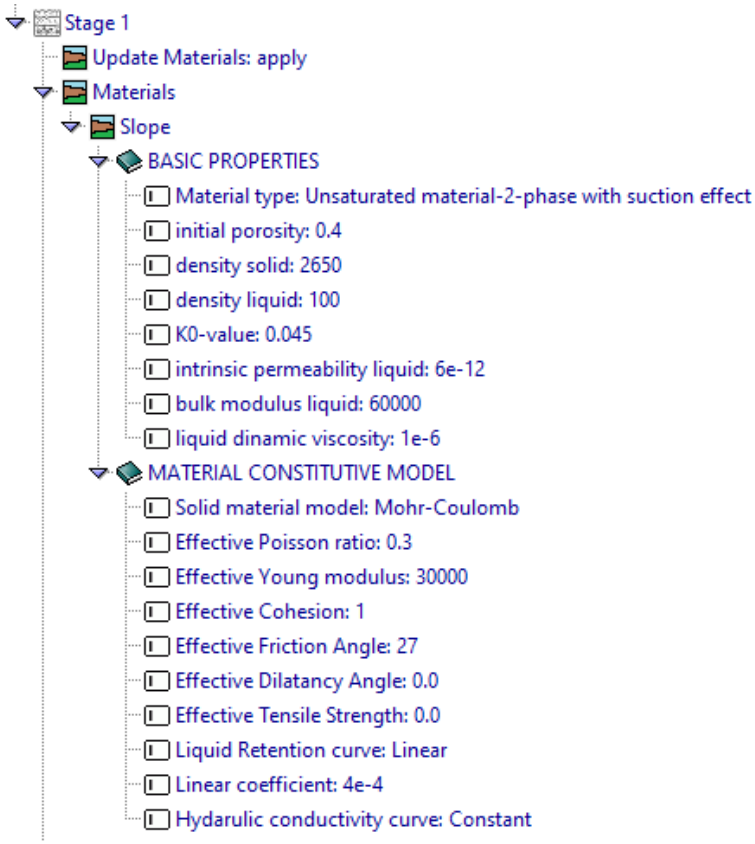


Figure 13.36: *Defining material properties in GiD for stage 1 of the rainfall infiltration problem.*

13.3.1.4.2.2 *Defining hydraulic boundary conditions*

In this stage, an infiltration rate of $2 \cdot 10^{-4}$m/s is applied to simulate a heavy rainfall. To set up the infiltration condition, a rectangular area must be defined.

Specify the coordinates of the minimum and maximum points that delimit the rectangular area where the condition will apply. Then, specify the infiltration rate value in y-direction, as shown in Figure 13.37.

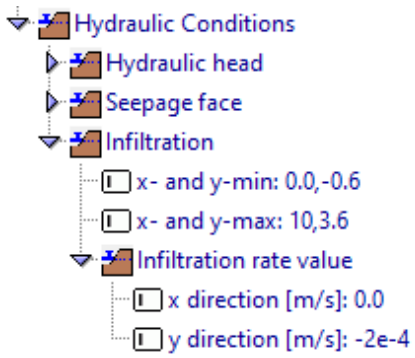


Figure 13.37: *Specify the coordinates defining the infiltration area for the rainfall infiltration problem.*

13.3.1.4.2.3 Defining calculation data

Figure 13.38 shows the calculation data of *Stage 1*.

Calculation step data
This stage of the analysis will last up to calculation step 22 and each calculation step will last 1s. Set *number of calculation steps [-]* to 22 and *time per calculation step [s]* to 1.

Quasi-static convergence
This stage is performed trough a dynamic calculation. Thus, set the *QUASI-STATIC CONVERGENCE* to *do not apply convergence criteria*.

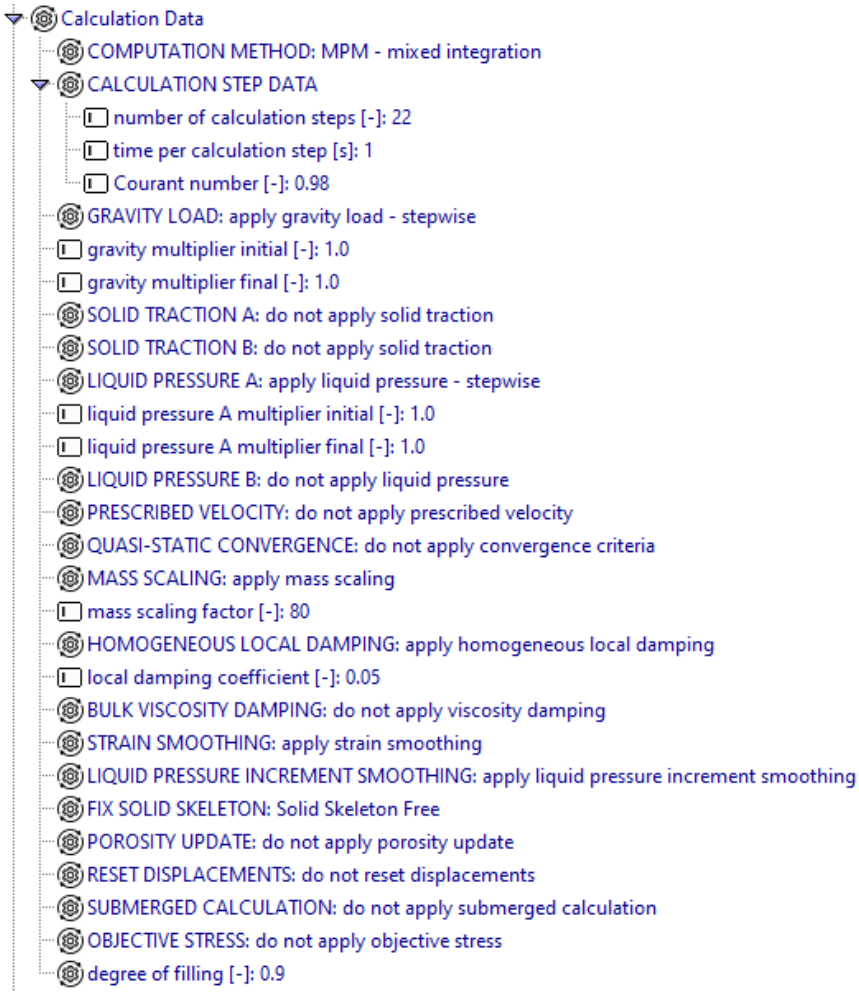


Figure 13.38: *Dialogue box to define calculation parameters in GiD for Stage 1 of the rainfall infiltration problem.*

13.3.1.4.3 Stage 2 - Slope stabilization

To create *Stage 2*, right-click on *Stage 1* and select *copy (with assigned groups)*. In this stage, since the material properties remain unchanged from the previous stage, set the option *Update Materials* to *do not apply*.

13.3.1.4.3.1 Unassign infiltration boundary condition

At this stage, rainfall is deactivated, and the seepage face is re-established. Set the coordinates of the points defining the infiltration area to "0.0" and assign the infiltration rate value in -y direction to "0.0", as shown in Figure 13.39.

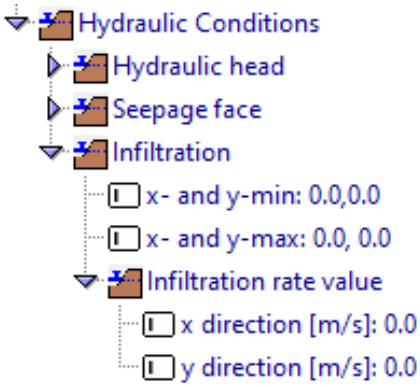


Figure 13.39: *Unassign the infiltration condition.*

13.3.1.4.3.2 Defining calculation data

Figure 13.40 shows the calculation data of *Stage 2*.

Calculation step data
This stage of the analysis will last up to calculation step 32 and each load step will last 1s. Set *number of calculation steps [-]* to 32 and *time per calculation step [s]* to 1.

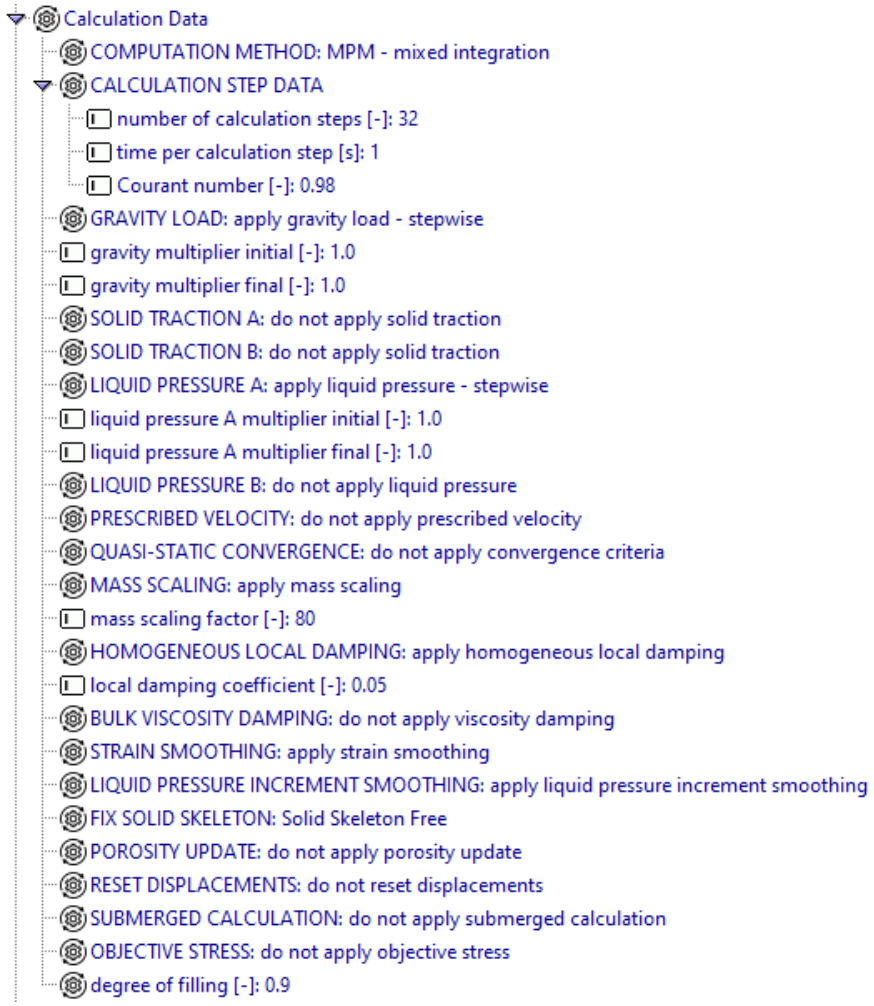


Figure 13.40: *Dialogue box to define calculation parameters in GiD for Stage 1 of the rainfall infiltration problem.*

13.3.1.5 Creating mesh

To generate the mesh follow the steps given in Section 6.1.5. As in the previous example, an unstructured mesh is used with the specifications detailed in 13.2.1.5. Figure 13.41 shows the generated mesh made of 571 triangular elements and 322 nodes.

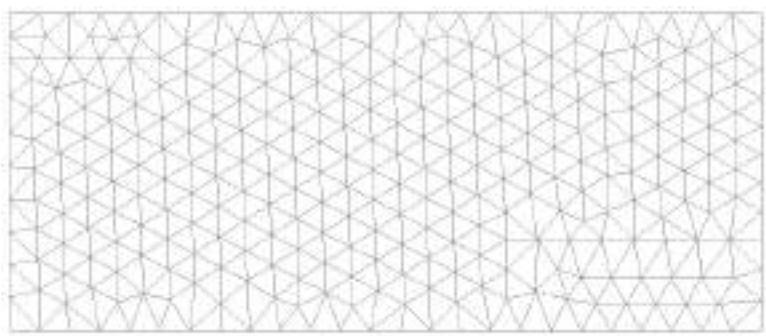


Figure 13.41: *Computational mesh of the rainfall infiltration problem.*

13.3.2 Performing calculation in Anura3D

To start the calculation go to <Anura3D> → <Calculate>. The three stages will be computed automatically.

The generated input files for Stage 0 of Anura3D simulation can be found in the newly created folder `RainfallInfiltration.A3D`, within the directory specified earlier by the user (Sec. 13.2.1.1):

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.GOM_stage0
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.OPD_stage0
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.CPS_001
```

Once calculation of Stage 0 is finished the necessary files for Stage 1 are generated automatically:

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.GOM_stage1
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.OPD_stage1
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.CPS_002
```

The same will occurr for Stage 2 after Stage 1 is calculated:

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.GOM_stage2
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.OPD_stage2
```

```
D:\..\RainfallInfiltration.A3D\RainfallInfiltration.CPS_023
```

Alternatively, if you wish to calculate only specific stages, click on the *Stage Manager* icon, adjust their status by setting them to 'Calculate' or 'Not to Calculate,' and then click on *Calculate* to proceed (Figure 13.42).

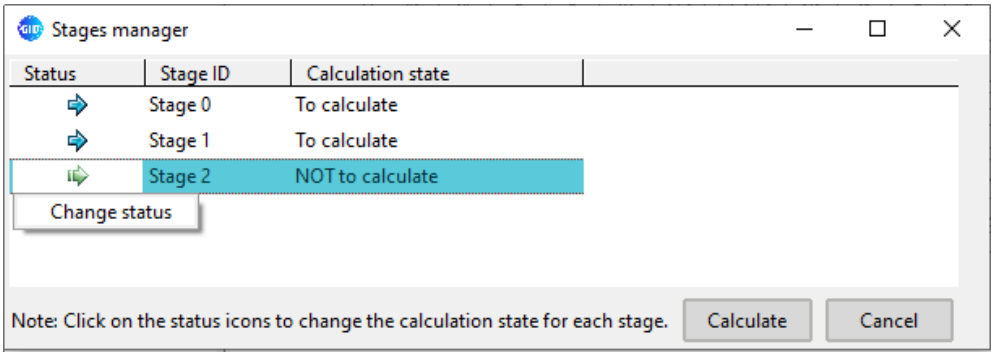


Figure 13.42: *'Stage Manager' window for the rainfall infiltration problem.*

13.3.3 Visualisation of results in ParaView

The recommended way to visualise Anura3D results is by using the ParaView software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results.

Figure 13.43 shows the contour of liquid pressure (a) and total vertical stress (b) at the end of load step 1. The liquid pressure are positive for suction. Total stress account for slope material weight plus the water weight above the soil surface.

After stress initialisation, rainfall infiltration is applied at the top surface inducing the slope failure. Total displacements at the end of the simulation are shown in Figure 13.44.

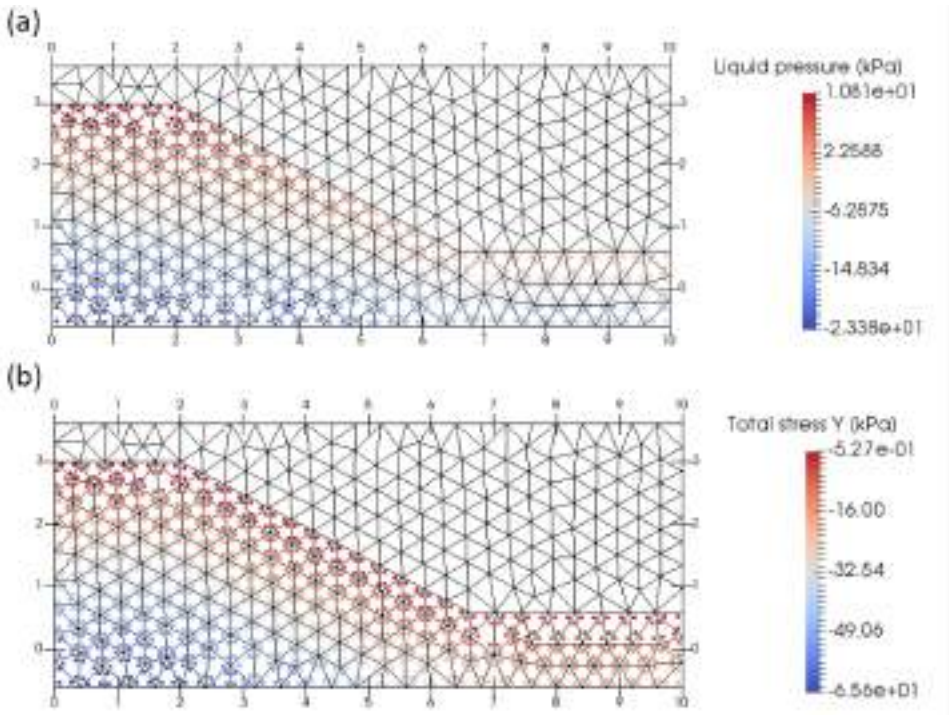


Figure 13.43: *Liquid pressure (a) and total vertical stress (b) at the end of Stage 0.*

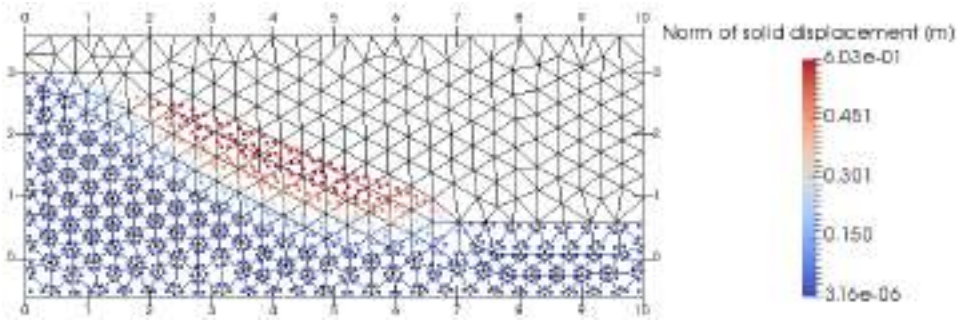


Figure 13.44: *Norm of solid displacement at the end of the simulation.*

13.3.4 Variation on triggering action: application of zero suction

A variation of the rainfall infiltration problem is proposed in this section.

In Stage 1, slope instability is not induced by activating the infiltration feature but by applying a suction equal to 0 at the slope surface.

Since in Anura3D the default value of liquid pressure on the boundaries is zero, is sufficient to deactivate the seepage face condition.

Definition of Stage 2 remains unchanged from the reference rainfall infiltration example.

The final displacements contour at load step 32 is reported in Figure 13.45.

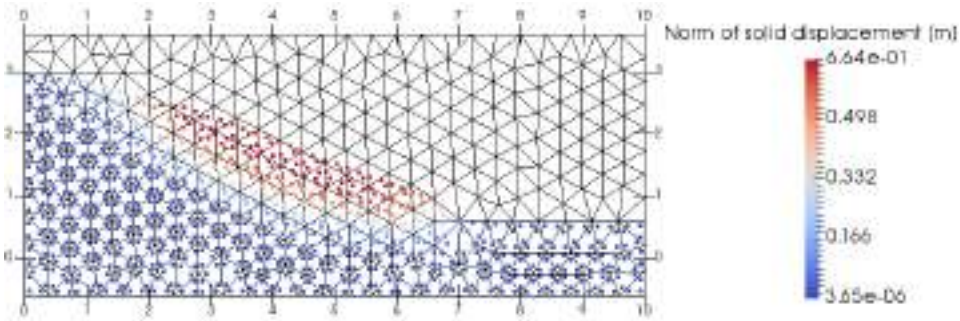


Figure 13.45: *Norm of solid displacements at the end of the simulation for the case with zero suction at the slope surface.*

13.4 Stress initialization from file

In the previous sections of this chapter, the initial stress state is generated using K0-procedure and gravity loading; in many cases this procedure may be computationally expensive. For example, in many rainfall or seepage-induced slope failures, the initiation phase may take several days, when only very limited displacement are observed. Usually, traditional FEM analyses are computational less expensive than MPM simulations in the initiation phase, thus it may be convenient to initialize the stress state for the MPM analyses using the results of a previous calculation. In this section, the option of initializing the material point stress state from a text file is explained. The text file must contain the values of stress and pore pressure at given points; it can be obtained by exporting the results from a previous calculation performed with a different software. The location of the give points does not need to coincide with the position of the material points or nodes of the computational mesh in MPM because Anura3D will interpolate automatically the stress at material point using a kernel interpolation technique. More details can be found in [**?**].

In the following, the same example of the previous section (Sec 13.3) is solved replacing the gravity loading stage with a stress initialization from file.

13.4.1 Creating input data

The geometry, material, discretization and boundary conditions are the same of the previous case, see Sec 13.3.1.1. Save the project as RainfallInfiltration-mapping.gid.

13.4.1.1 Preparation of input file for stress initialization

In this example, the results of a FEM analysis is used to create the input file for material point stress initialization in Anura3D. The simulation is 2D-plane strain, and the two-phase single-point formulation is used, thus the required variables are:

- the coordinates of the points in which the stress state is known: X[m] and Y[m];
- the entries of the total stress tensor at these points: σ_{xx} [kPa], σ_{yy} [kPa], σxy [kPa];
- the pore water pressure at these points: Pwp [kPa]

for a 3D analyses the coordinate Z and the stress σzz, σxz, σyz are also necessary. Pore water pressure is not necessary in simulations with dry material. Effective stresses are calculated automatically applying the principle of effective stress; degree of saturation and hydraulic conductivity are calculated from suction using the soil water retention curve and the hydraulic conductivity curve.

A reference value called "SmoothingLength[m]" is required in the last column. This is used as a reference length in the cubic spline function applied in Anura3D for kernel interpolation technique. An appropriate value is 90% of the average distance between the given points. If the given points are nodes of the mesh of a previous FE calculation, than the smoothing length can be 90% of the average element size. The mapping procedure is very robust, but the best results are obtained when the distribution of given points is similar to the distribution of element nodes in the MPM simulation.

The input file must be formatted as shown in Fig. 13.46

X[m]	Y[m]	Z[m]	sigx[kPa]	sigy[kPa]	sigxy[kPa]	Pwp[kPa]	SmoothLength[m]
10	-0.6	0	-15.38859706	-21.25294484	0.13558725	-10.05151020	0.3
10	-0.3	0	-13.11894043	-17.63903731	0.123536742	-8.741256436	0
10	0	0	-9.433797852	-11.92594531	0.10115116	-5.669439453	0
10	0.3	0	-5.827570213	-6.182984785	0.072063712	-3.148677783	0
10	0.6	0	-3.114430941	-3.460770264	0.056461135	-1.101267944	0
0	-0.6	0	-41.02462813	-60.706125	0.300821991	-23.82550643	0

Figure 13.46: *Example of text file for mapping of stresses for initialization purposes (.MAPF).*

The order of the columns is not important but make sure you do not change the name of the headers or add extra spaces otherwise Anura3D will not recognize correctly the input and will give an error.

Note that initial strains and displacements are assumed to be 0, this is acceptable when the deformations occurred during the initialization phase are negligible compared with the post-failure displacements.

Save the file as *projectname*.MAPF (in this case RainfallInfiltration-mapping.MAPF)

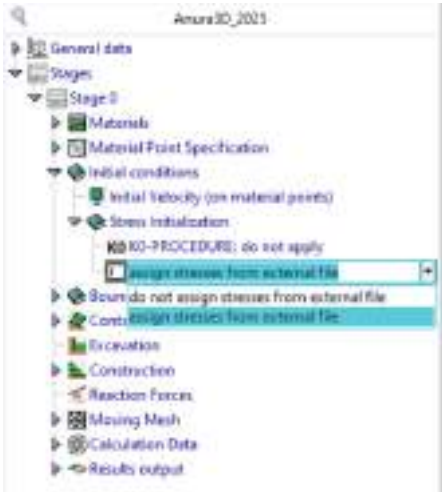


Figure 13.47: *View of Initial conditions menu.*

13.4.1.2 Defining calculation stages

In this example, only one stage of calculation is simulated, corresponding to the slope failure stage (*Stage 1* of the previous example), however the stabilization phase can also be included as in **??**. To do this, right-click on *Stage 2* and *Stage 1*, select <Delete>, and then modify *Stage 0* as outlined below:

Material properties
In this case, it is unnecessary to assign high cohesion and permeability values during the initialization phase; set the cohesion to 1kPa and the intrinsic permeability to $6e-12m^2/s$.

Initial conditions
First, deactivate the K_0 procedure by setting the <K0-procedure> option to <do not apply> in the <Initial conditions → Stress initialization> section. Next, to enable stress initialization from file, select the appropriate option under <Initial conditions → Stress Initialization → FROM FILE> as show in Figure 13.47.

Infiltration boundary conditions
Assign the infiltration condition as done in Sec **??**.

13.4.1.3 Defining calculation data

Computation method
The option *MPM - mixed integration* should be selected.

Calculation step data
Set 21 load steps (number of calculation steps = 21), each will be 1s long (time per calculation step [s]=1.0).

Gravity data
Use standard gravity value and unit vector down: $\mathbf{g} = 9.81 * (0, -1.0, 0)$

Gravity load
Gravity is applied from the beginning, therefore choose *apply gravity load - stepwise* and set

the multipliers for gravity loading as 1.0 (initial and final value). The gravity multipliers will be maintained constant throughout the calculation.

Liquid pressure
From the beginning of the calculation the linear distribution of liquid pressure is applied at the left of the model, thus choose *apply liquid pressure - stepwise* and set the multipliers for gravity loading as 1.0 (initial and final value).

Quasi-static convergence
With the stress initialization from file this is not necessary. Set the *QUASI-STATIC CONVERGENCE* to *do not apply convergence criteria*.

Mass scaling
In this example the mass scaling is used to speed up the computation and a value of 80 is assigned to the mass scaling factor.

Homogeneous local damping
A local damping coefficient for all active elements of 0.05 is applied.

Strain smoothing
The strain smoothing algorithm will be switch on to minimise the kinematic locking.

Liquid pressure increment smoothing
The liquid pressure smoothing will be switch on to minimise pressure oscillations during the post-failure phase.

Default values
All other calculation options are not used in this example and the provided default values can be kept.

13.4.2 Performing calculation in Anura3D

To start the calculation go to <Anura3D> → <Calculate>.

13.4.3 Visualisation of results in ParaView

The recommended way to visualise Anura3D results is by using the ParaView software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results.

Figure 13.48 shows the contour of liquid pressure (top) and total vertical stress (bottom) at the very first time step, which are the results of the stress initialization procedure that uses the input file prepared as explained in Sec. 13.4.1.1. It is always recommended to check that this output matches the expectations to verify that the settings of the initialization procedure are correct.

After stress initialization, rainfall infiltration is applied at the top surface inducing the slope failure. Total displacements at the end of the simulation are shown in Figure 13.49.

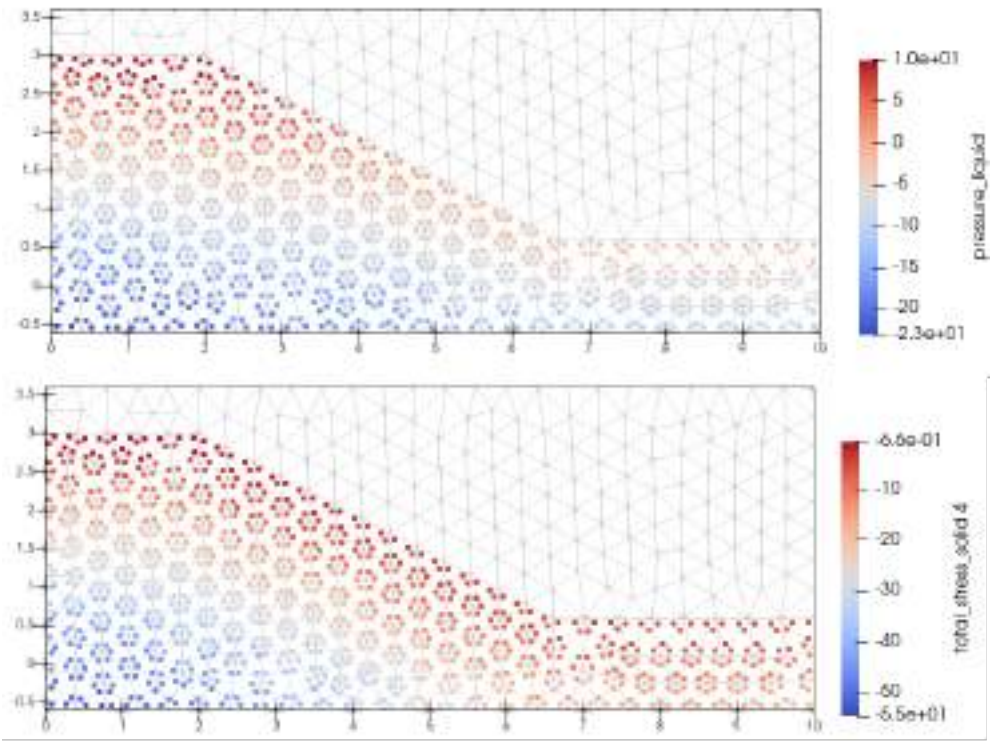


Figure 13.48: *Initial pore pressure and total vertical stress at the beginning of the simulation.*

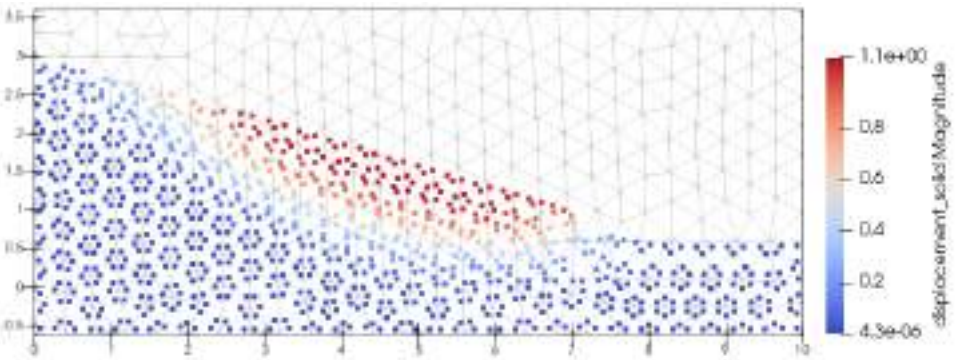


Figure 13.49: *Total displacement at the end of the simulation.*

References

[1] E.J. Fern, A. Rohe, K. Soga, and E.E. Alonso. *The Material Point Method for Geotechnical Engineering: A Practical Guide*. CRC Press, 2019.

[2] I.K.J. Al-Kafaji. *Formulation of a dynamic material point method (MPM) for geomechanical problems*. PhD Thesis, Institut für Geotechnik der Universität Stuttgart, Germany, 2013.

[3] A. Yerro. *MPM modelling of landslides in brittle and unsaturated soils*. PhD Thesis, Universitat Politècnica de Catalunya (UPC Barcelona), Spain, 2015.

[4] L. Beuth. *Formulation and application of a quasi-static material point method*. PhD Thesis, Institut für Geotechnik der Universität Stuttgart, Germany, 2012.

[5] Francesca Ceccato, Lars Beuth, and Paolo Simonini. Analysis of piezocone penetration under different drainage conditions with the two-phase material point method. *Journal of Geotechnical and Geoenvironmental Engineering*, 142(12):04016066, 2016.

[6] F. Ceccato, L. Beuth, P.A. Vermeer, and P. Simonini. Two-phase material point method applied to the study of cone penetration. *Computers and Geotechnics*, 80:440–452., 2016. doi: 10.1016/j.compgeo.2016.03.003.

[7] Luis Zambrano-Cruzatty and Alba Yerro. Numerical simulation of a free fall penetrometer deployment using the material point method. *Soils and Foundations*, 60(3):668–682, June 2020. ISSN 0038-0806. doi: 10.1016/j.sandf.2020.04.002. URL http://www.sciencedirect.com/science/article/pii/S0038080620336179.

[8] H. Tresca. Mémoire sur l'écoulement des corps solides soumis à de fortes pressions. *Rendus hebdomadaires des séances de l'Académie des Sciences*, 59:754–758, 1864.

[9] Francesca Ceccato, Alba Yerro, Veronica Girardi, and Paolo Simonini. Two-phase dynamic mpm formulation for unsaturated soil. *Computers and Geotechnics*, 129:103876, 2021.

A Shallow foundation in 3D-cylindrical coordinates

Authors: Francesca Ceccato

This appendix solves the shallow foundation problem of Chapter 8 in 3D-cylindrical coordinates. We will simulate a circular footing with a diameter of 4m. This can be done with a 2D-axisymmetric model or with a 3D model using cylindrical coordinates.

The 2D-axisymmetric model is similar to the case presented in chapter 8, but the option "'2D-axisymmetric"' should be selected in the Calculation data. The 3D case is illustrated in the following. The geometry of the problem is similar to Chapter 8, but a 3D model will be used in this case. In order to reduce the computational cost, only a 30 degrees-wedge of a circular footing is modelled. The foundation is elastic and the soil is modelled with the Mohr-Coulomb model. The properties for both materials are given in Table 8.2.

A load is applied at the top of the foundation, and its magnitude increases progressively by means of the *load multipliers*. The use of *Moving mesh* feature in 3D problems is illustrated.

A.1 Creating input data

The geometry, discretization, material properties, and all other input parameters will be defined using the GiD pre-pocessor. This follows a very similar procedure as described in Section 8.2. Here only the differences with respect to previous examples will be highlighted.

A.1.1 Defining problem dimensions

The calculation data are identical to those in Chapter 8.3.1, whith the only exception of the Dimension that should be "'3D:Axissymmetric"' (Fig. A.1).

A.1.2 Defining geometry

Start the GiD pre-processor following the steps as described in Section 3.2.1. Save the project in the desired directory on the hard disk with an appropriate name (e.g. "foundation-3Dcylindrical").

Create a 2D geometry following the instructions given in Section 8.2.3 considering the dimensions given in Table 8.1. We will create the 3D geometry applying a rotation as explained in the following and shown in Figure A.2:

(1) select Utilities>Copy
(2) choose Surfaces on Entity types
(3) choose Rotation on Transformation
(4) set 30 degrees angle
(5) select the bottom right point as a first point: (0,0,0)
(6) select the top right point as a second point: (0,13,0)
(7) select do extrude Volumes
(8) press Select and choose all the surfaces of the model
(9) press Escape and you will get the geometry shown in Figure A.3

Figure A.1: *Dialogue box to define calculation parameters in GiD for the shallow foundation problem in 3D - cylindrical coordinates.*

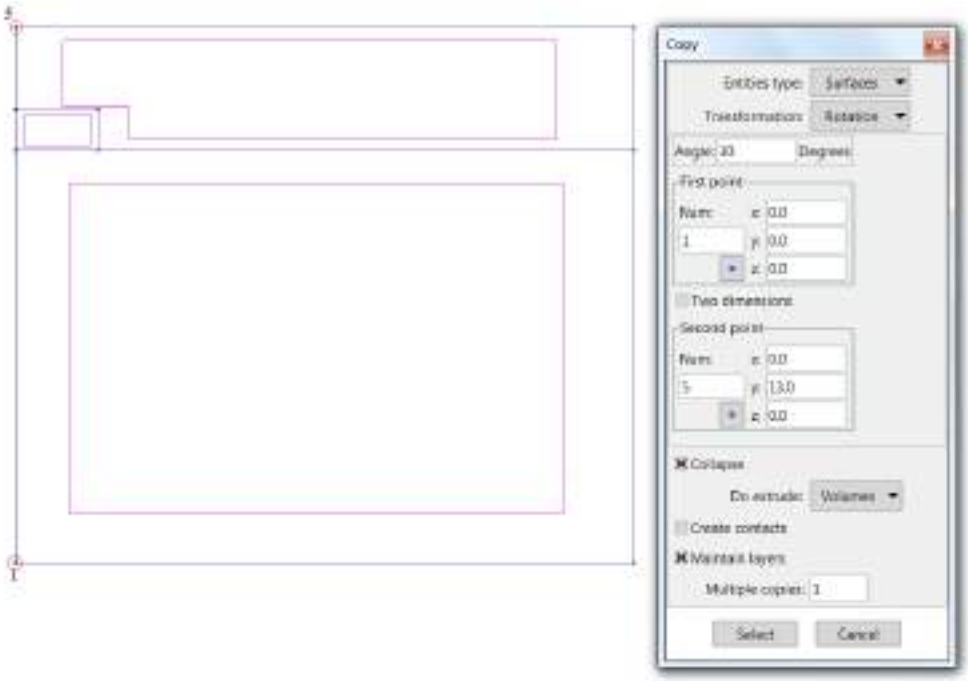


Figure A.2: *Definition of the geometry of circular footing.*

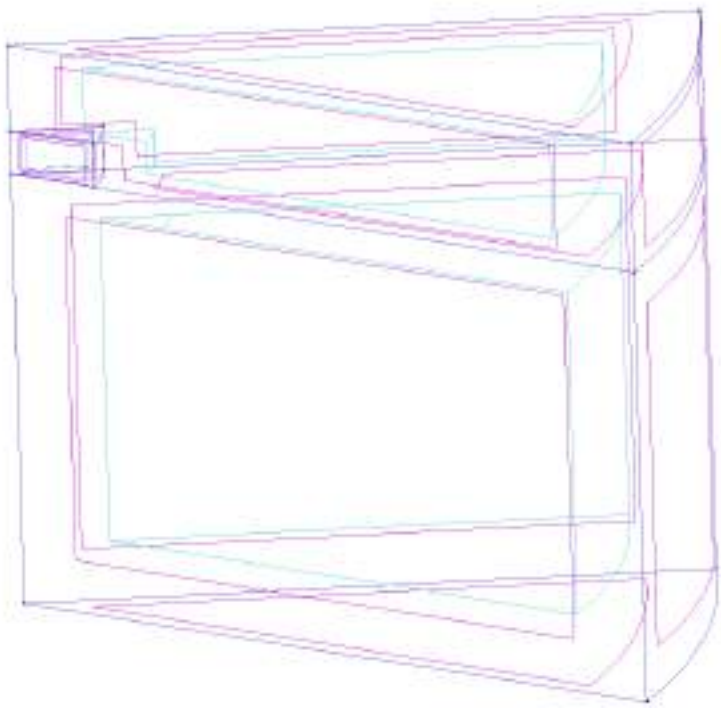


Figure A.3: *Geometry of circular footing.*

A.1.3 Defining material properties

The materials are the same of Chapter 8. The parameters are given in Table 8.2. Follow the instructions given in Section 3.2.5 to define each material and assign it to the corresponding volumes. Checking the assigned materials, it should look like shown in Figure A.4.

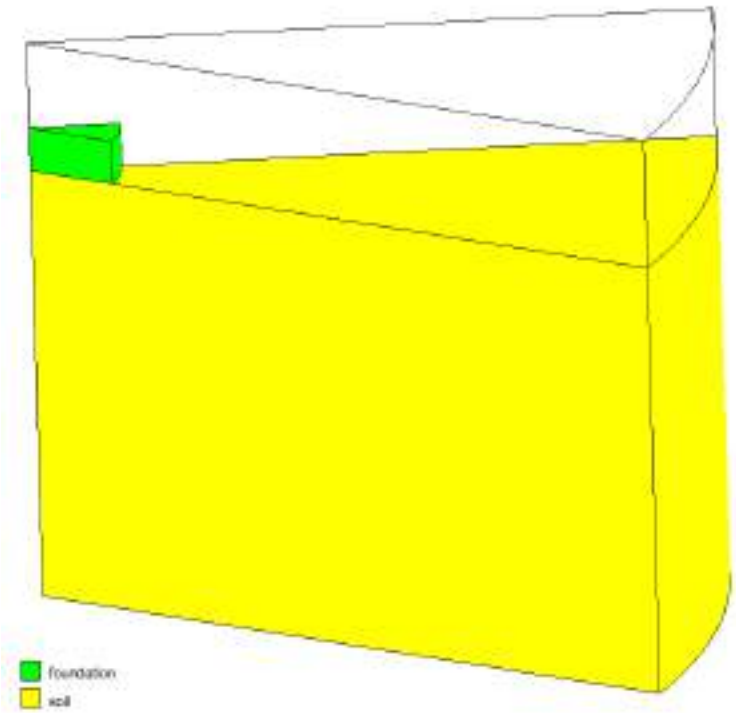


Figure A.4: *Materials of the shallow foundation problem in 3D.*

A.1.4 Specifying material points

Specify the initial material point distribution following instructions from Section 3.2.6. In this example choose 4 material points per element and assign it to the soil and the foundation volumes. Check if the material points are assigned correctly as shown in Figure A.5.

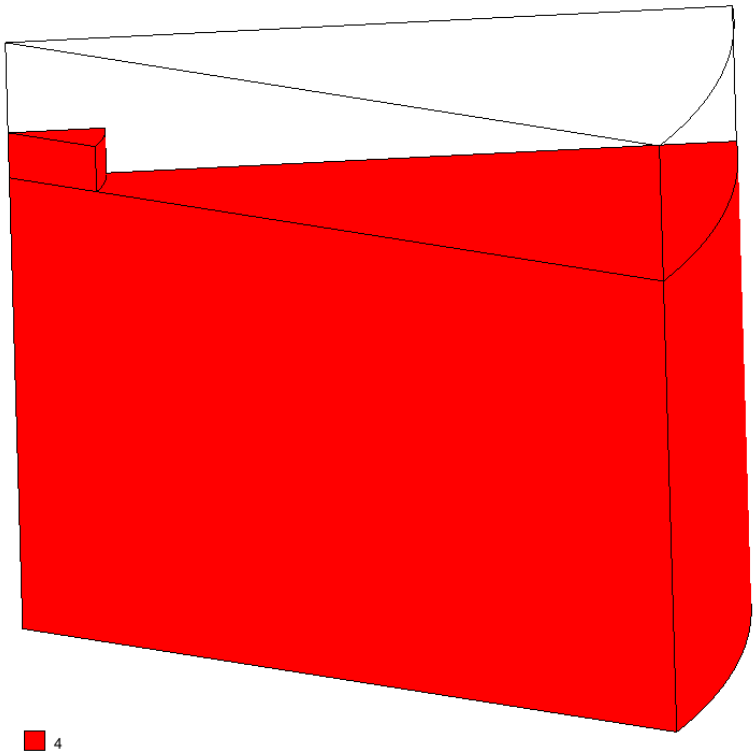


Figure A.5: *Specification of the number of material points per element in the shallow foundation problem.*

A.1.5 Defining boundary conditions

Define the boundary conditions following the steps as described in Section 3.2.7.

Applying fixities
Since in this example materials are in dry conditions, only the solid phase needs to be considered. Displacements in normal direction will be fixed at all boundaries; which means

(1) radial planes: fixity in θ-direction (third component);
(2) circumference surface: fixity in radial direction (first component);
(3) Top and bottom planes: vertical fixity in y-direction;

Check if the applied conditions are correct as shown in Figure A.6.

Applying loading conditions
A vertical load equal to –100 kPa will be applied in y-direction on the top of the foundation as shown in Figure A.7.

A.1.6 Defining a surface for computation of reaction forces

Follow the steps in Section 8.2.8 and select for output the surface at the interface between soil and foundation (see Figure A.8).

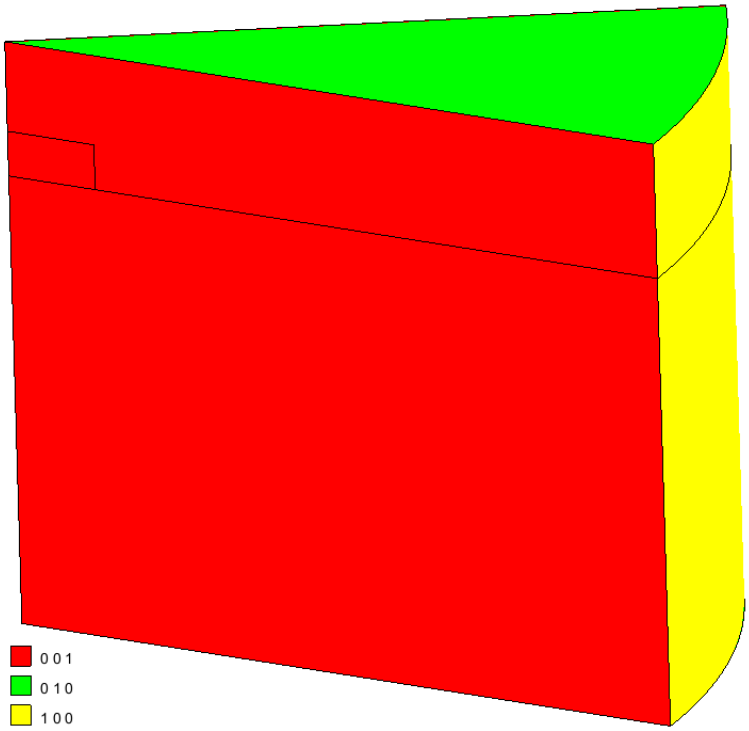


Figure A.6: *Solid fixities of the shallow foundation problem in GiD.*

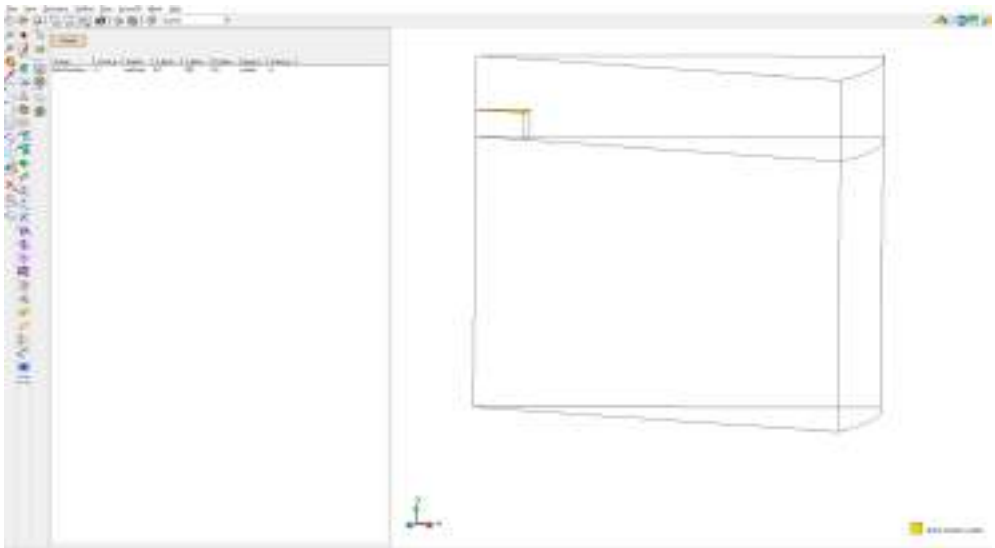


Figure A.7: *Applied load on top of the foundation at the material points.*

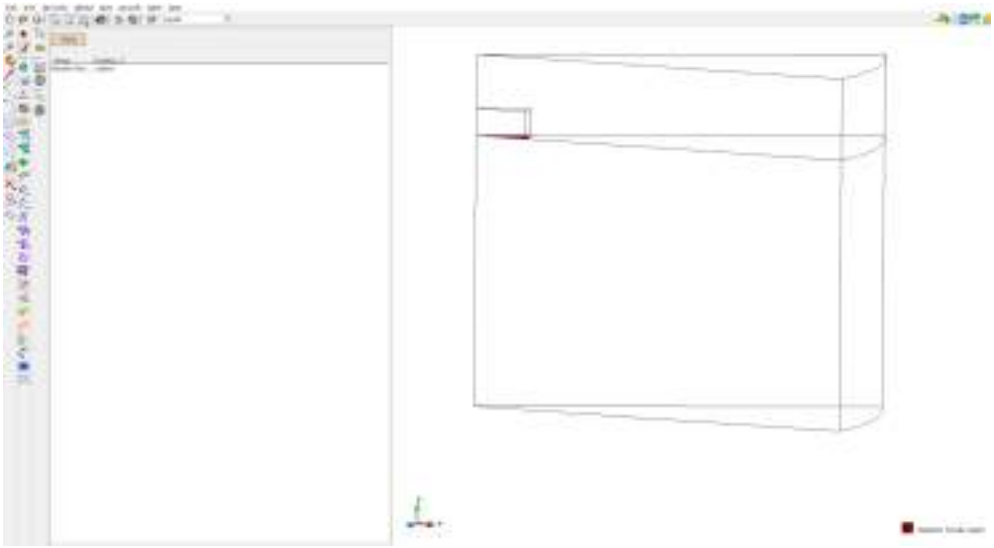


Figure A.8: *Output reation force.*

A.1.7 Defining the moving mesh

The moving mesh input parameters are defined similarly as in Chapter 8.2.9, but in 3D-cylindrical coordinate 6 corner nodes must be specified for each area, i.e. compressing mesh, extending mesh and moving mesh. The three domain areas must be connected with each other and all elements of the discretised domain must belong to either one of them.

The following steps have to be taken to define the moving mesh:

(1) Select "Compressing Mesh" from the drop down menu.

(2) Click <Select> and select the 6 corner nodes that delimit the zone that will be compressed by the movement of the moving mesh as shown in Figure A.9. Note that the volume delimited by these corner nodes must be a wedge, thus exactly 6 nodes must be selected of which 4 must be on an external boundary and 2 on the symmetry axis. Click <OK>. Alternative shapes of the compressing/extending/moving mesh are not allowed.

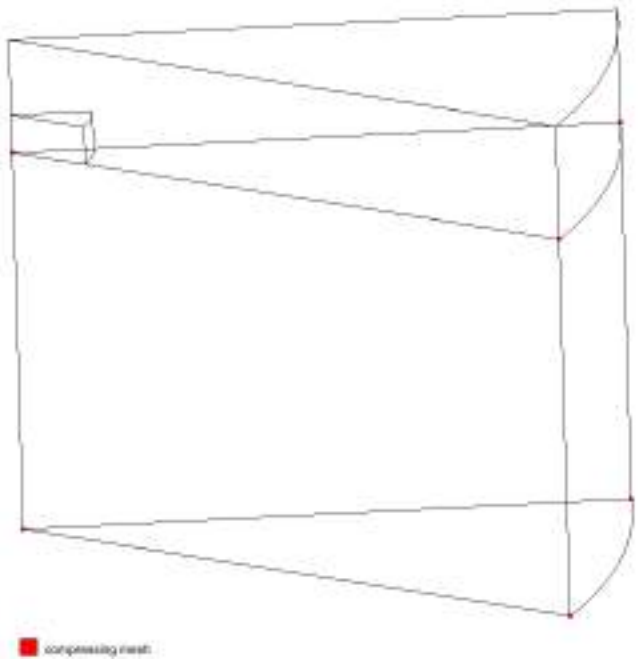


Figure A.9: *Definition of compressing mesh corner nodes.*

(3) In this example no "Extending Mesh" will be used, thus no corner nodes must be selected for this option.

(4) Select "Moving Mesh" from the drop down menu.

(5) Specify the direction of movement. In this example the foundation should move in vertical direction, thus choose "y-direction" next to "moving mesh direction".

(6) Click <Select> and select the 6 corner nodes that delimit the moving mesh as shown in Figure A.10. 3 of these nodes must coincide with the 2 nodes of the compressing mesh, i.e. the nodes of the top soil surface. The remaining 2 nodes must be on the free boundary, i.e. top surface. Click <OK>. Note that when using both compressing and extending mesh, it is necessary that exactly 3 nodes of the moving mesh coincide with 3 nodes of the compressing mesh and the remaining 3 nodes coincide with 3 nodes of the extending mesh.

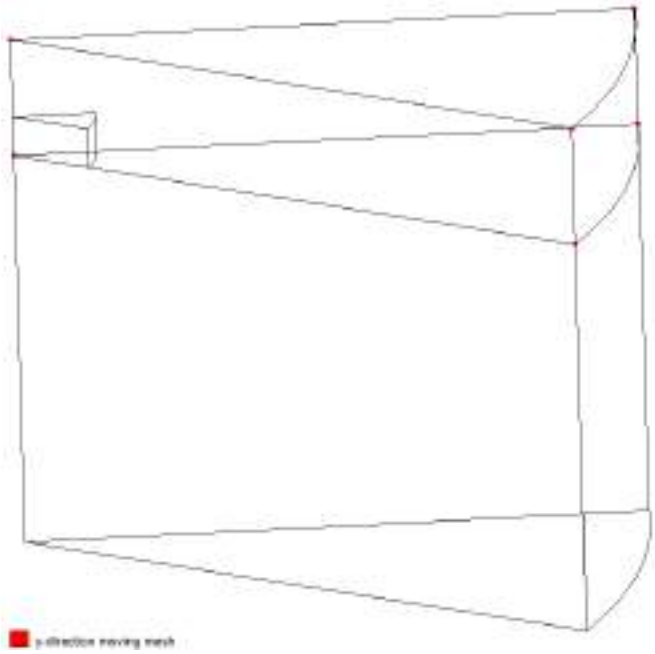


Figure A.10: *Definition of moving mesh corner nodes and direction of movement.*

(7) Define the reference material that will determine the displacement of the moving mesh.
(8) Click and select the structure surface, i.e. the foundation, as shown in Figure A.11.

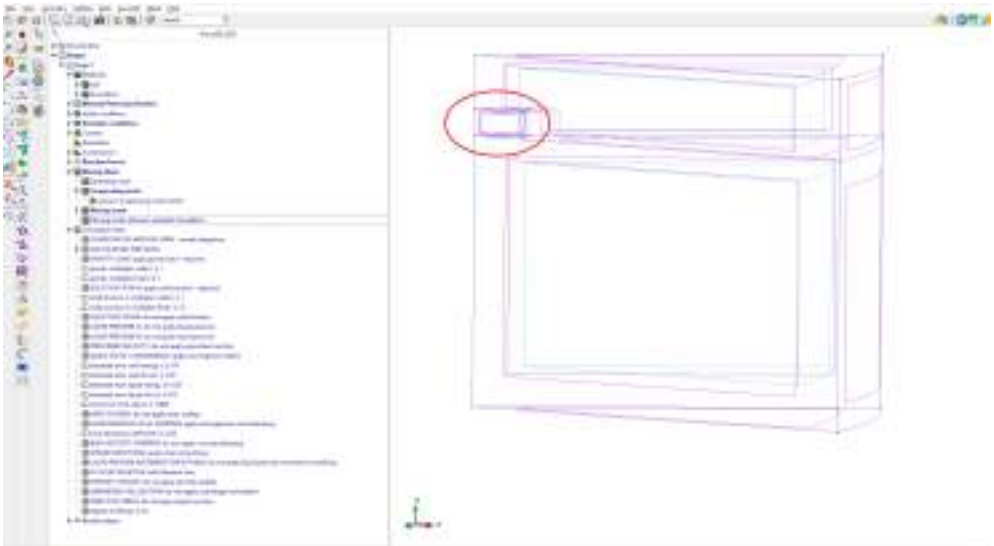


Figure A.11: *Definition of moving mesh reference material.*

It should be noted that the number of corner nodes necessary to identify correctly each moving mesh zone are:

- ◇ 4 corner nodes in 2D models
- ◇ 6 corner nodes in 3D-cylindircal models
- ◇ 8 corner nodes in 3D-cartesian models

A.1.8 Creating mesh

An unstructured mesh is used for this example. To generate the mesh follow the steps given in Section 6.1.5. For this example set the mean element size to "0.8", and assigned a refined size of 0.5 to the foundation bottom surface. The mesh should look as shown Figure A.12.

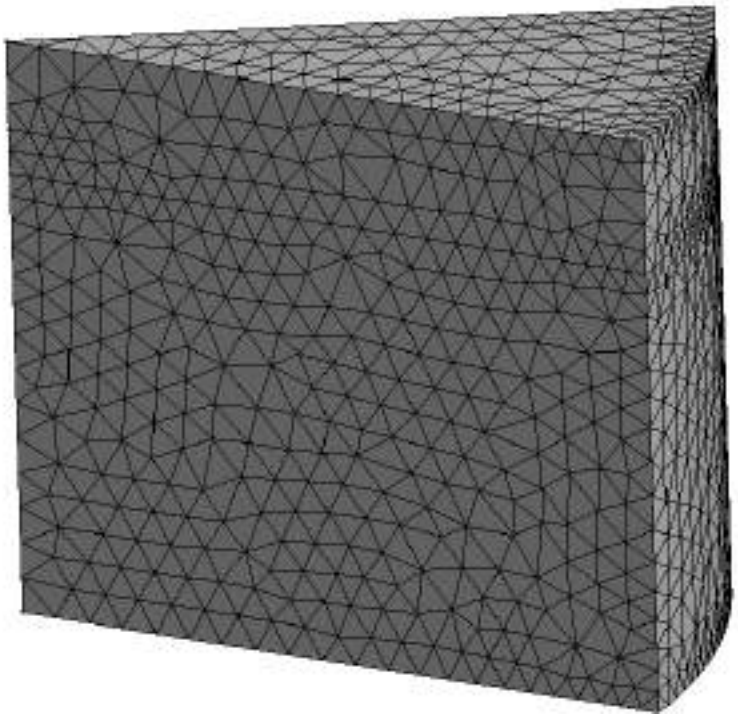


Figure A.12: *Computational mesh of the shallow foundation problem in 3D-cylindrical co-ordinates.*

A.1.9 Generating Input Files

After the geometry, material properties, boundary conditions, and calculation parameters are defined and the mesh is generated the input files for the Anura3D analysis, calculation should be performed. This can be simply done by clicking <Calculate> in Anura3D icon in taskbar. This would create relevant *GOM* and *CPS* file.

A.1.10 Performing calculation in Anura3D

The calculation can be performed as explained in Section 3.3.

A.1.11 Visualisation of results in ParaView

The recommended way to visualise Anura3D results is by using the ParaView software (see Sections 2.5 and 2.1.3). Instructions presented in Section 3.4 can be followed to open and analyse the results from the shallow foundation problem. Figure A.13 shows the evolution of vertical effective stress.

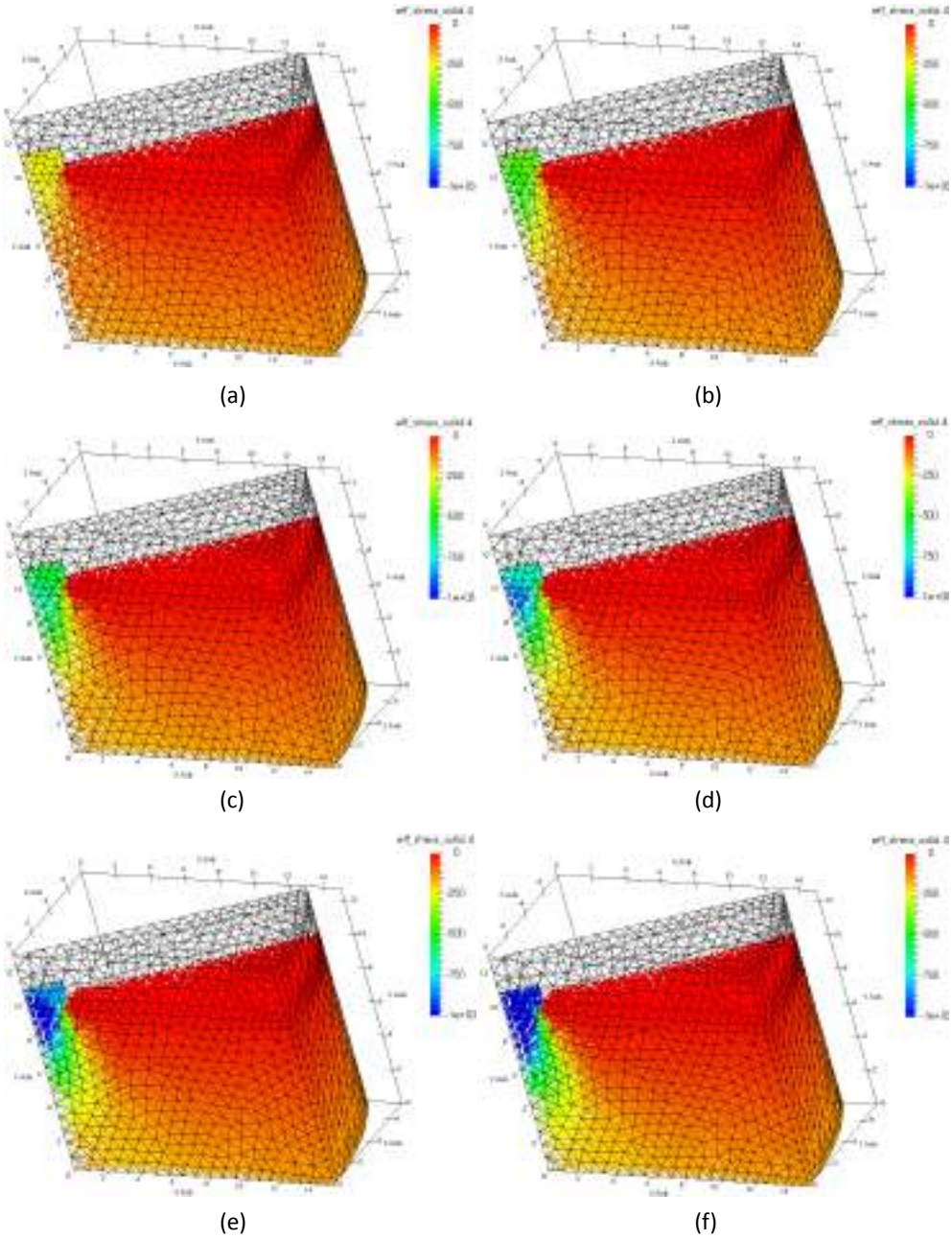


Figure A.13: *Vertical effective stress after: a) load step 1, b) load step 2, c) load step 3, d) load step 4, e) load step 5 and f) load step 6.*