

TABLE OF CONTENTS

Table of Figures	5
1. Introduction	15
2. Navigation	17
3. Application modules	25
3.1 Module: Part	25
3.2 Module: Property	26
3.3 Module: Assembly	27
3.4 Module: Step	27
3.5 Module: Interaction	27
3.6 Module: Load	28
3.7 Module: Mesh	29
3.8 Module: Optimization	29
3.9 Module: Job	29
3.10 Module: Visualization	30
3.11 Module: Sketch	30
4. 3D frame analysis	31
4.1 Keywords	31
4.2 Aims	31
4.3 Problem description	31
4.4 Solving steps	32
4.5 Results interpretation	58
4.6 Things to remember	60
5. 2D elastic continuous problem	61

5.1	Keywords.....	61
5.2	Aims	61
5.3	Problem description.....	61
5.4	Solving steps.....	62
5.5	Results interpretation.....	86
5.6	Things to remember.....	86
6.	3D elasto-plastic continuous problem.....	87
6.1	Keywords.....	87
6.2	Aims	87
6.3	Problem description.....	87
6.4	Solving steps.....	88
6.5	Results interpretation.....	107
6.6	Things to remember.....	107
7.	Mecanical coupling of solid 3D parts with their simplified 1D or 2D geometry.....	109
7.1	Keywords.....	109
7.2	Aims	109
7.3	Problem description.....	109
7.4	Solving steps.....	110
7.5	Results interpretation.....	130
7.6	Things to remember.....	131
8.	2D analysis of an elastic-perfect plastic rectangular sample subjected to triaxial compression	133
8.1	Keywords.....	133
8.2	Aims	133
8.3	Problem description.....	133
8.4	Solving steps.....	134
8.5	Results interpretation.....	143
8.6	Things to remember.....	144
9.	Mechanical–pore pressure (consolidation) analysis of a 3D sample .	145
9.1	Keywords.....	145
9.2	Aims	145
9.3	Problem description.....	145
9.4	Solving steps.....	146

9.5	Results interpretation.....	165
9.6	Things to remember.....	165
10.	Crushable particles problem.....	167
10.1	Keywords.....	167
10.2	Aims	167
10.3	Problem description.....	167
10.4	Solving steps.....	168
10.5	Results interpretation.....	181
10.6	Things to remember.....	183
11.	Time - dependent seismic loading of a system with continuous elements.....	185
11.1	Keywords.....	185
11.2	Aims	185
11.3	Problem description.....	185
11.4	Solving steps.....	187
11.5	Results interpretation.....	197
11.6	Results interpretation.....	199
12.	Equivalent liquid – solid (Euler-Lagrange) coupling.....	201
12.1	Keywords.....	201
12.2	Aims	201
12.3	Problem description.....	201
12.4	Solving steps.....	203
12.5	Results interpretation.....	226
12.6	Things to remember.....	226

TABLE OF FIGURES

Fig. 3.1: Window example	19
Fig. 3.2: Checkbox and drop-down example	19
Fig. 3.3: Module, model and part fast selection pop-down.....	20
Fig. 3.4: Example of bottom menu	20
Fig. 3.5: Navigation in menus	21
Fig. 3.6: Navigation in model tree	21
Fig. 3.7: The Material Manager window	22
Fig. 3.8: The Edit Material window	23
Fig. 5.1: Create shell menu	32
Fig. 5.2: Create Lines: Connected button	33
Fig. 5.3: The sketch of the shell and leaving the drawing area.....	34
Fig. 5.4: The imported 3D frame geometry	35
Fig. 5.5: The definition of the material properties: left) Mass Density; right) Elastic properties.....	36
Fig. 5.6: Creating the truss section left) selecting the Beam category; b) Edit Beam Section	37
Fig. 5.7: Create Profile menu: left) Selecting the shape; right) Submitting the radius value	38
Fig. 5.8: The Edit Beam Section's window, ready to be closed	39
Fig. 5.9: Edit Section Assignment window.....	40
Fig. 5.10: The selected beam and its tangent vectors.....	41
Fig. 5.11: The local axes of the selected beam	41
Fig. 5.12: The rendered beam profile	41

Fig. 5.13: Creating the shell section.....	42
Fig. 5.14: Filling the additional values (thickness and integration points number)	43
Fig. 5.15: Shell section assignment.....	44
Fig. 5.16: Importing the parts to the instance assembly.....	45
Fig. 5.17: Create a calculation step procedure: left) choosing the calculation type; right) selecting the calculation time period	46
Fig. 5.18: The Create Constraint window	47
Fig. 5.19: Selecting the Master Node Regions.....	48
Fig. 5.20: Selecting the Slave Surface	48
Fig. 5.21: Creating the Boundary Condition: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used	49
Fig. 5.22: The pop-up menu of Partition Face	49
Fig. 5.23: Selecting the two points (red dots) in order to divide the slab	50
Fig. 5.24: The slab equally divided along its edges	50
Fig. 5.25: Creating the Load: left) Selecting the general load type and acting step; right) Selecting direction on which the load acts	51
Fig. 5.26: Creating the Load: left) Selecting the general load type and acting step; right) Filling the pressure value.....	52
Fig. 5.27: The model loaded and fixed	53
Fig. 5.28: The modified slab Assigned Mesh Controls.....	54
Fig. 5.29: The model selected and the Global Seeds window	54
Fig. 5.30: The Meshed Slab	55
Fig. 5.31: The Create Job window and Edit Job	56
Fig. 5.32: The Job Manager window	56
Fig. 5.33: The deformed model and the Common Plot Option window set on Auto-compute Deformation Scale Factor	57
Fig. 5.34: The Field Output window	59
Fig. 5.35: Stress variation on the three dimensional frame.....	60
Fig. 5.36: Vertical displacement variation on the slab.....	60
Fig. 6.1: Model geometry.....	62
Fig. 6.2: Create part menu.....	63
Fig. 6.3: Part manager	63

Fig. 6.4: Input of elastic material properties for concrete and soil	65
Fig. 6.5: Create section and assigning material for concrete	65
Fig. 6.6: Create section and assigning material for soil	65
Fig. 6.7: Section assignment for Concrete part.....	66
Fig. 6.8: Section assignment for Soil part	66
Fig. 6.9: Creating the Independent instance.....	66
Fig. 6.10: Calculation steps.....	67
Fig. 6.11: Creating interaction property	68
Fig. 6.12: Definition of the contact tangential behaviour	69
Fig. 6.13: Definition of the contact normal behaviour.....	69
Fig. 6.14: Creating interactions	69
Fig. 6.15: BR - Select the master surface.....	70
Fig. 6.16: BR - Select the slave type.....	70
Fig. 6.17: BR - Select the slave surface	70
Fig. 6.18: Edit Interaction pop-up window	71
Fig. 6.19: Creating and assigning the boundary conditions	72
Fig. 6.20: Creating the loads	73
Fig. 6.21: Assigning Global Seeds.....	74
Fig. 6.22: Assigning the Mesh Controls	74
Fig. 6.23: Fully defined model	75
Fig. 6.24: Creating the analysis Job	76
Fig. 6.25: Editing the Job - Parallelization.....	76
Fig. 6.26: Job manager window	77
Fig. 6.27: Analysis Monitor window	77
Fig. 6.28: Stresses (Mises) displayed as Contours on Deformed Shape and colour legend.....	78
Fig. 6.29: Create Field Output window.....	79
Fig. 6.30: Field Output - component selection.....	81
Fig. 6.31: Relative U2 nodal displacements	81
Fig. 6.32: View Cut.....	82
Fig. 6.33: Creating a Node List Path.....	83
Fig. 6.34: Creating XY Data from path.....	84
Fig. 6.35: <input checked="" type="checkbox"/> Include intersections	84
Fig. 6.36: <input type="checkbox"/> Include intersections	84

Fig. 6.37: Copying the tabular data.....	85
Fig. 6.38: XY Data Manager.....	85
Fig. 6.39: XY Data Plot	85
Fig. 7.1: The considered model geometry.....	88
Fig. 7.2: Importing the CAD geometry: top) the two lithological layers; bottom) the raft.....	89
Fig. 7.3: The model's imported geometry: left) the lithology; right) the slab	90
Fig. 7.4: Assigning the Mohr Coulomb plastic properties of the soil layers: top) Soil Layer 1; bottom) Soil Layer 2	91
Fig. 7.5: The model's assembly	92
Fig. 7.6: Using the colour code to check the material/section assignment ..	93
Fig. 7.7: The Step Manager dialogue window	94
Fig. 7.8: The manual sequence of creating an interaction between the raft and the lithology	95
Fig. 7.9: Creating a variation pressure to which the raft is subjected	98
Fig. 7.10: The three different supporting conditions applied: white) on the 0Y (U2) axis; green) on the 0X (U1) axis; brick red) on the 0Z (U3) axis.....	100
Fig. 7.11: Creating the initial stress state in the lower lithological layer...	101
Fig. 7.12: Manually seeding the edges of a continuous part	102
Fig. 7.13: The meshed assembly	103
Fig. 7.14: The applied pressure variation	104
Fig. 7.15: Contact pressure variation and vertical displacement variation	104
Fig. 7.16: Paths along the raft's footing	105
Fig. 7.17: The subgrade modulus variation along the length of the raft: blue) Centre path; orange) Edge path.....	106
Fig. 7.18: Vertical stresses: left) initially declared; right) computed from the own weight and raft's load.....	107
Fig. 8.1: The considered model geometry.....	110
Fig. 8.2: The Create part from ACIS File in the case of the beam: left) Name-Repair tab; right) Part Attributes tab	111
Fig. 8.3: The Create part from ACIS File in the case of the wall: left) Name-Repair tab; right) Part Attributes tab	111

Fig. 8.4: The Create part from ACIS File in the case of the column: left) Name-Repair tab; right) Part Attributes tab	112
Fig. 8.5: The Create Part from IGES File in the case of the linear segment of the column: left) Name-Repair tab; right) Part Attributes tab	112
Fig. 8.6: The Create Part from IGES File in the case of the linear segment of the column: left) Name-Repair tab; right) Part Attributes tab	113
Fig. 8.7: Edit Material window while creating "Concrete Full"	114
Fig. 8.8: Edit Material window while creating the "Concrete Half"	115
Fig. 8.9: Creating the 3D homogeneous section "Concrete Full"	115
Fig. 8.10: Creating the 3D homogeneous section "Concrete Half"	116
Fig. 8.11: Creating the rectangular column profile	116
Fig. 8.12: Creating the linear column section	117
Fig. 8.13: Creating the shell section.....	118
Fig. 8.14: The assembly created, presenting the position of the considered parts	119
Fig. 8.15: Creating the loading calculation step.....	120
Fig. 8.16: The Find Contact Pairs window displaying the found contacts	121
Fig. 8.17: Creating the Interaction Property.....	122
Fig. 8.18: Creating the Coupling: left) The Create Constraint window; centre) Selecting the Coupling Constraint point and surfaces; right) The Edit Constraint window	123
Fig. 8.19: Creating the second constraint.....	124
Fig. 8.20: Creating the Embedded constraint: left) The Create Constraint window; centre) The two regions selected; right) The Edit Constraint window	125
Fig. 8.21: The encastered assembly, subjected to lateral load	126
Fig. 8.22: The Meshed assembly	126
Fig. 8.23: Overview of the assembly's deformed aspect.....	127
Fig. 8.24: Displacements along the OX axis on the column.....	128
Fig. 8.25: Sectional Forces along 0X axis being calculated only for the 1D or 2D elements.....	128
Fig. 8.26: Searching the lateral face existent set	129
Fig. 8.27: Plotting the resultant moment (orange) and force (red) of the section	129

Fig. 8.28: Deformed aspect of the column	130
Fig. 8.29: Comparison between the displacement obtained on the coupled wall: left) 3D element; right) shell	131
Fig. 9.1: The model's geometry	134
Fig. 9.2: Importing the modelled geometry: left) Create Part window; right) the geometry of the part	135
Fig. 9.3: Creating the material: left-top) density property; right-top) elastic property; bottom) Mohr Coulomb plasticity properties	136
Fig. 9.4: Creating a dynamic step.....	137
Fig. 9.5: The boundary conditions and external loads applied to the model	139
Fig. 9.6: Example of quad and tri-based meshing of the same geometry as presented in ABAQUS User's manual	140
Fig. 9.7: The Mesh Controls dialogue window	140
Fig. 9.8: The equivalent plastic strain variation (PEEQ): top-left) quad mesh&seed of 5mm; top-right) quad mesh&seed of 2.5mm; bottom-left) tri mesh&seed of 5mm; bottom-right) tri mesh&seed of 2.5mm	141
Fig. 9.9: Plotting the geometrical variation with respect to the original situation: left) the deformed and undeformed shapes and the SR's selected buttons; right) the pressure state of the deformed shape overlapped on the initial geometry	142
Fig. 9.10: Strain energy variation.....	142
Fig. 9.11: Different ODB display options: left) sweep; centre) mirror; right) circular pattern	143
Fig. 12.1: The model geometry	146
Fig. 12.2: Importing the geometry of the sample	147
Fig. 12.3: Creating the mechanical properties of the material: top-left) Density assigning; top-right) Linear elastic properties; bottom) Plastic Mohr Coulomb properties.....	148
Fig. 12.4: Creating the hydraulic properties of the material	149
Fig. 12.5: Creating the section and assigning the material	150
Fig. 12.6: Importing the part to the assembly	150
Fig. 12.7: Creating the consolidation step.....	151
Fig. 12.8: Partition Cell menu	152

Fig. 12.9: Partitioning the sample	153
Fig. 12.10: Creating the Set: left) Create Set window; right) Selecting the interest point.....	153
Fig. 12.11: The Edit History Output Request window.....	154
Fig. 12.12: Creating the mechanical boundary condition	155
Fig. 12.13: Creating the hydraulic boundary condition	156
Fig. 12.14: Creating the Cell Pressure stress	157
Fig. 12.15: Defining the initial void ratio	158
Fig. 12.16: The meshed soil sample.....	159
Fig. 12.17: Changing the element type	160
Fig. 12.18: The Monitor window	161
Fig. 12.19: Pore pressures at the end of the calculation step	163
Fig. 12.20: Deformations at the end of the consolidation step.....	163
Fig. 12.21: The effective pressure acting on the sample.....	163
Fig. 12.22: Obtaining the history data output	164
Fig. 12.23: Pore pressure variation of the middle and top of the sample... 164	164
Fig. 12.24: Vertical displacement variation of the middle and top of the sample	164
Fig. 14.1: Model's geometry	168
Fig. 14.2: The three considered parts: left) Bottom Plane; centre) Sample; right) Top Plane.....	168
Fig. 14.3: Creating the material	169
Fig. 14.4: The final state of the assembly	173
Fig. 14.5: Creating the calculation step.....	173
Fig. 14.6: The contact property definition	174
Fig. 14.7: Creating the interaction	175
Fig. 14.8: Creating the Rigid Body constraint	176
Fig. 14.9: Creating the velocity boundary condition on the platen.....	177
Fig. 14.10: The boundary conditions applied to the model.....	177
Fig. 14.11: The meshed model.....	178
Fig. 14.12: Internally generated particles per parent element illustrated for three particles per isoparametric direction as presented in ABAQUS User's manual	179

Fig. 14.13: Displacements on the analysed model: left) Classical FEM approach; right) FEM SPH approach.....	179
Fig. 14.14: Plastic strain on the analysed model: lef) Classical FEM approach; right) FEM SPH approach.....	180
Fig. 14.15: Contact pressures on the two platens: left) Classical FEM approach; right) FEM SPH approach.....	180
Fig. 14.16: The two element types of the concrete sphere: left) Continuous 3D elements (C3D4); right) SPH particles (PC3D)	180
Fig. 14.17: The evolution of the conversion of elements from C3D4 type to PC3D	182
Fig. 14.18: The total consumed energy of the whole model	183
Fig. 15.1: The considered model's geometry	186
Fig. 15.2: The time dependent acceleration function.....	187
Fig. 15.3: The two components of the model: the poleand the mass	188
Fig. 15.4: The rendered pole section.....	189
Fig. 15.5: The assembly's geometry.....	190
Fig. 15.6: The Edit Step dialogue window of the "Free vibration" step	191
Fig. 15.7: The connected sphere's regions to the pole's top point	192
Fig. 15.8: Creating an Amplitude function: left) Choosing the amplitude type to create; right) Edit Amplitude dialog window	194
Fig. 15.9: The vibration load options: left) applying the Amplitude function; right) disabling the load during the second step.....	195
Fig. 15.10: The assembly submitted to the external loads and the boundary conditions	196
Fig. 15.11: The meshed assembly	197
Fig. 15.12: Graphically displaying the difference between the initial equilibrium position and the one obtained: top) options to be checked; bottom) the final figure	198
Fig. 15.13: The displacement variation for the three interest points.....	199
Fig. 15.14: Displacement variation along the excitation direction	200
Fig. 15.15: Velocity variation along the excitation direction.....	200
Fig. 15.16: Acceleration along the excitation direction	200
Fig. 16.1: The parts of the model and their positioning	202
Fig. 16.2: The import procedure	204

Fig. 16.3: The “create part” window for the import of the solid part	205
Fig. 16.4: The “create part” windows for the import of the water parts	206
Fig. 16.5: The solid material (concrete) definition	207
Fig. 16.6: The EOS definition for water material	208
Fig. 16.7: The definition of the viscosity for the water material.....	209
Fig. 16.8: The section creation for the solid.....	210
Fig. 16.9: The section creation for the liquid.....	210
Fig. 16.10: Create step procedure for the analysis	212
Fig. 16.11: Create interaction property window	212
Fig. 16.12: The choosing of the interaction properties	213
Fig. 16.13: The create interaction procedure	214
Fig. 16.14: Seed window.....	215
Fig. 16.15: The Element Type assignment for the solid part.....	216
Fig. 16.16: The Element Type assignment for the Eulerian parts.....	216
Fig. 16.17: The meshed model.....	217
Fig. 16.18: The volume fraction tool path.....	218
Fig. 16.19: The volume fraction tool window.....	218
Fig. 16.20: The predefined field path.....	219
Fig. 16.21: The predefined field window	219
Fig. 16.22: The edit predefined field window	220
Fig. 16.23: Creating the Boundary Condition for the solid part: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used	221
Fig. 16.24: Creating the Boundary Condition for the Eulerian medium part: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used	221
Fig. 16.25: Creating the Load: left) Selecting the general load type and acting step; right) Selecting direction on which the load acts	222
Fig. 16.26: The Create Job window and Edit Job	223
Fig. 16.27: The Job Manager window	223
Fig. 16.28: The view cut manager.....	225
Fig. 16.29: The view cut manager window	225
Fig. 16.30: The stresses induced in the fluid part (a) and solid part (b)....	226

1. INTRODUCTION

This book addresses the Geotechnical Engineering professionals who are looking for a versatile software application to solve all the multiphysics problems they face when modelling soil and its interaction with various types of structures. The book is built as a step-by-step tutorial cookbook for solving simple problems that may easily be combined into more complex simulations.

The chosen tool for numerical modelling is ABAQUS, whose flexibility allows the implementation of most common practice and special case problems in Geotechnical Engineering.

Some background knowledge is assumed for the reader of this book, especially in the field of Linear Algebra, Mathematical Physics, Theory of Elasticity and Plasticity, Statics and Dynamics of Structures, Soil Mechanics and Foundation Engineering.

The book is restraint to minimum theoretical proofs, just postulating, where deemed necessary the governing equations implemented in the application, being more addressed to designers rather than researchers. The book is built to drive the reader from the simplest structure to the more intricate ones with a how-to approach. Virtually, after reading the first chapters, defining the general working procedures of the software application, the reader may jump directly into the problems they are interested in, however this would not be advisable since some specific procedures such extracting specific results from

the post-processor interface are explained as they become necessary. The only bibliography used for compiling this book is ABAQUS base documentation.

Even if ABAQUS itself has a very powerful pre-processor, other dedicated instruments may be used for creating the geometrical shapes of the bodies to be analysed, according to the skills of the user. One very common such tool is AutoCAD, very commonly used by design civil engineers. After defining the geometry, the bodies are imported as parts in ABAQUS pre-processor provided the exported formats and three dimensional skew shapes are supported.

As most superior Finite Element Method applications, ABAQUS does not implement a system for interrogating the user for parameters with measuring units, being ultimately the user concern to use a self-consistent system. In this book, the main measuring units used are:

- length: m
- time: s
- density: tons/m³ (for rendering compatible with the forces expressed in kN)
- force: kN
- pressure: kPa = kN/m²
- temperature: °C

The derived measuring units (such as area, volume, speed, acceleration and so on) are either composed from the ones mentioned before or defined in the problem they are used.

2. NAVIGATION

In this book, the navigation sequence shall be written in bold letters using the following convention:

<LMB> <RMB> <MMB> <Scroll> - mouse commands: left mouse button, right mouse button, middle mouse button, scroll wheel

<shift> - keyboard button

[Cancel] - software button, Fig. 2.1 ①

○ - radio selection Fig. 2.1 ②

Name:<Demo> - field box / text box Fig. 2.1 ③

☒ - checkbox selection Fig. 2.2 ①

Distribution ▾ Uniform - pop-down menu Fig. 2.2 ②

File - menu command Fig. 2.3 ①

|Model| - window tab Fig. 2.3 ②

[+] **Model** – tree branch Fig. 2.3 ③

Module: **Property** - module selection (the name of the modules is not user changeable) Fig. 2.3 ④

Model: **Demo** - model selection Fig. 2.3 ⑤

Part: **Demo** - part selection Fig. 2.3 ⑥

SR  **Create part** - side ribbon button command. Instead of **SR** (side ribbon) it may be used **TR** (top ribbon) or **BR** (bottom ribbon) Fig. 2.3 ⑦

SR  **Create part manager** - Manager command button (Since the manager buttons  to be clicked are all the same, we place it next to the command it manages). Fig. 2.3 ⑧

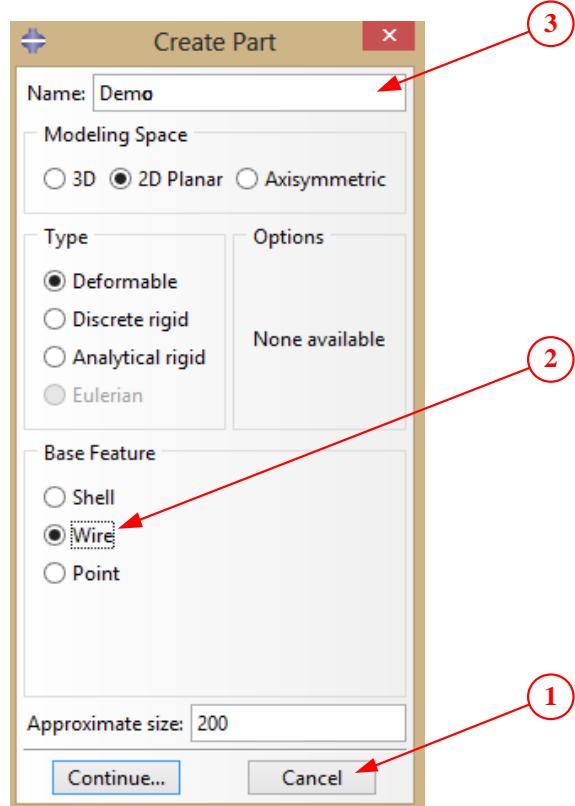


Fig. 2.1: Window example

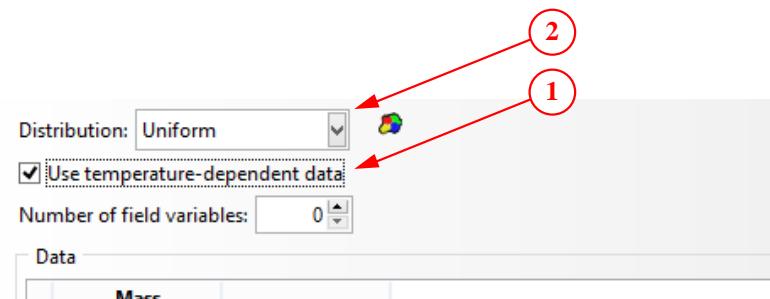


Fig. 2.2: Checkbox and drop-down example

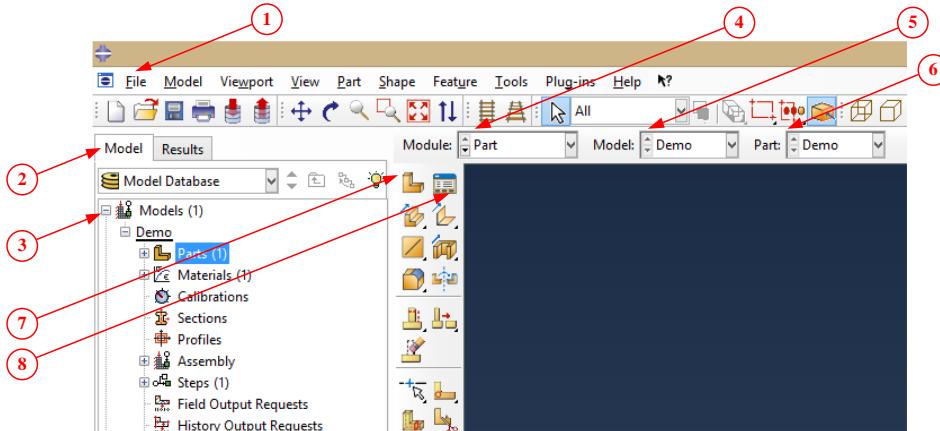


Fig. 2.3: Module, model and part fast selection pop-down

[Done] - bottom menu command Fig. 2.4

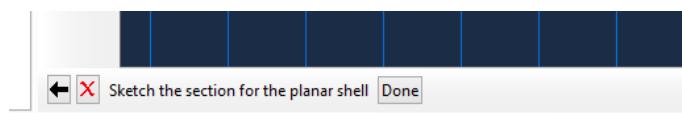


Fig. 2.4: Example of bottom menu

► - jump to the next submenu/command

... - jump from menu chain to window

As an example, the navigation described in Fig. 2.5 shall be written: **File** ► **Import** ► **Part...**, while if the command is issued starting from the model tree (Fig. 2.6) is <ctrl> + <T> to activate the model database, followed by **|Model|** ► **Model Database** ► **[+ Models** ► **[+ Demo** ► **Parts** ► **RMB** ► **Import...**

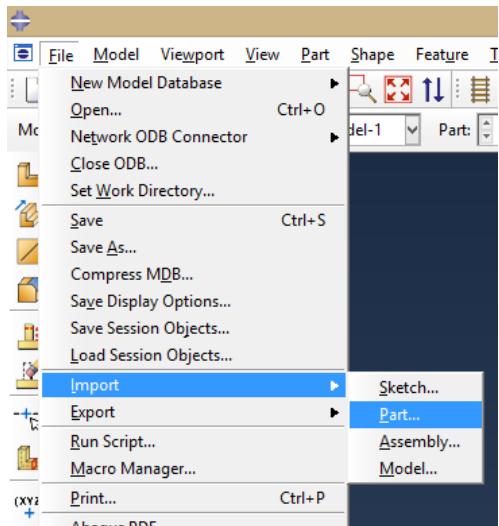


Fig. 2.5: Navigation in menus

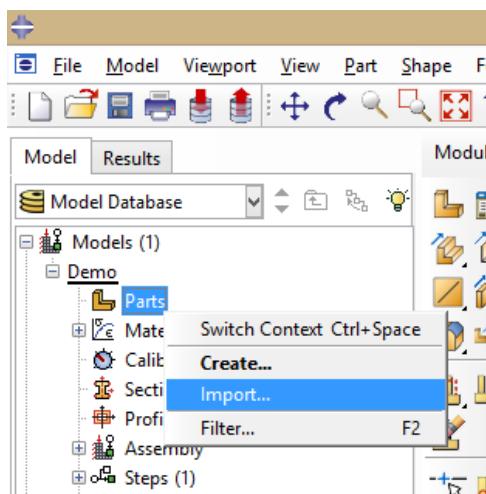


Fig. 2.6: Navigation in model tree

The software offers multiple possibilities to navigate through the interface in order to obtain the same results. For example, let's consider deleting or modifying the material law previously created.

The first, and one of the most direct ways to do so, is to go **Module: Property**, click LMB on the SR  Material Manager and the Material Manager window appears, where all the created properties appear. Select the desired material and click either the **Delete...**, if the user wishes to delete or **Edit...**, if it is desired to modify certain aspects.

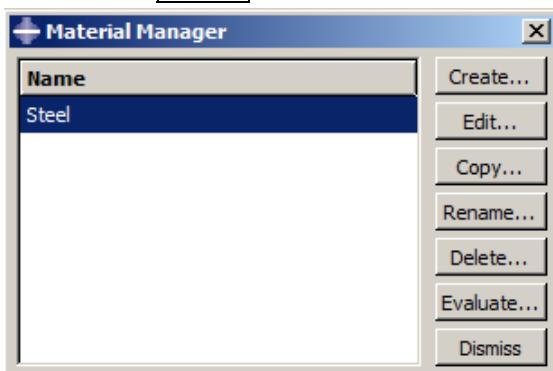


Fig. 2.7: The Material Manager window

The second way to delete or edit the material properties is by accessing through the upper menus, along the path **Material** ▶ **Edit** ▶ «**Material Name**» ▶ **LMB**.

The third and last way is by using the model tree: **[+]** **Models** ▶ **[+]** «**Model Name**» ▶ **[+]** **Materials** ▶ «**Material Name**» ▶ **RMB** ▶ **Edit...**

All of the above described ways bring the user to the Edit Material dialog window, where all previously declared properties will be displayed (Fig. 2.8: The Edit Material window).

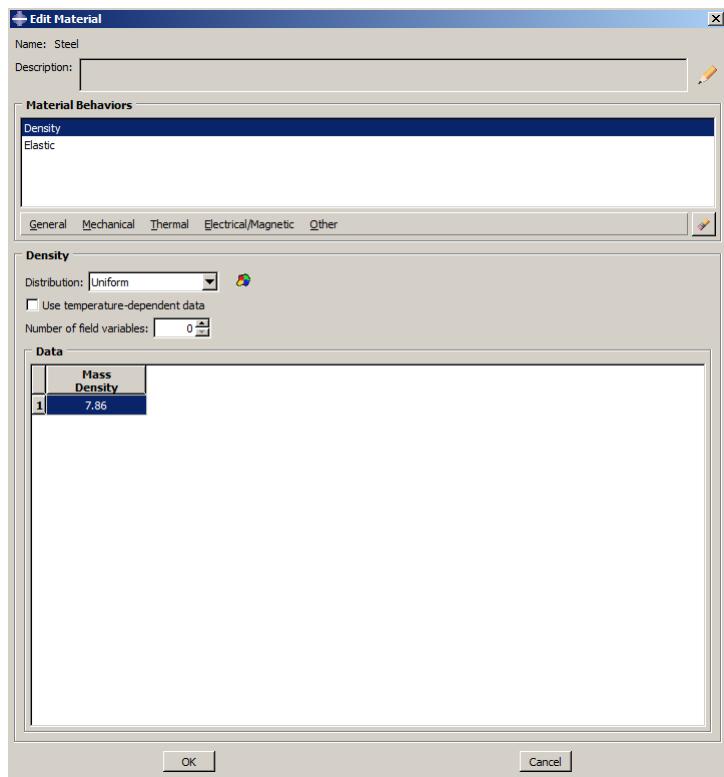


Fig. 2.8: The Edit Material window

3. APPLICATION MODULES

The actions regarding the creation of a numerical model are grouped in several modules, which cover, in a logical manner, the path from designing the parts' geometry, to assigning loads, optimizing their sections, calculating and reviewing the obtained results. These modules can be accessed either from the **model tree**, either from the **TR** pop-down menu named **Module:**

3.1 Module: Part

This is the first of the modules, where the geometry of the numerical model is declared, modified or imported from geometry files created using CAD software. If the case of importing the geometry, only the following formats are supported: ACIS SAT (*.sat), IGES (*.iges, *.igs), VDA (*.vda), STEP (*.stp, *.step), CATIA V4 (*.model, *.catdata, *.exp), CATIA V5 (*.CATPart, *.CATProduct), Parasolid (*.x_t, *.x_b, *.xmt) or ProE/NX/IDEAS Elysium Neutral (*.enf).

Also sketches can be imported under the formats of ACIS SAT (*.sat), IGES (*.iges, *.igs), STEP (*.stp, *.step), AutoCAD DXF (*.dxf), or entire assemblies, if presented under the following formats: Assembly Neutral (*.eaf), CATIA V4 (*.model, *.catdata, *.exp), Parasolid (*.x_t, *.x_b, *.xmt) or ProE/NX/IDEAS Elysium Neutral (*.enf).

The geometry can be also exported in order to be further used, as:

- sketch: ACIS SAT (*.sat), IGES (*.iges, *.igs), STEP (*.stp);
- part: ACIS SAT (*.sat), IGES (*.iges, *.igs), VDA (*.vda), STEP (*.stp, *.step);
- assembly: ACIS SAT (*.sat);
- VRML (Virtual Reality Modeling Language) (*.wrl; *.wrz);
- 3DXML (*.3dxml);
- OBJ (*.obj).

The **Part** menu allows the creation, editing or deletion of the parts considered necessary for the simulation.

The **Shape** menu allows the application of different geometrical techniques in order to obtain the desired geometry for each of the considered part.

3.2 Module: Property

The second module suggested is Property. In this case, the materials, sections and profiles needed for each part can be created, modified, assigned or deleted, according to user's needs.

The **Material** menu allows the creation, modification and deletion of the material laws needed during the simulation, which range from state properties such as density to mechanical, thermal, electrical or acoustic laws.

The **Section** menu gives the possibility to create computational sections for all the types of the possible used elements – one, two or three dimensional, rigid bodies etc. It also allows the management of the assignment of the created sections on each part.

If the case of using one dimensional elements in the model, the **Profile** menu allows to create, edit, assign and delete these, as needed.

Similar to Profile is the **Composite** menu, which allows the creation, editing, assigning or deletion of composite sections.

3.3 Module: Assembly

The module Assembly allows the user to geometrically position the parts in their desired arrangement, copy multiple parts, translate or rotate each of them so that the considered set up is created. All the aforementioned options are available under the **Instance** menu.

Also geometrical constraints may be created, edited or deleted, including Parallel Face or Parallel Edge, Coincident Point or Coaxial, from the **Constraint** menu.

3.4 Module: Step

The forth module suggested by the software is Step. It allows through the use of the **Step** menu the creation, editing and suppression of the calculation steps and their order.

The **Output** menu allows the user to demand certain variables (results) to be computed, as function of time or not (Historical Output). Also Integration Output Sections may be attached to three dimensional continuous parts, in order to obtain both stresses and sectional forces.

3.5 Module: Interaction

This module allows the creation, editing and deletion of the interactions the user reasoned to exist between the parts belonging to the assembly. The **Interaction** menu offers the possibility of Automatic identifying the contact pairs between two or more adjacent parts, the manual creation of new interactions, editing and deleting the existing ones, creating, modifying and deleting interaction properties, contact controls, contact initialization and contact stabilization.

The **Constraint** menu gives the possibility of creating, editing and deleting constraints between adjacent parts, such as Tie, Rigid body, Coupling, Shell to solid coupling, Embedded region etc.

The **Connector** menu offers the possibility of building connectors, assigning sections, geometry and assignment manager.

3.6 Module: Load

The Load module allows the user to define the external loads acting on the considered numerical model and the enforcement of different boundary conditions.

The **Load** menu permits the creation, editing and deletion of different loads (mechanic, thermic, electric etc.) on the model. These loads can be considered, depending on the necessities, either uniform, distributed, varying in space and time etc.

The **Boundary Conditions** menu gives the possibility of imposing different boundary conditions such as fixities, boundary thermic or electric limits etc. on the considered model.

The **Predefined Field** menu offers the possibility of creating, modifying or deleting more options related to the state of the assembly, such as initial saturation degree, initial void ratio etc.

In the case of considering linear variation steps, for example, the **Load Case** menu allows the user to create, modify or delete load cases, made of singular external loads, reunited under the same case, each with a magnitude factor of its own.

3.7 Module: Mesh

The Mesh module gives the possibility to discretize according to the user's best fit consideration the instance's parts. The **Seed** menu allows the user to enforce limits of the development of the elements in which the parts will be meshed. This can be done either by uniform input, either by manual edge by edge control.

The **Mesh** menu permits the user to change the type of element the mesh is made of, the type of the used discretization technique, but also to recover lost part geometry from the saved mesh.

The **Adaptivity** menu allows the software, following an initial calculation, to minimize the errors due to a faulty mesh, by using appropriate techniques to reconstruct the discretization.

3.8 Module: Optimization

The Optimization module is comprised of the **Task**, **Design Response**, **Objective Function**, **Constraint**, **Geometric Restriction** and **Stop Condition** menus. The overall aim of this module is to enhance the performance of the designed parts and assemblies, both in terms of structural resistance, structural stability and economic performance.

3.9 Module: Job

The Job module is the last of the pre-processing modules the user can access before submitting the entire model to the processor unit to be calculated. The **Job** menu offers the possibility of managing and monitoring the jobs, managing the information of the already created input files, observing the results and exporting the model to Nastran input file.

The **Adaptivity** menu allows the creation of processes which, based on the results already obtained, to allow the reconstruction of the mesh, as presented in the 3.7 subchapter.

The **Co-execution** menu permits the user to calculate in the same time of multiple jobs that interact in terms of shared results, state of the assembly etc.

The Optimization menu allows the user to create processes aiming at raising the performance level of the design, as described during the 3.8 chapter.

3.10 Module: Visualization

The Visualization module offers varying possibilities to extract data obtained following the calculation phase.

The **Result** menu gives the possibility to dissect the results on various points of interest (time, frame, variable output type etc.) and create new fields of interest.

The **Animate** menu permits the user to modify the conditions in which a mini-movie and create it concerning the evolution of the assembly's state.

The **Report** menu allows to plot and extract the data, in order to be used further by the user.

3.11 Module: Sketch

This module is a easily usable in order to sketch the forms of simple parts, later to be considered their geometry.

4. 3D FRAME ANALYSIS

4.1 Keywords

Linear element, Surface elements, Elastic behaviour, Tie (interaction)

4.2 Aims

- Importing parts to, and creating parts in ABAQUS
- Defining the linear elastic behaviour of materials
- Creating and assigning materials to sections
- Creating and assigning sections to parts
- Creating analysis steps and demanding additional results
- Defining the interaction properties between parts
- Defining boundary conditions and loads
- Surface processing and meshing
- Processing and displaying the output results

4.3 Problem description

This first problem is considering a three dimensional metal frame, made of circular bars Ø100mm, which is having at the upper level a 25mm thick slab welded to it. The structure is subjected to a gravitational load ($g=9.81\text{m/s}^2$), during the first calculation step, and an unsymmetrical pressure applied to the plate ($p=50\text{kPa}$). The vertical elements are base-encastred.

4.4 Solving steps

The shell is to be drawn in the ABAQUS preprocessing interface. In **Module: Part**, at **SR**  **Create part**, select 3D (Modelling Space tab), Deformable (Type tab), Shell (Base feature), Planar (Type).

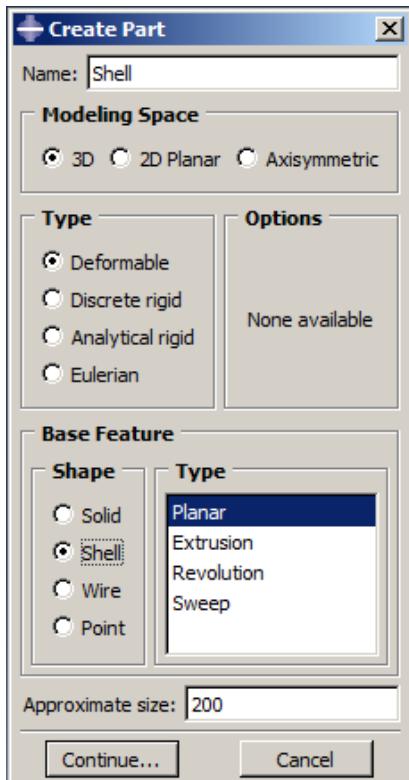


Fig. 4.1: Create shell menu

Inside the drawing space, the **SR**  **Create Lines: Connected** is to be selected using **LMB**.

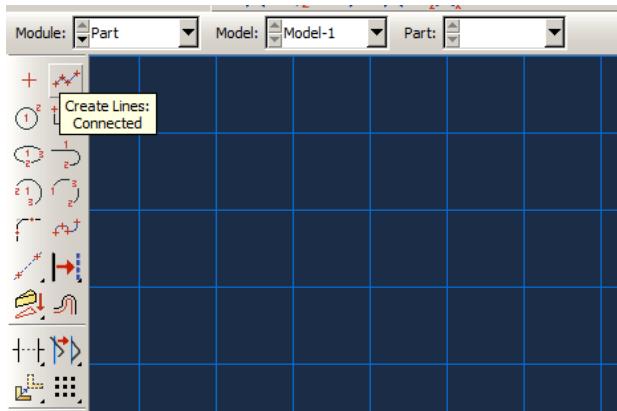


Fig. 4.2: Create Lines: Connected button

The four corners of the shell will be inserted, as coordinates in (X, Y) system.

The sequence to be followed for these points is presented below:

- (0,0) for the first corner;
- (10,0) for the second corner;
- (10,10) for the third corner;
- (0,10) for the fourth corner;
- and inserting again (0,0) in order to close the figure.

In order to end the sketching of the shell, in the base, the **Done** button is to be pressed.

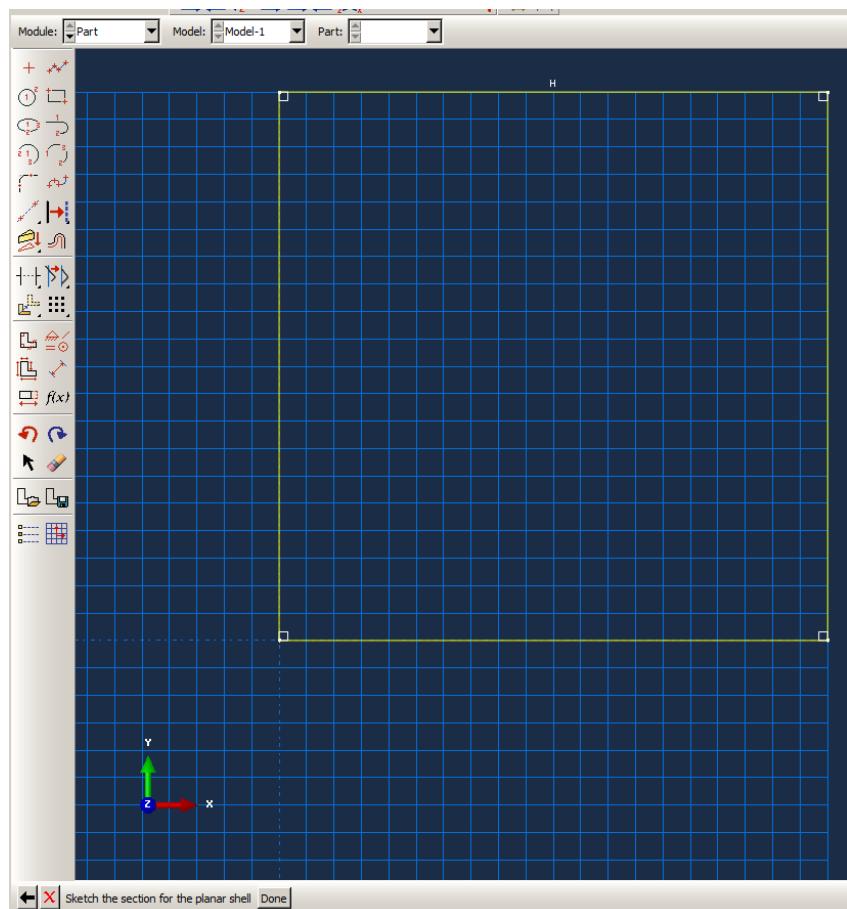


Fig. 4.3: The sketch of the shell and leaving the drawing area

The frame geometry will be created using any CAD software that can export it under the IGES (.igs; .iges) format. All the spans are 10m wide, the height of the first beam is 5m, while the upper level is at 10m height. In order to import this part, the following command sequence is to be followed **File ▶ Import ▶ Part...** and select from the work folder the file containing the geometry.

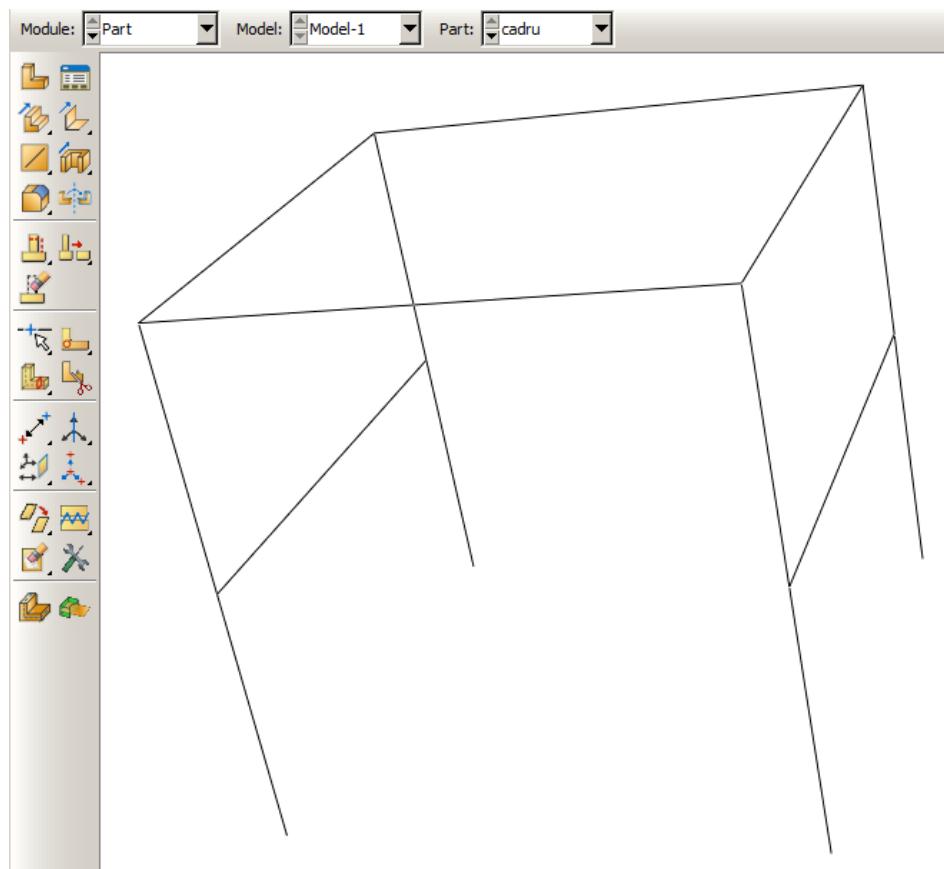


Fig. 4.4: The imported 3D frame geometry

At this moment, the two necessary parts are available to be used. To continue, we will move to **Module: Property**, where the material and the sections will be created and assigned to the parts' geometry. Select the **SR** **Create material** button, using the **LMB**.

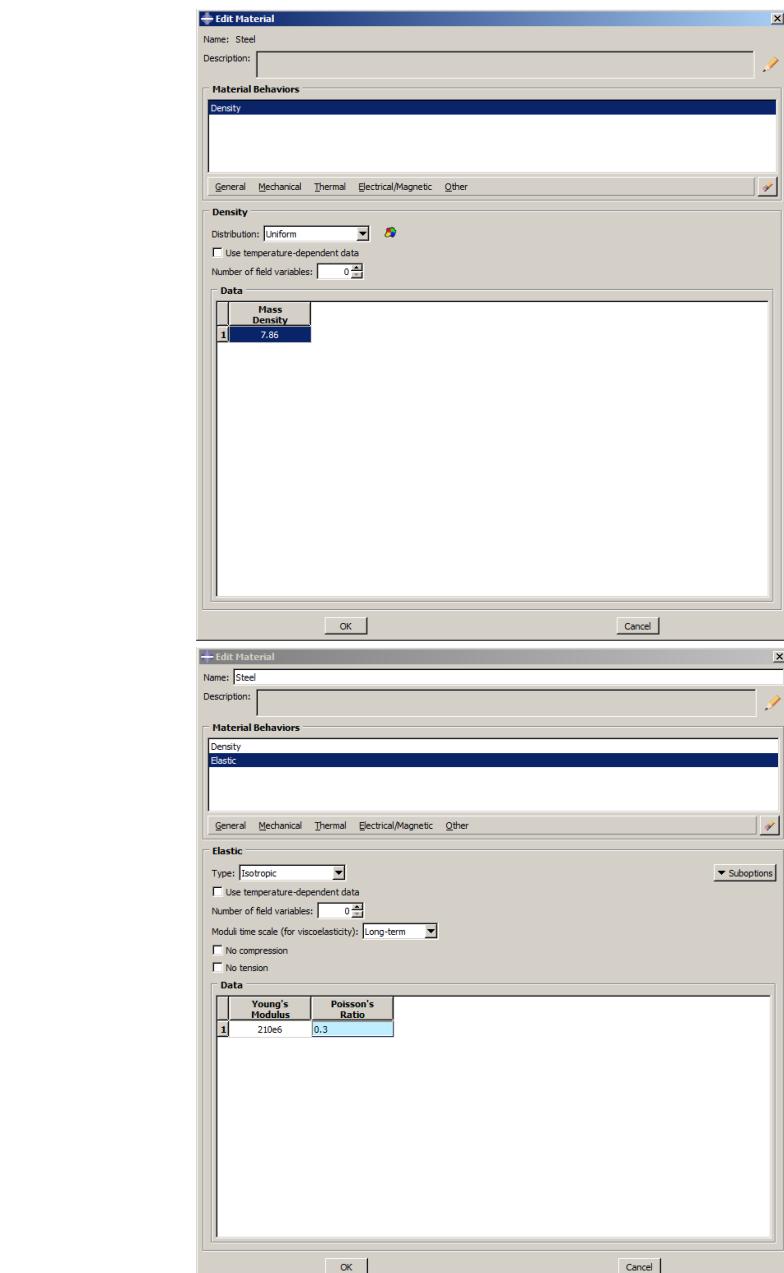


Fig. 4.5: The definition of the material properties: left) Mass Density; right) Elastic properties

Under the **Edit Material** window, at **General:** **Density**, where we will input the value of 7.860(to/m^3). Following his step, at **Mechanical:** **Elasticity** ► **Elastic**, the Young's Modulus will be filled with the $210\text{E}^6(\text{kPa})$ and a Poisson's Ratio value of 0.3, respectively. In order to end the creation of the material, click **LMB** on the **OK** button.

We move to the creation of the sections: the circular truss section to be applied to the three dimensional frame and the shell section to be applied to the slab. In order to create the truss section, click **LMB** on the **SR**  **Create Section** button and select **Beam**, type Beam.

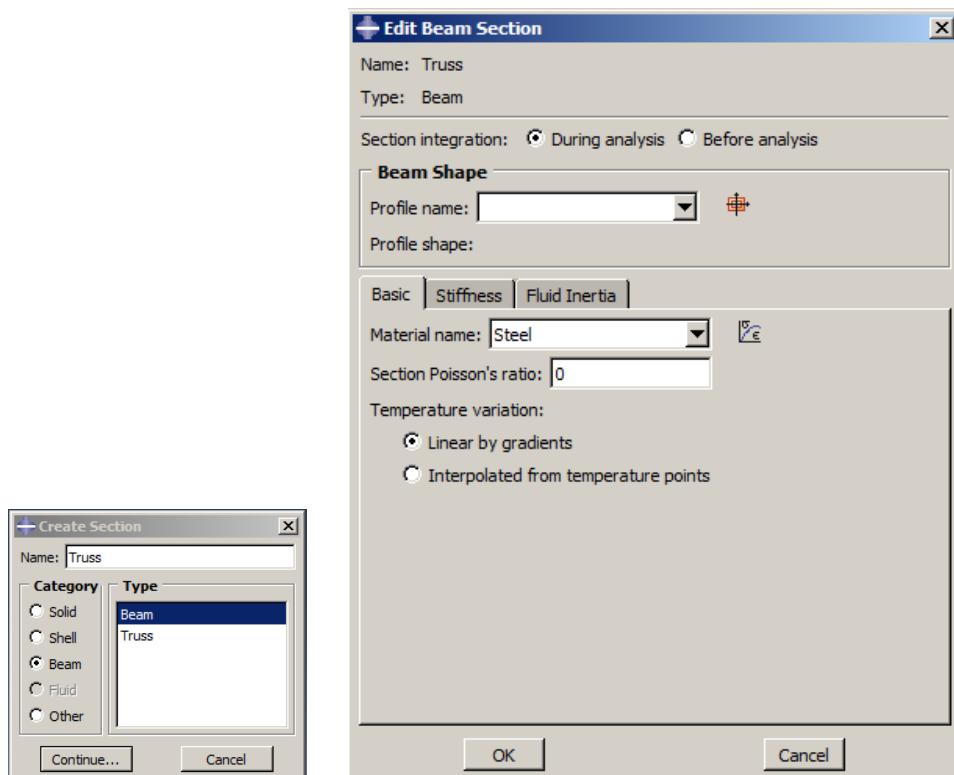


Fig. 4.6: Creating the truss section **a)** selecting the Beam category; **b)** Edit Beam Section

Click LMB on the  **Create Beam Profile** button and select under the **Create Profile** window the **Circular** type. Click the **Continue ...** button and in the **r:** window, insert the value of 0.05m (50mm).

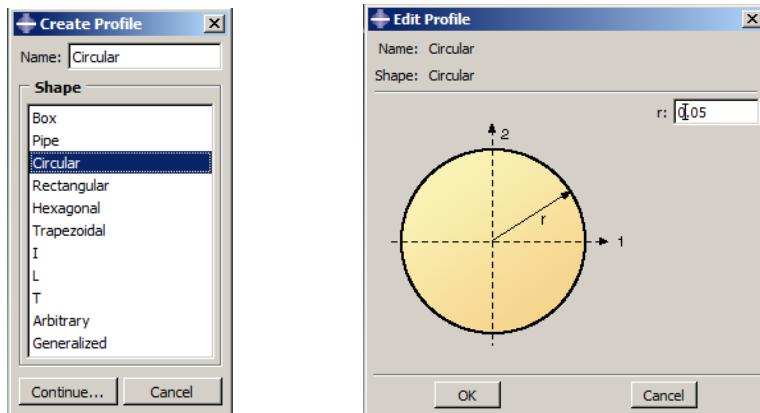


Fig. 4.7: Create Profile menu: left) Selecting the shape; right) Submitting the radius value

Automatically, the name of the created profile will appear under the **Profile name** selection pop-down menu. Also, fill in the **Section Poisson's ratio** with a value of 0.3 (equal to the one used for the material creation). Finish the instance, click LMB the **OK** button.

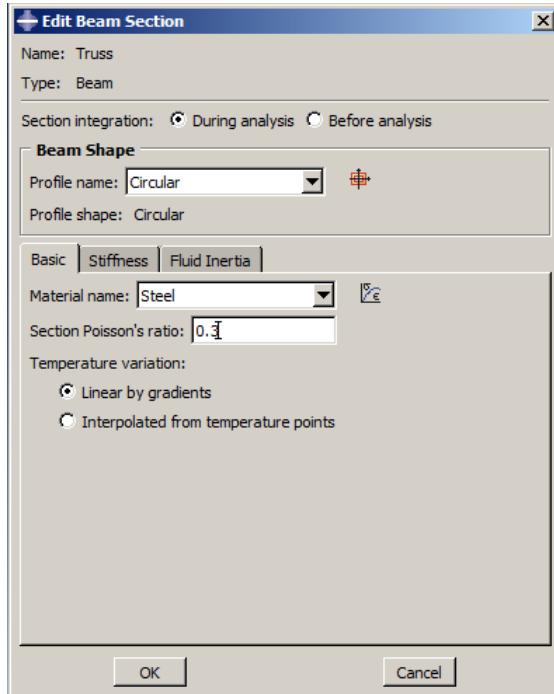


Fig. 4.8: The Edit Beam Section's window, ready to be closed

Assigning the section to the truss is done using the **SR** **Assign Section** button, on which click the **LMB**. The software prompts you in the bottom menu bar to Select the regions to be assigned a section; to do so, hold the LMB clicked and drag a window, selecting the whole frame. At the end of the operation, it will be underlined (the white default colour will turn red). Click **Done** and the Section “Truss” will appear, following which the **OK** button will finish the assignment procedure.

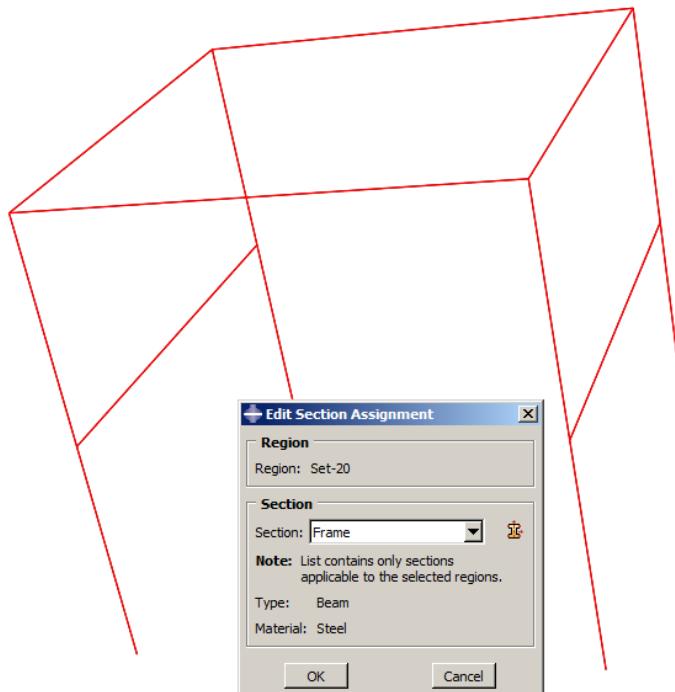


Fig. 4.9: Edit Section Assignment window

If correctly assigned, the frame will turn pale-green. In the case of the beams, the section orientation is needed, which is inputted using a unit vector parallel to the axis of the truss. This is done by clicking the **LMB** on the **RS** **Assign Beam Orientation** button. This will bring forward, in the bottom menu, the requirement to Select the regions to be assigned a beam section orientation. Select one of the beam and click the **OK** button. The beam will be underlined (red colour), and arrows along its length denote the tangent vectors. In the window that appears in the bottom menu, insert the position of the tip of the vector (X, Y, Z coordinate system). Confirming by clicking the **OK** button will end the assigning of the local axis on the selected element.

In order to confirm the validity of the assignment, click **View** **Part Display Options...**, and in the **Part Display Options** window, check **Render beam profiles**. After selecting the **OK** button, the previously selected beam is

drawn at its full thickness, provided by the considered profile (in this case, a circle).

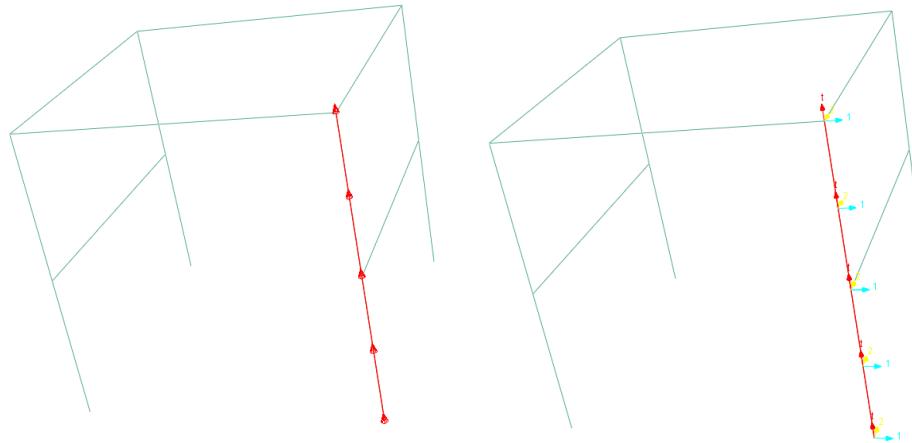


Fig. 4.10: The selected beam and its tangent vectors

Fig. 4.11: The local axes of the selected beam

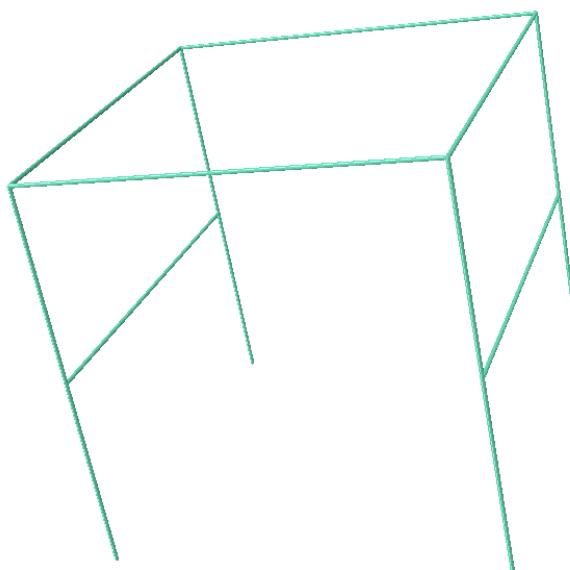


Fig. 4.12: The rendered beam profile

The local axis assignment procedure is to be repeated until all the components are fully rendered.

Concerning the slab, a new section will be created, using the **SR**  **Create Section** button. In the Create Section window, for the Category tab, check the **Shell**, Homogeneous and select **Continue ...**. In the Edit Section window, fill the Shell thickness **Value** with 0.025(m), according to the problem description, and instead of using the default Thickness integration points equal to 5 value, increase it to 11. This will ensure a balanced calculation of the slab's stresses.

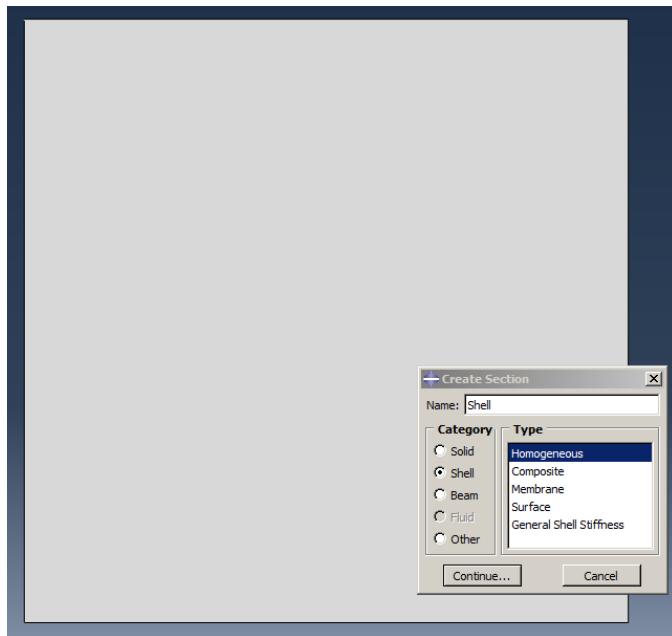


Fig. 4.13: Creating the shell section

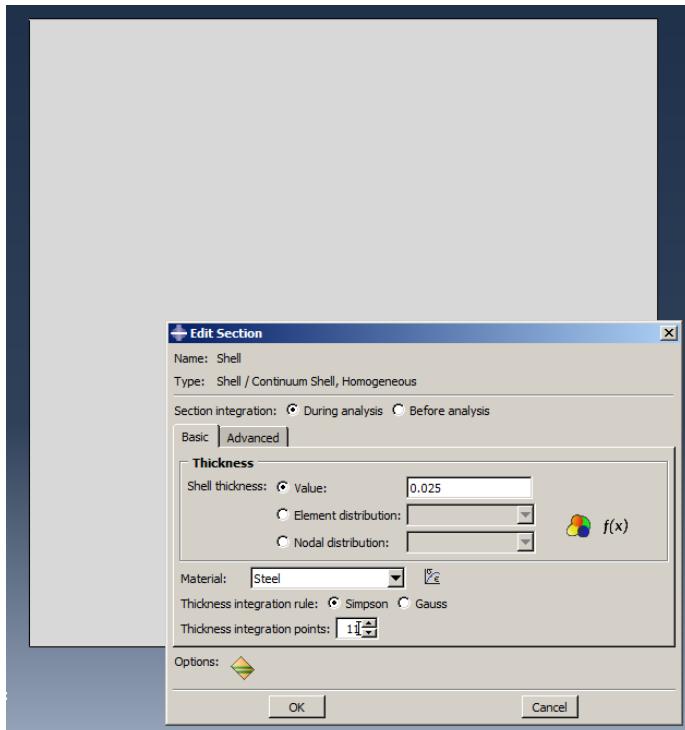


Fig. 4.14: Filling the additional values (thickness and integration points number)

We assign this section to the part by clicking LMB on the SR Assign Section, selecting the part, maintain the default Middle surface, under the Shell offset and click the **OK** button.

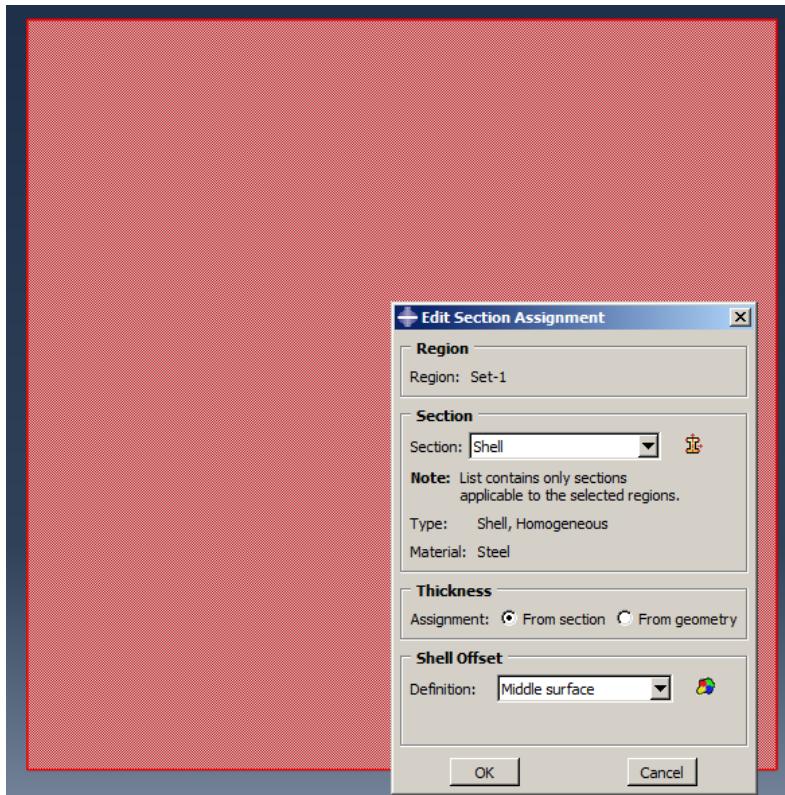


Fig. 4.15: Shell section assignment

Moving to the **Module: Assembly**, in order to insert the two components into the whole system, **LMB** click the **SR** **Instance Part** button, check **Independent** (mesh on instance) and while holding the **Shift** button, click **LMB** on the two parts in order to bring them to the same instance. Finish the procedure by clicking the **OK** button.

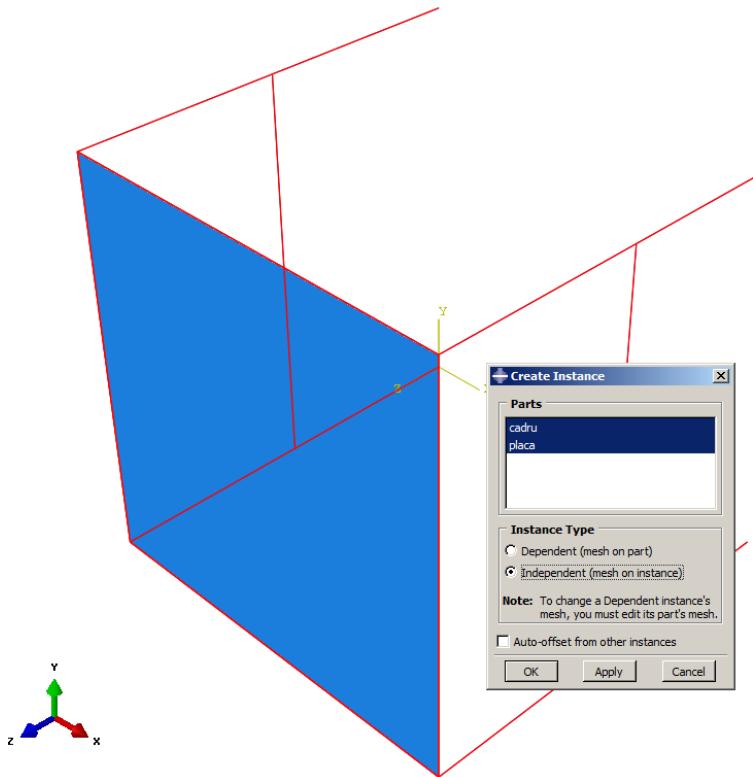


Fig. 4.16: Importing the parts to the instance assembly

The two parts will appear in the required position, as presented in the Fig. 4.16. If a translation of the slab is needed to the upper part of the frame, use the RS Translate Instance button, which will start the translation procedure: first it is asked to select the part to be translated (this will be underlined with red colour), click **Done**, afterwards it is asked to provide a starting point for the translation vector and pick one of the slab's corners, and in the end select a point where the first one should be positioned. A preview of the translated part will appear, and if the user agrees, the procedure will be ended successfully by clicking on the **OK** button.

In order to rotate the representation of the parts, go to the **TR** Rotate View button, or simply hold **Ctrl+Alt+LMB** and drag the cursor on the model

space area. If the user wishes to pan the view on the model, click the **TR**  **Pan View** button, or access the shortcut of **Ctrl+Alt+RMB** and drag it across the screen.

Moving forward, to **Module: Step**, click on the **SR**  **Create Step** button, and name the calculation step “Load”. The default Procedure type is General and the type of calculation step is Static, General. Click the **Continue ...** button, and a new window will appear. In the description field, write “Loading step”. The time period will remain 1, as 1s. This implies 100% of the forces acting on the system to be applied to the structure, as the calculation is not time-dependent. Click the **OK** button, in order to complete the creation of the calculation step.

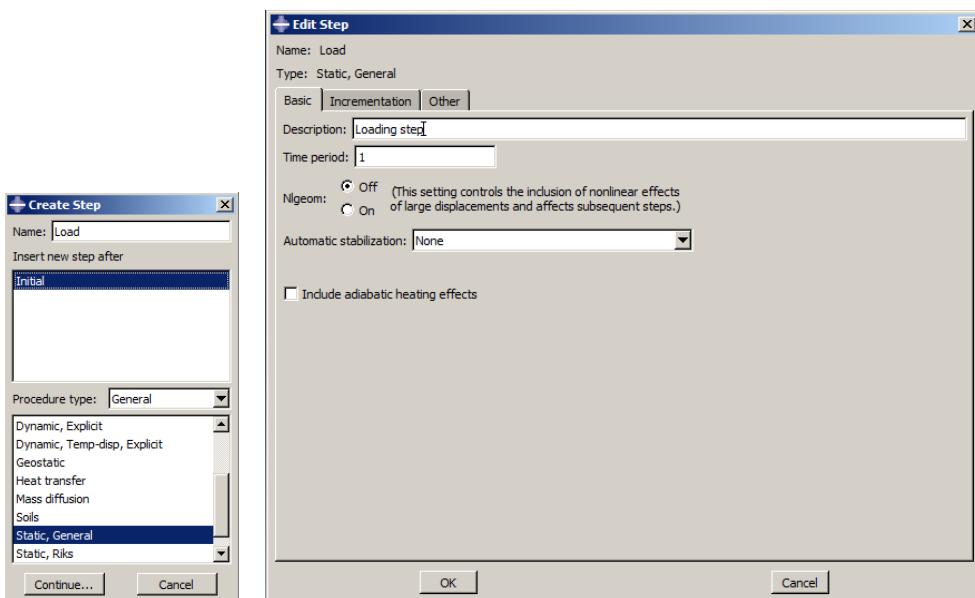


Fig. 4.17: Create a calculation step procedure: left) choosing the calculation type; right) selecting the calculation time period

Because the structure has been modelled as linear elements and shell elements, for the results of the calculations one expects to obtain forces (axial, shear and bending moments). They are not covered with the default

calculation demands, the user will go to **Output ▶ Field Output Requests ▶ Manager...** and a window appears, containing the existing field output. Clicking the **Edit...** button brings forth the Edit Field Output Request window, where the categories of Output Variables are displayed. Full check Forces/Reactions group (not grey, as default but black check).

At the **Module: Interaction**, click the **SR  Create Constraint** button. This will bring forward the Create Constraint window, in which the name field will be filled with “Weld”, and the Type to select is Tie. As described in the 4.3 subchapter, the connection between the two parts is done by welding, which implies, from the static point of view that the common nodes of the frame and the slab have the same displacements. Click the **Continue ...** button, in order to advance through the procedure.

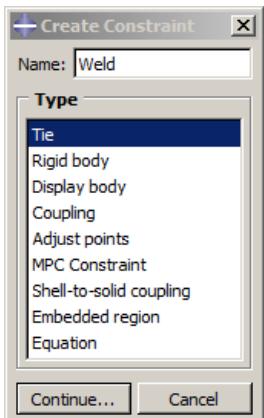


Fig. 4.18: The Create Constraint window

In the bottom menu, the software will prompt the user to Choose the master type and select Node region. In order to view only the frame, go to the **TR  Remove Selected** button and, after **LMB** clicking, select the slab and click **Done**. Following this procedure, the user should be able to view only the frame. Select only the upper boundary beam elements and click **Done**. Next, the software demands the type of the slave part and select Surface. In order to bring forth again the slab, go to the **TR  Replace All** button, click

on it, and continue with the slave surface selection, by clicking on the slab. Press the Enter key and, when prompted Choose a side for the shell or internal faces, choose the default Brown. This last option is useful only in the cases where the part has a three dimensional continuum geometry, and different faces are available to be chosen. In the following window – Edit Constraint, click the **OK** button, in order to complete the interaction assignment.

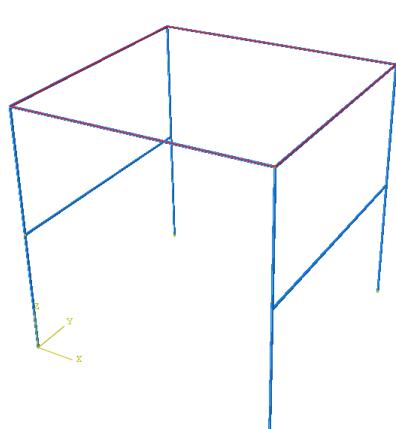


Fig. 4.19: Selecting the Master Node Regions

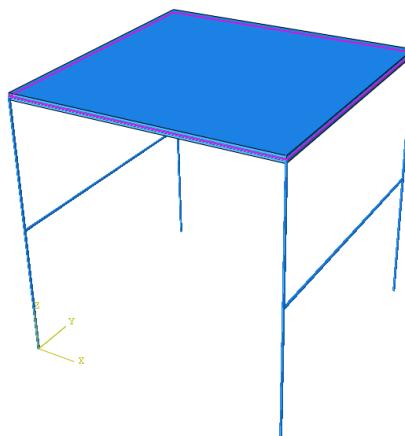


Fig. 4.20: Selecting the Slave Surface

The analysis input is almost complete: we need to fill the boundary conditions, the loads and mesh the parts. In order to do so, go to **Module: Load** and click the **SR  Create Boundary Condition** button, which brings forth the Create Boundary Condition window: name this restraint “Fixing”, verify the step in which it acts is “Load” and in the category type select **Mechanical**, while for the Types for Selected Step choose Symmetry/Antisymmetry/Encastre. Clicking **Continue ...** will advance to the next step, where the user is prompted to Select the regions for the boundary condition. Select the base points (while hovering over them, they will appear thickened orange dots) of the frame and click **Done**.

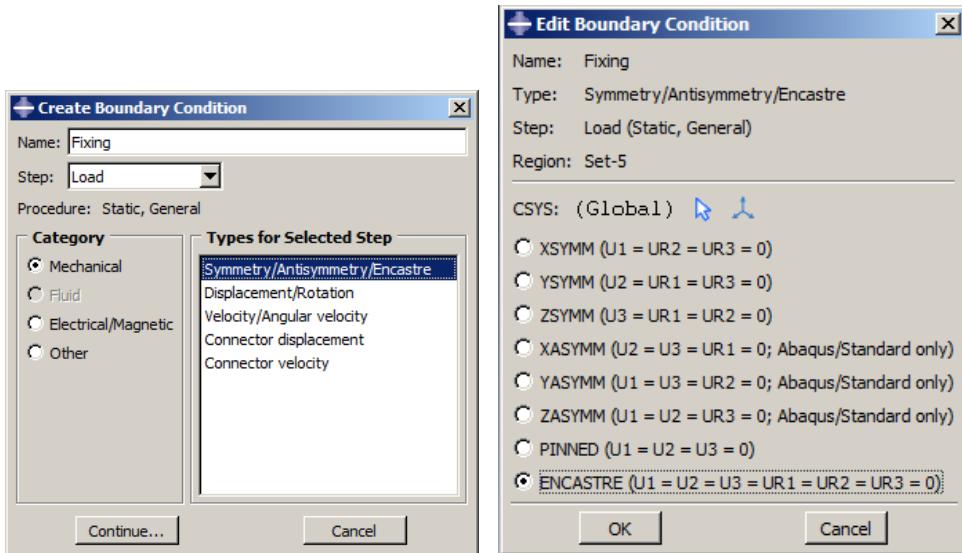


Fig. 4.21: Creating the Boundary Condition: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used

Clicking the **OK** button will end the creation of the boundary condition, and, from the visual point of view, around each of the selected nodes, a series of arrows (blue and orange) will appear, denoting it is fixed against both translation along the three axis and rotation around them.

In order to set the antisymmetric pressure, we have to divide the slab into four equal squares. This is done, by clicking and holding **LMB** on the **SR** **Partition Face: Sketch** button, which will bring forth a pop-up menu (Fig.

4.22). Select the **Partition Face: Use Shortest Path between 2 Points** button and the software will ask to provide a first section point. In order to select the middle of one of the slab sides, make the frame disappear by using the **TR** **Replace Selected** button and click on the slab, while in the bottom menu make sure that Faces entity type is selected.



Fig. 4.22: The pop-up menu of Partition Face

Now, returning to the dividing procedure, click on one of the middle points of the sides. When prompted to provide an end point, rotate the view of the model, using the aforementioned technique (either by using **Ctrl+Alt+LMB**, either the **TR**  **Rotate View** button) and select the second point. Click the **Create Partition** button. Now, a line dividing the slab should appear. Repeat the procedure on the orthogonal faces, in order to obtain a four square divided slab. Clicking **Done** will end the partitioning procedure.

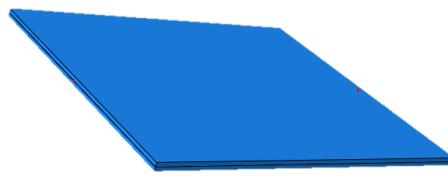


Fig. 4.23: Selecting the two points (red dots) in order to divide the slab

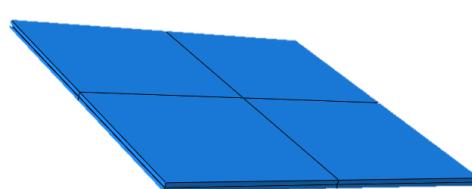


Fig. 4.24: The slab equally divided along its edges

As loads of interest, we will consider the self-weight of the system, using a gravitational acceleration applied to the whole model and a pressure that is acting only on one of the four squares. Go to the **RS**  **Create Load** button, click it, and the Create Load window will appear. Name the load “g” for gravitational, make sure the set step is “Load” and choose **OMechanical** for the load category and Gravity for the Types for Selected Step. Click **Continue...** and a second window (Edit Load) appears in which it is required to provide the domain on which the gravity load should be applied and on the three directions its components. Therefore, as it can be observed, the region is the whole model, by default, and in the field near the Component 3 fill with the $9.81 \text{ (m/s}^2)$ value.

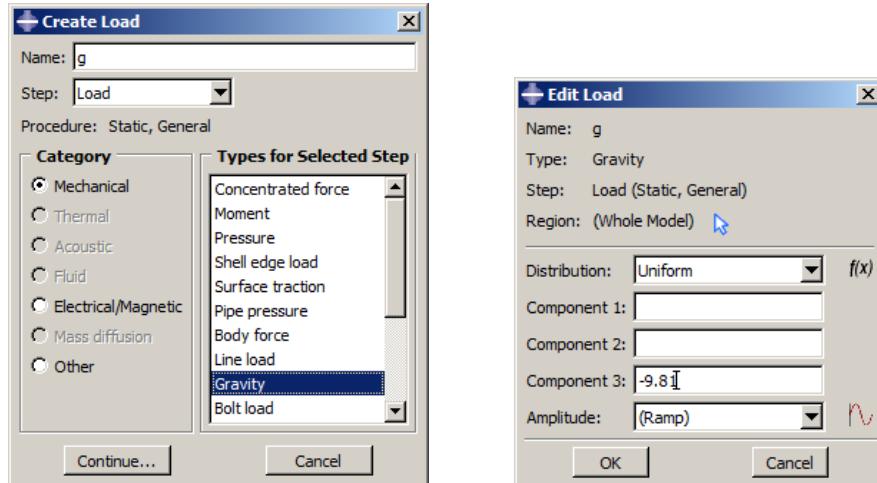


Fig. 4.25: Creating the Load: left) Selecting the general load type and acting step; right) Selecting direction on which the load acts

In order to create the pressure on the slab, click again on the **SR** **Create Load** button, and now, select Pressure under the Types for Selected Step options menu, having the name “Press”. Clicking **Continue ...** will bring forth the model and the software demands the user to pick the surfaces on which the pressure will act. Select one of the squares and click **Done**. Again choose the brown side to advance and the last Edit Load window will appear. In the Magnitude filed, fill the 50 (kPa) value and click the **OK** button to finish the sequence.

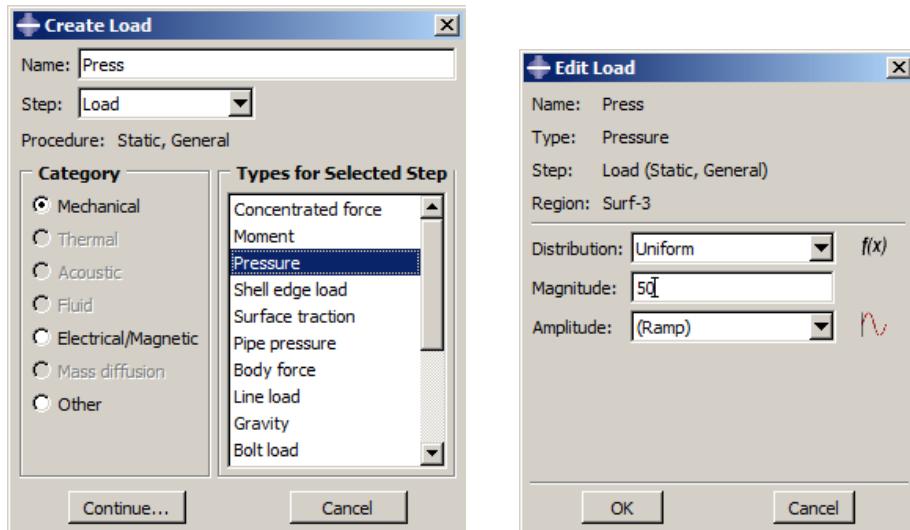


Fig. 4.26: Creating the Load: left) Selecting the general load type and acting step; right) Filling the pressure value

At the end of the creation of both boundary conditions and loads, the model will display them as arrows acting upon it, having different meanings: restraints, body forces or pressure.

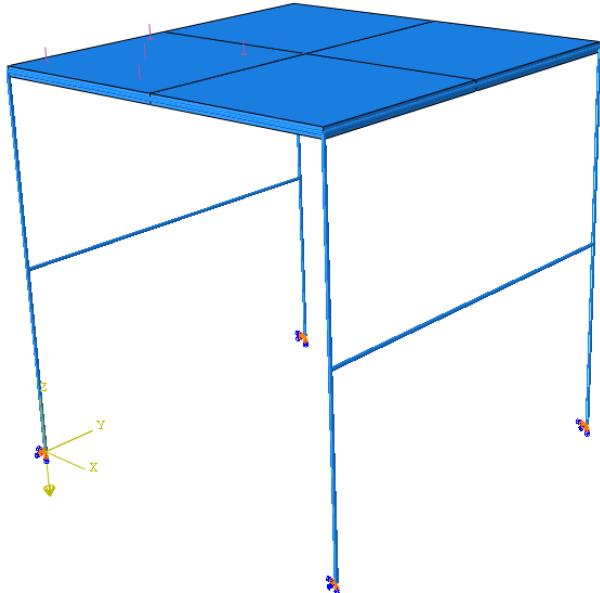


Fig. 4.27: The model loaded and fixed

Advancing to **Module: Mesh**, one can observe that the whole model is pink coloured. This denotes that a triangular (in the case of 2D)/ tetrahedral element geometry will be used. This problem considers the usage of quad

type elements. In order to change the element type, click the **SR**  **Assign Mesh Controls** and, after selecting the whole model, in the Mesh Controls window, check Quad under the Element Shape category and Structured for the Technique. Finally, click **OK**.

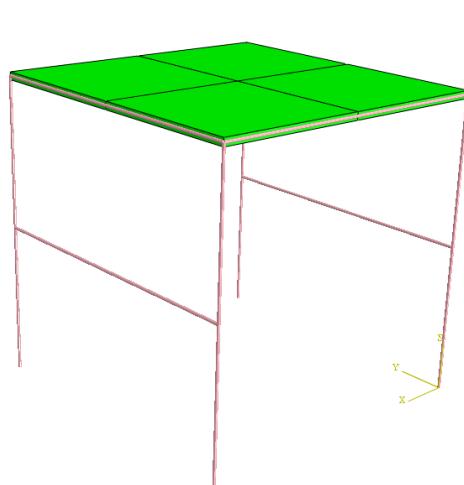


Fig. 4.28: The modified slab Assigned Mesh Controls

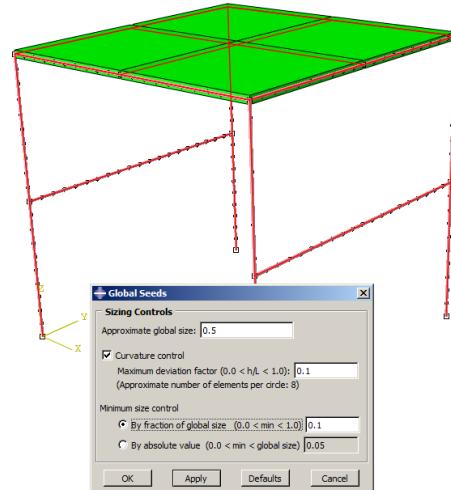


Fig. 4.29: The model selected and the Global Seeds window

Furthermore, it is needed to assign the size of the cells in which the geometry will be divided. Therefore, click the **SR** **Seed Part Instance** button and select the whole model. After clicking the **Done** button, a Global Seeds window will appear. Fill in the field of Approximate global size the value of 0.5(m) and click **OK**. At the end of this operation, multiple white squares will appear along the edges of the structure, displaying the limits of the elements to be created. In order to complete the meshing technique, go to **SR** **Mesh Part Instance**, click **LMB** and select the whole model. Click **Done** in order to end the discretization.

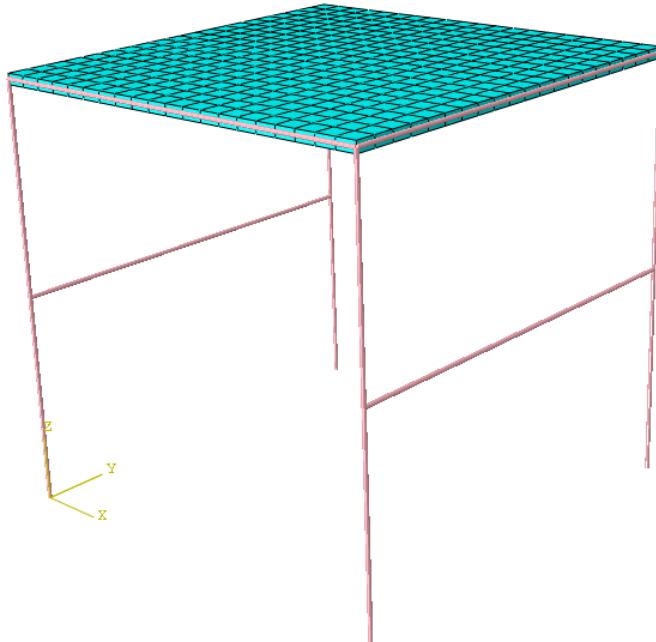


Fig. 4.30: The Meshed Slab

It can be observed that visible elements have been created only on the slab, because it is the only part that expands on at least two directions. The discretization elements exist also on the linear beam, having the size imposed during the seeding phase.

At this moment, the model is ready to be submitted to the calculation phase. Therefore, we advance to the **Module: Job**. Go to **SR**  **Create Job**. A Create Job window will appear in which the name shall be changed from the default “Job-1” to “Frame”. The source is the Model-1, which has been created. Click **Continue ...**. Under the Edit Job window, go to the Parallelization tab and, if the case, select Use multiple processors and change to the number of cores your CPU have, in order to improve (reduce) the calculation period.

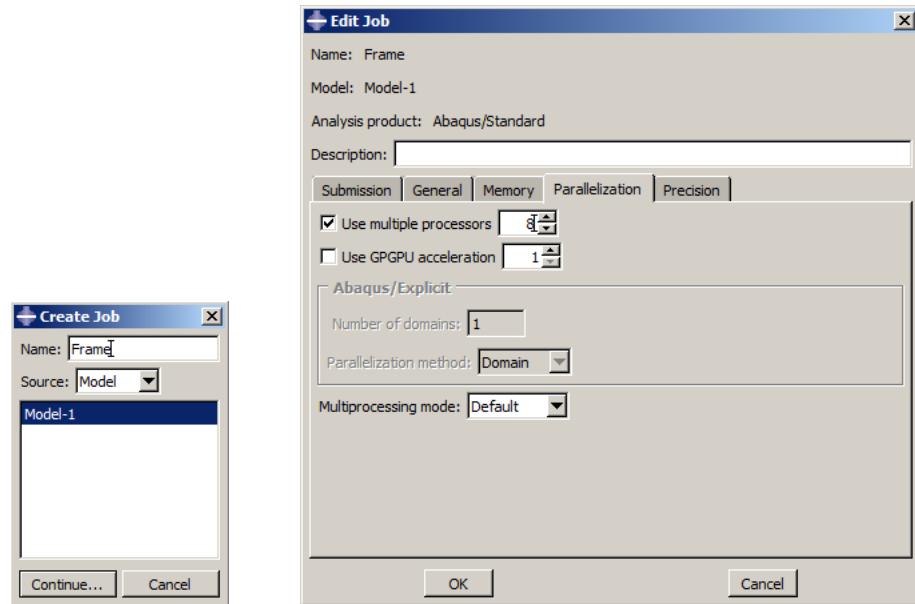


Fig. 4.31: The Create Job window and Edit Job

In order to submit the job to the calculation process, go to **SR** **Job Manager**, make sure the job is selected and **LMB** click the **Submit** button.

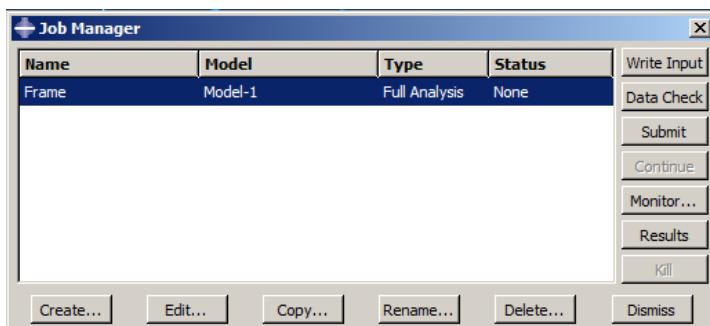


Fig. 4.32: The Job Manager window

On submitting the job to be calculated, the status will change from “None” to “Submitted”, and after finalization of the calculation process, will turn into “Completed”.

In order to visualize the result, in the Job Manager window, click the **Results** button. Now, the interface has moved to the post-processing part of the software, as it can be observed under the module section: **Module: Visualization**. Click the **SR**  **Plot Contours on Deformed Shape** and the model will provide both the contours of the variable (default is S Misses) and the deformed shape of the model (default scale is Auto-compute). In order to change the deformed scale, go to **RS**  Common Options button, click LMB, and the Common Plot Options window will come forth. Under the Deformation Scale Factor group, check Uniform, and fill 1 for the field. Click **OK**. Now the real scaled deformations are plotted.

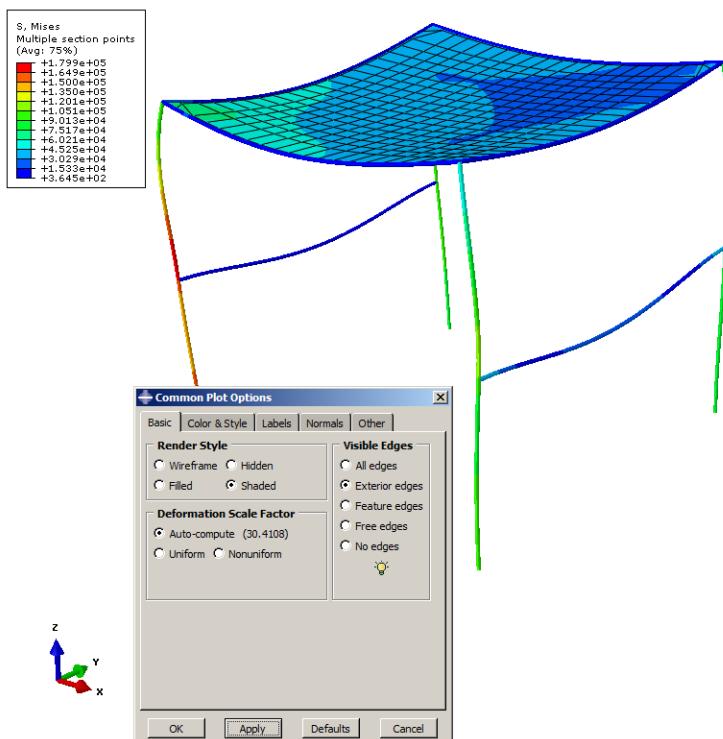


Fig. 4.33: The deformed model and the Common Plot Option window set on Auto-compute Deformation Scale Factor

In order to change the variable type that is displayed, go to the TR pop-down menus , and from the second menu, choose “SF” (Section Forces), instead of “S” (Stresses). In the menu to the right, choose either “SF1” (Section Forces along the 0X direction), “SF2” (Section Forces along the 0Y direction) or “SF3” (Section Forces along the 0Z direction). The legend to the left will display the variation of the forces from minimum to maximum reached values, in the force units conveniently chosen at the beginning of the modelling (kN, in this case).

In order to display the section moments, instead of “SF”, choose “SM” (Section Moments), which is also divided in three vectors, around the axis that acts as rotational support – “SM1”, “SM2” and “SM3”. Another way to choose the output variable is by going to **Result ▶ Field Output...**, which will bring forward the Field Output window, where all the available variables are presented, along with their Description.

Using the **Apply** button, the user may preview the requests made, without the window disappearing, allowing him to easily change the output variable.

4.5 Results interpretation

The user should always bear in mind that the results of different output variables are displayed in their units conveniently chosen at the beginning of the modelling.

It is easily observable that the most loaded vertical pole of the frame is situated near the corner of the square upon which the pressure is acting (Fig. 4.35).

Also, from the displacement point of view, one can observe that the slab’s maximum deflection values are obtained in the centre of the slab, but with a slight extent towards the edges upon the pressure is acting (Fig. 4.36).

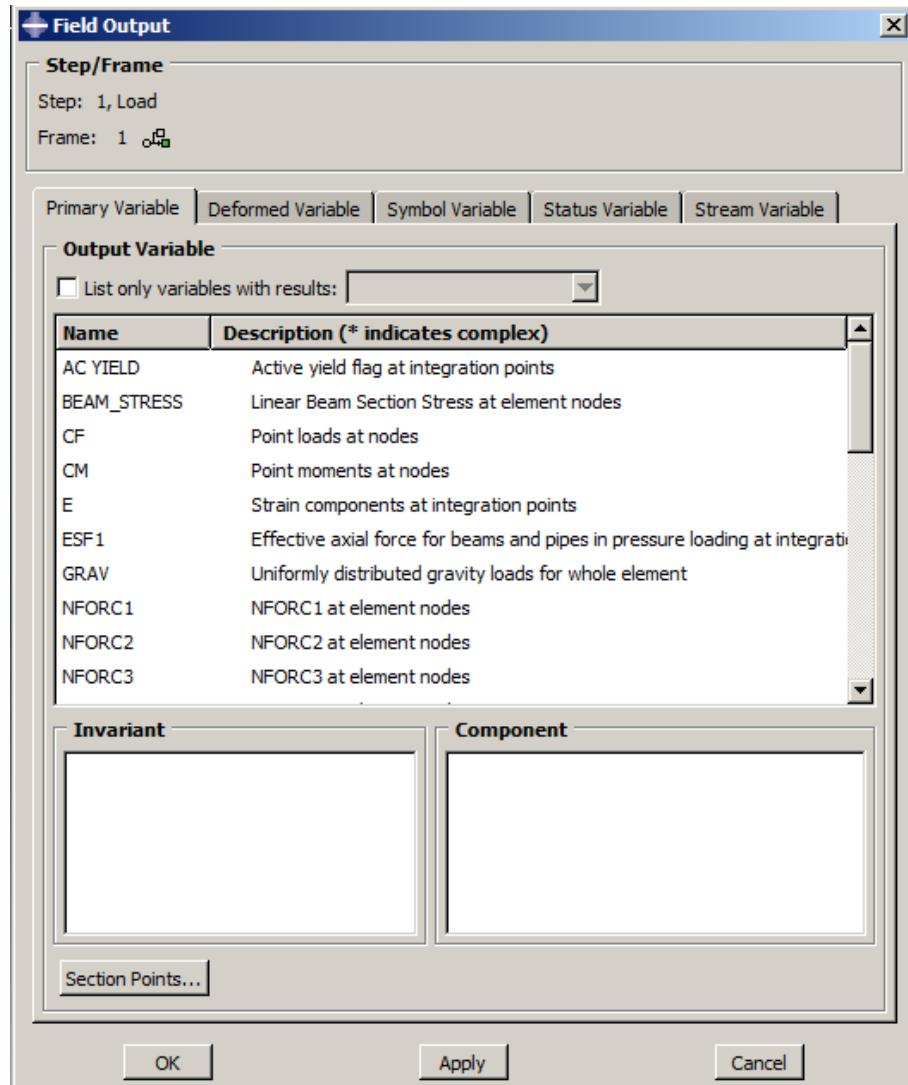


Fig. 4.34: The Field Output window

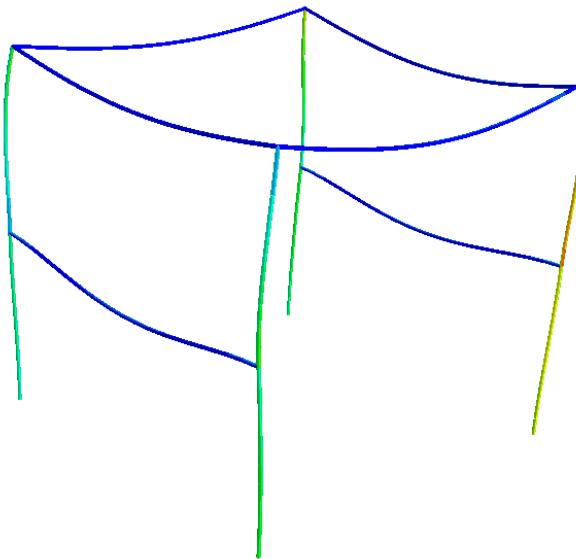


Fig. 4.35: Stress variation on the three dimensional frame

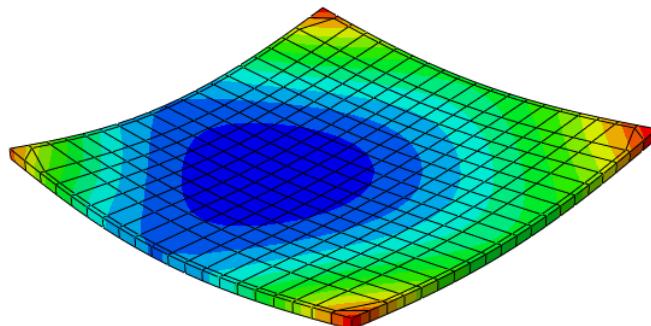


Fig. 4.36: Vertical displacement variation on the slab

4.6 Things to remember

This model introduces the user to the basic utilization of the software, creating and submitting the model to calculation and general overviewing the data obtained following the calculation phase.

Every step presented in this chapter represents a minimal approach to the numerical modelling one can use, further developed during this book.

5. 2D ELASTIC CONTINUOUS PROBLEM

5.1 Keywords

Shell, 2D Planar, Linear Elastic, Plane strain, Field Output processing

5.2 Aims

- Creating parts in ABAQUS
- Defining the linear elastic behaviour of the materials
- Creating and assigning materials to sections
- Creating and assigning sections to parts
- Creating analysis steps and demanding additional results
- Defining the interaction properties between parts
- Defining boundary conditions and loads
- Surface processing and meshing
- Field output manipulation
-

5.3 Problem description

For this problem we will create a simple model that analyses the interaction of a raft with the soil underneath. The model consists of two rectangular parts that are in direct contact. One part will be made out of a material with the elastic parameters of a generic soil type, and the other part will be made out of concrete. Also, the analysis of the settlement shall be done in two separate steps, namely "Geostatic" and "Loading". This will later be useful to determine the settlement given by the raft loading.

The model geometry is given in Fig. 5.1: the concrete raft is 10m long and 0.5m thick and the soil mass is 50m by 50m. The raft is centred on the soil mass and it sits above it.

The concrete part (raft) will be acted by a uniform pressure of 15kPa. The assembly will also be subjected to gravity load ($g=9.81\text{m/s}^2$).

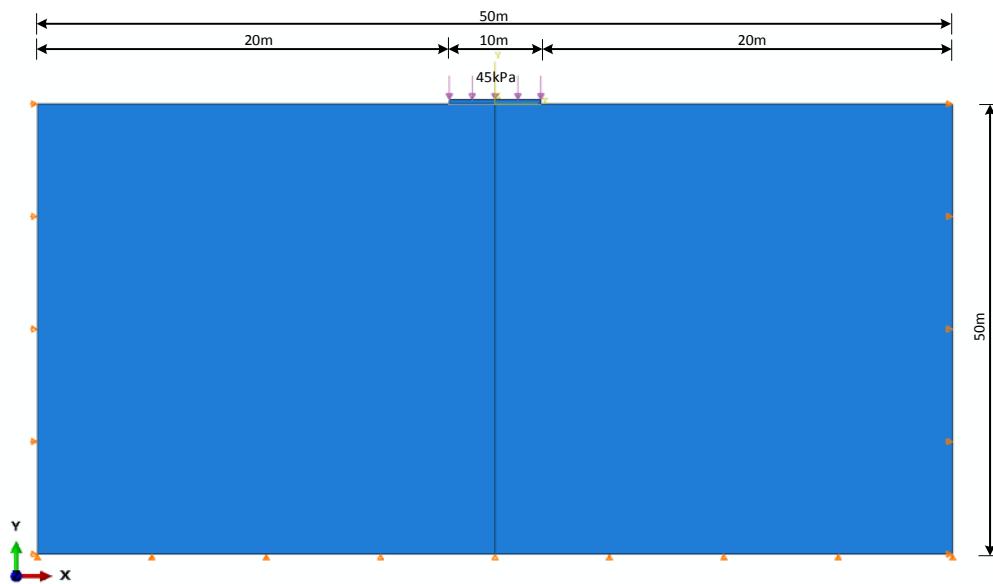


Fig. 5.1: Model geometry

5.4 Solving steps

The geometry of the model may be drafted in a CAD software, though this one in particular is simple enough to be sketched in ABAQUS preprocessing interface. In Module: **Part**, at **Create part**, select **2D Planar** (Modelling Space tab), **Deformable** (Type tab), **Shell** (Base feature).

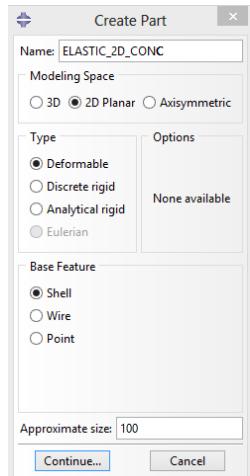


Fig. 5.2: Create part

Name	Description	Status	Create...
ELASTIC_2D_CONC	2D Deformable		Copy...
ELASTIC_2D_SOIL	2D Deformable		Rename...

Fig. 5.3: Part manager menu

Inside the drawing space, the **SR** **Create Lines: Connected** is to be selected using **LMB**. The four corners of the first concrete part will be introduced as coordinates in (X, Y) system:

- (-5, 0) for the first corner;
- (-5, 0.5) for the second corner;
- (5, 0.5) for the third corner;
- (5, 0) for the fourth corner.

In order to end the sketch for the first part (ELASTIC_2D_CONC) the **BR** **Done** button is to be pressed.

The same operations shall be done for creating the second part (ELASTIC_2D_SOIL), with the following coordinates:

- (-50,-50) for the first corner;
- (-50,0) for the second corner;
- (50,50) for the third corner;
- (50,-50) for the fourth corner.

The assembled model (that will be discussed later) will have the coordinate centre (0, 0) at the interface between the two parts, in the middle of the model.

In order to define the elastic material properties of the parts we must move to **Module: Property**, and **SR**  shall be clicked. For the "Concrete", under the **Edit Material** window, at **General: Density**, where we will input the value of 2.50(to/m³). Following this step, at **Mechanical: Elasticity ► Elastic**, the Young's Modulus will be inputted as 27E⁶(kPa) and Poisson's Ratio as 0.25, respectively. In order to end the creation of the material, click **LMB** on the **OK** button (Fig. 5.4).

Similar for the "Soil": density is 1.80(to/m³), Young's Modulus is 10000kPa and Poisson's Ratio is 0.35.

Next, the sections will be created and assigned to the parts. For the raft (part ELASTIC_2D_CONC), click **LMB** on the **SR**  **Create Section** button and select **©Solid**, type Homogeneous. Upon clicking **Continue**, the **Edit Section** pop-up appears where the user shall select "Concrete" for the Material and check **☒** for **Plane stress/strain thickness** (value shall remain 1m).

To assign a section to a part, **LMB** on the **SR**  **Assign Section**, select the part and then click **Done**. In the next pop-up menu the section "Concrete" will be selected under the pop-down menu and the Assignment will be **©From Section**. Similar operations are made for assigning the "Soil" section for the second part. (In order to scroll the displayed parts use the **TR, Part:** ▾).

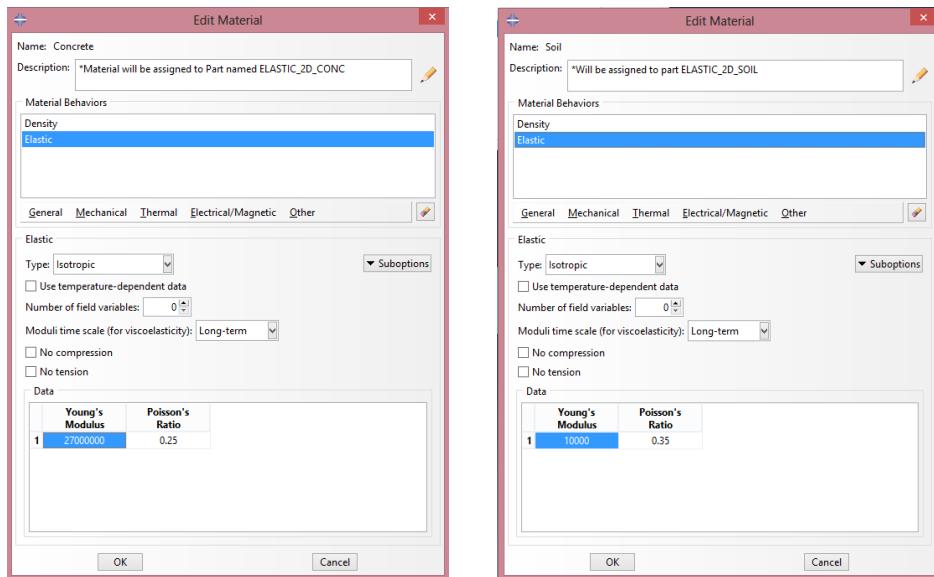


Fig. 5.4: Input of elastic material properties for concrete and soil

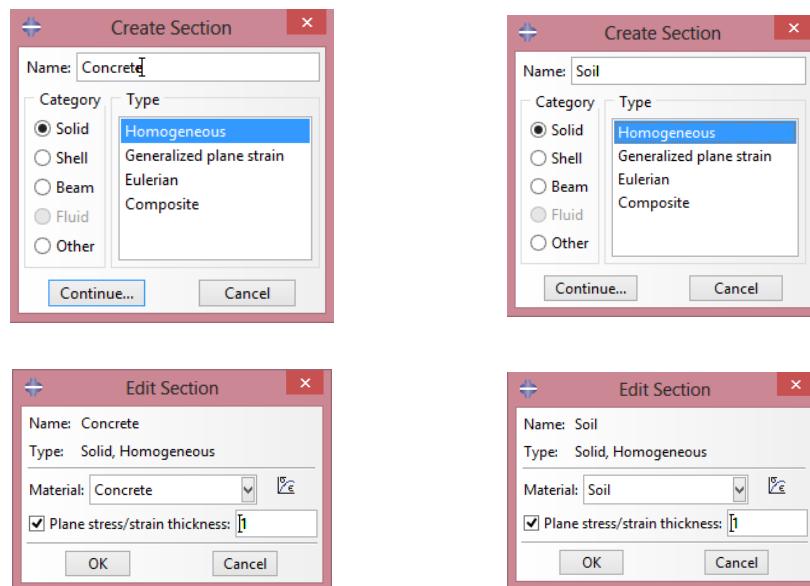


Fig. 5.5: Create section and assigning material for concrete

Fig. 5.6: Create section and assigning material for soil

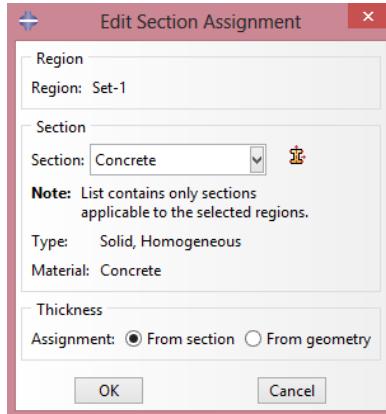


Fig. 5.7: Section assignment for Concrete part

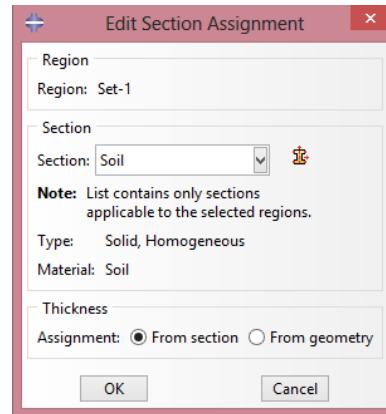


Fig. 5.8: Section assignment for Soil part

Moving to the **Module: Assembly**, in order to insert the two components into the whole model, LMB click the **SR** **Instance Part** button, check **Independent (mesh on instance)** and while holding the **Shift** button, click **LMB** on the two parts in order to bring them to the same instance, as in Fig. 5.9. Finish the procedure by clicking the **OK** button, and the two parts should appear in the correct position in the **Model Space**.

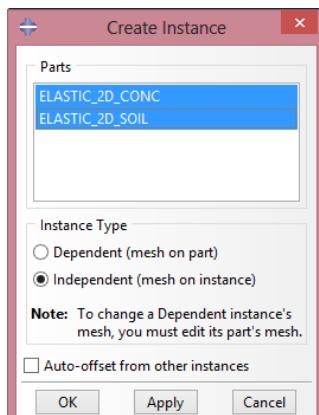


Fig. 5.9: Creating the Independent instance

If one needs to obtain the variation of one given **Field Output** with the Y coordinate (or any other direction), as we will see later, one option that eases the process is to create a **Partition**. In this case, we shall make a partition through the middle of the model (from point [0, 0.5] to point [0, -50], cutting both parts, namely the concrete raft and the soil massive). This may be done by LBM click  **Partition Face: Sketch**, and selecting, in turns, each part and using  sketch the partition geometry.

The **Partition** also enables the user to select different sub-surfaces of a given part, making it useful in assigning unsymmetrical loads, boundary conditions and so on. If one wishes to check, identify or edit these, they can be found under **|Model| ▶ Model Database ▶ + Models ▶ + "your model" ▶ Assembly ▶ Features**. In this example, they should be found as **Partition face-1** and **Partition face-2**.

The following operation will be the **Module: Step**. Click on the  **Create Step** button, and name the first calculation step, named “Geostatic”. The default Procedure type is General and the type of calculation step is Static, General. Click the **Continue ...** button, and a new window will appear. In the description field, write “Geostatic + Raft” (or any other thing that helps you identify it if needed). The time period will remain 1 (as 1second). This implies 100% of the forces acting on the system to be applied to the structure, as the calculation is not time-dependent. Click the **OK** button, in order to complete the creation of the calculation step.

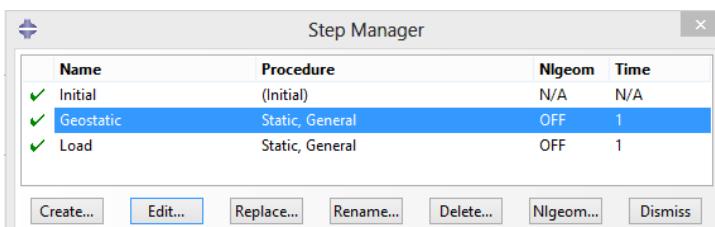


Fig. 5.10: Calculation steps

Repeat the procedure for the second step, named "Load" with no further interfering with the Step Editing fields (except the description field where you may write "Loads application"). The **Step Manager** should look like Fig. 5.10.

The default calculation output should cover the necessary Field Output variables (stresses, strains, forces, displacements and contact).

At the **Module: Interaction**, click the **SR**  **Create Interaction Property** button, with **Contact** Type property (Fig. 5.11). After clicking **Continue ...** the Edit Contact Property window will appear, in which the following contact property options will be selected:

- **Mechanical Tangential Behaviour - Rough** - at Friction formulation (selected from the pop-down menu) as depicted in Fig. 5.12;
- **Mechanical Normal Behaviour - "Hard" Contact** - at Pressure-Overclosure (selected from the pop-down menu) while the Constraint enforcement method shall remain as **Default**, as depicted in Fig. 5.13.

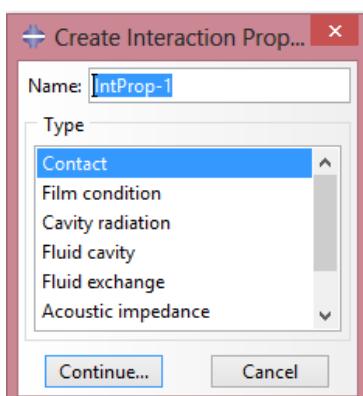


Fig. 5.11: Creating interaction property

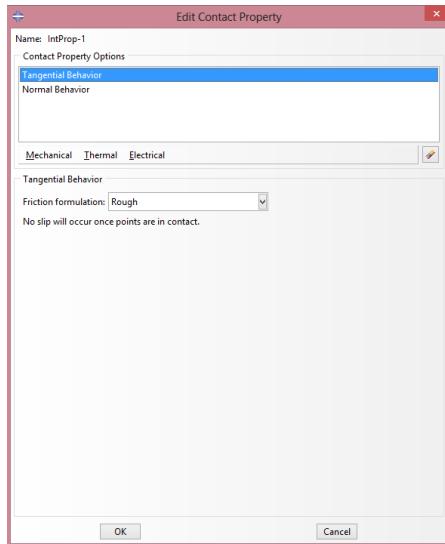


Fig. 5.12: Definition of the contact tangential behaviour

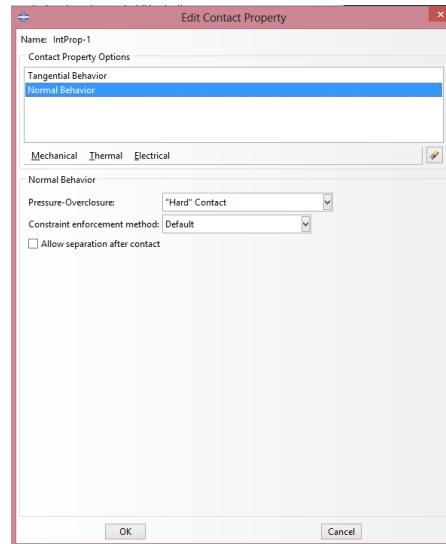


Fig. 5.13: Definition of the contact normal behaviour

Next, by clicking on **Create Interaction** the user must select the **Interaction Type**. We shall create a **Surface-to-surface contact (Standard)**, from **Step: Initial** (will be propagated throughout the following steps).

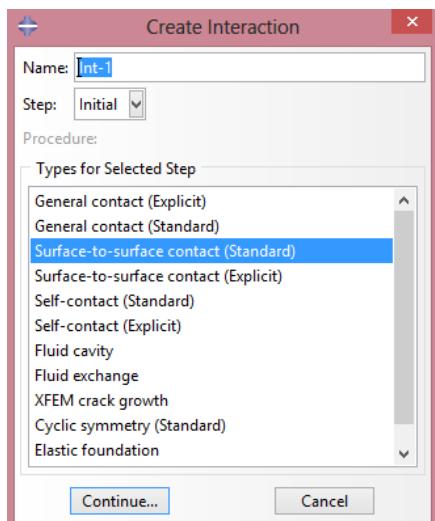


Fig. 5.14: Creating interactions

After clicking the **Continue ...** button, one must **select the master surface** by selecting the top edge of the Soil Mass part using **LMB**. In the **BR** the **individually** scroll down type must be active, check Create surface: **m_Surf-1** (default name).

The **BR** should look like Fig. 5.15. After selecting the master surface and clicking **Done** in the **BR**, the user must choose the slave surface type (click **Surface** button Fig. 5.16) and then select the slave surface which will be the bottom edge of the raft. For graphically selecting it, one must go to **TR** Remove Selected button to remove the master surface from the viewport, click on the bottom edge of the raft and then click on **Done**. The **BR** should look like in Fig. 5.17. Press **TR** Replace All button to render the previously removed master surface in the viewport.

When the **Edit Interaction** window pops-up just click the **OK** button as there is nothing to modify.

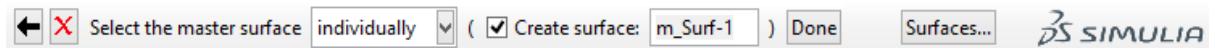


Fig. 5.15: BR - Select the master surface

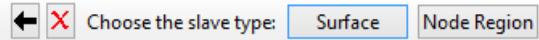


Fig. 5.16: BR - Select the slave type



Fig. 5.17: BR - Select the slave surface

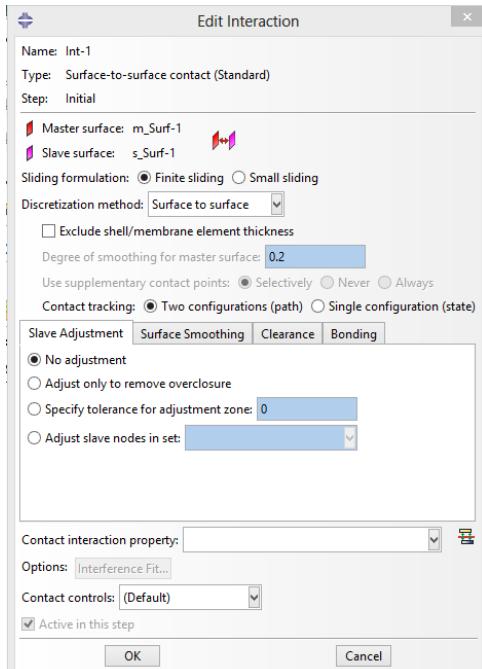


Fig. 5.18: Edit Interaction pop-up window

In order to input the boundary conditions and loads, we now must go to **Module: Load** and click the RS Create Boundary Condition button.

The Create Boundary Condition window will pop-up: name this restraint if you wish, verify the step in which it acts is “Initial” and in the category type select Mechanical, while for the Types for Selected Step choose Displacement/Rotation.

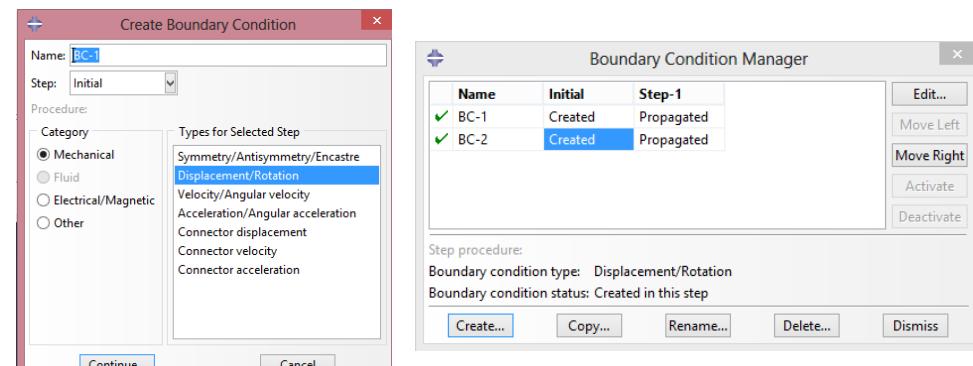


Fig. 5.19: Creating and assigning the boundary conditions

Clicking **Continue ...** will advance to the next step, where the user is prompted to Select the regions for the boundary condition. Select both vertical edges of the Soil Mass using **<Shift>** and click **Done**. The pop-out window prompts the Boundary Conditions, check U1 (displacement on X axis).

Similar procedure will be followed for restraining displacements on the Y axis, selecting only the bottom edge of the Soil Mass.

Next, the loads shall be created. Go to the **RS**  **Create Load** button, click it, and the Create Load window will appear. Name the load “g” for gravitational, make sure the set step is “Geostatic”, then choose Mechanical for the load category and Gravity for the Types for Selected Step. Click **Continue ...** and a second window (Edit Load) appears in which it is required to provide the domain on which the gravity load should be. Therefore, as it can be observed, the region is the whole model, by default, and in the field near the Component 2 fill with the $9.81 \text{ (m/s}^2)$. Amplitude shall remain as (Ramp).

In order to create the pressure on the raft, click again on the **RS**  **Create Load** button, and now, select Pressure under the Types for Selected Step options menu, having the name “Loading”. Make sure that the Step is set to “Load”.

Clicking **[Continue ...]** will bring forth the model and the software demands the user to pick the surfaces on which the pressure will act. Select the top edge of the raft and click **[Done]**.

In the Magnitude filed in **Edit Load** pop-up window, input 45 (kPa) and click the **[OK]** button to finish the sequence.

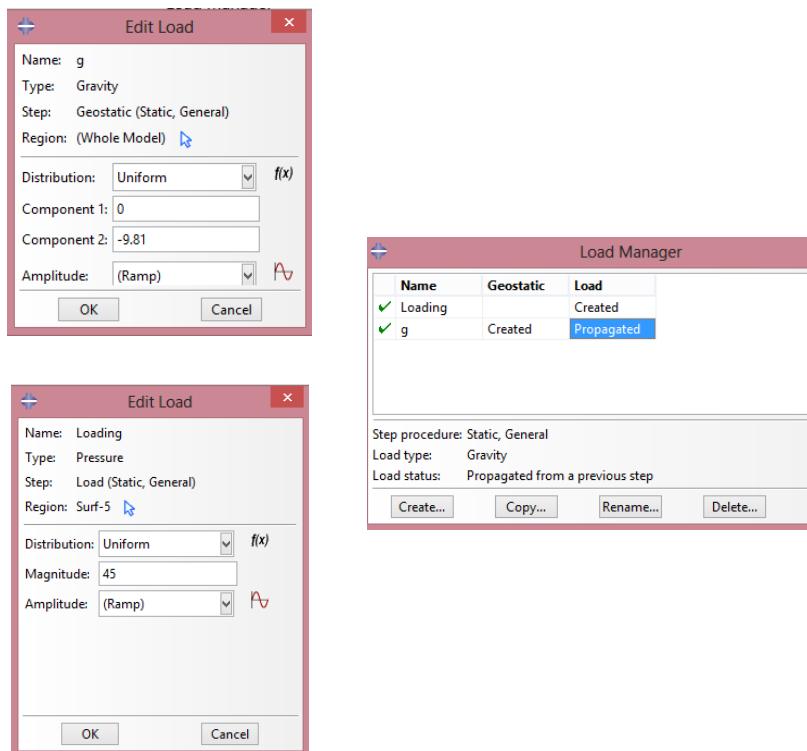


Fig. 5.20: Creating the loads

After creating the whole model in terms of geometry, boundary conditions, loads and analysis steps, it should be displayed as in Fig. 5.23.

Advancing to **Module: Mesh**, one can observe that the whole model is pink coloured. This denotes that a triangular element geometry will be used. This

problem considers the usage of quad type elements. In order to change the element type, click the **SR**  **Assign Mesh Controls** and, after selecting the whole model, in the Mesh Controls window, check **Quad** under the Element Shape category and **Structured** for the Technique. Finally, click **OK** (Fig. 5.22).

Furthermore, it is needed to assign the size of the cells in which the geometry will be divided. Therefore, click the **SR**  **Seed Part Instance** button and select the whole model. After clicking the **Done** button, a Global Seeds window will appear. Fill in the field of Approximate global size the value of 0.25 (as in 0.25m) and click **OK** (Fig. 5.21). At the end of this operation, multiple white squares will appear along the edges of the structure, displaying the limits of the elements to be created. In order to complete the meshing technique, go to **RS**  **Mesh Part Instance**, click **LMB** and select the whole model. Click **Done** in order to end the meshing process.

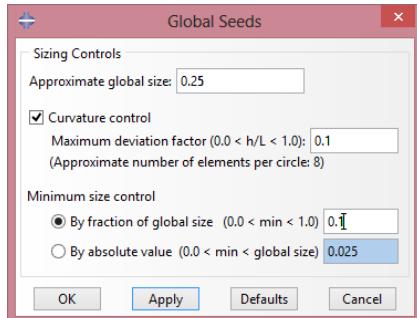


Fig. 5.21: Assigning Global Seeds

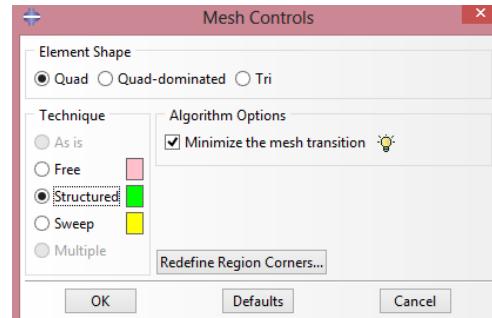


Fig. 5.22: Assigning the Mesh Controls

The Approximate global size for the Global Seeds may be attributed separately for the soil mass and for the concrete raft, depending on the desired refinement of the mesh. As you may know by now, this will affect to some degree the precision of the results.

In this manor, one may assign an approximate global size for the global seed in the case of the raft of up to 0.5 (as the raft is 0.5m thick), while for the soil it may be of 1.0 (multiple of 0.5), so that there are common intersection points in the differently seeded meshes.

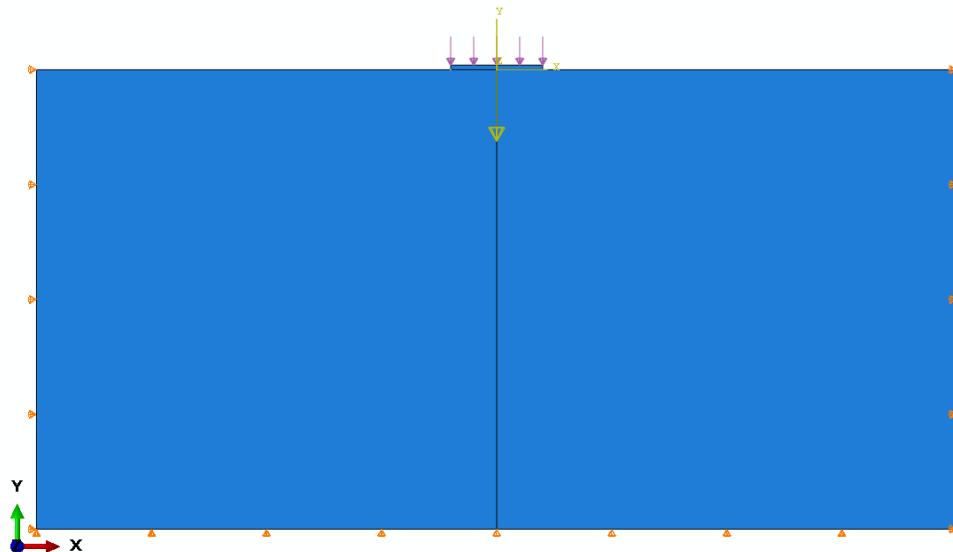


Fig. 5.23: Fully defined model

After the meshing of the analysis model is complete, move to **Module: Job**.

Go to SR Create Job. A Create Job window will appear in which the name shall be changed from the default “Job-1” to “ELASTIC_2D”. The source is the Model-1, which has been created up to this point. Click **Continue ...**. Under the Edit Job window, go to the Parallelization tab and, if the case, select Use multiple processors and change to the number of cores your CPU have, in order to improve (reduce) the calculation period.

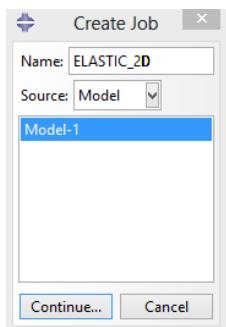


Fig. 5.24: Creating the analysis Job

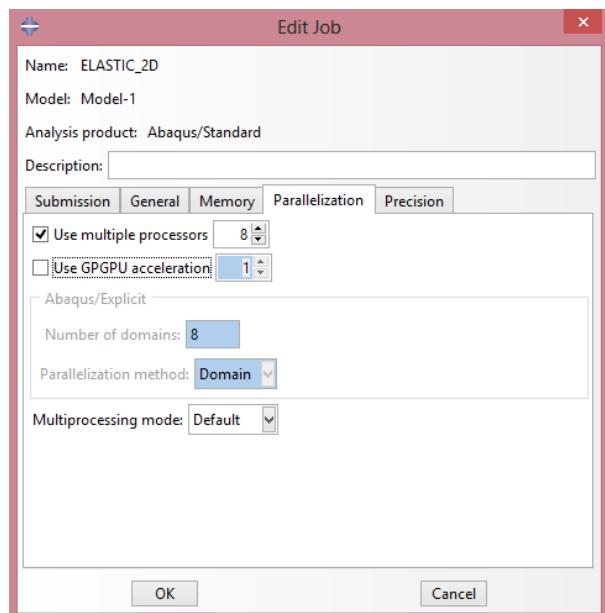


Fig. 5.25: Editing the Job - Parallelization

On submitting the job to be calculated, the status will change from “None” to “Submitted”, and after finalization of the calculation process, will turn into “Completed” (Fig. 5.26).

If clicking the **Monitor** button one can see the status of the analysis. In Fig. 5.27, one can see that the two analysis **Steps** were completed successfully. No matter the analysis type, the column named "Step time/LPF" shows the status of the calculations for each given step. If the step time is equal to the time period inputted in the **Edit Step** window under **Module: Step** (previously we declared this value to be equal to 1, or 1second, for both steps), the analysis is completed and it has converged. So, in this case, total time will be equal to 2.

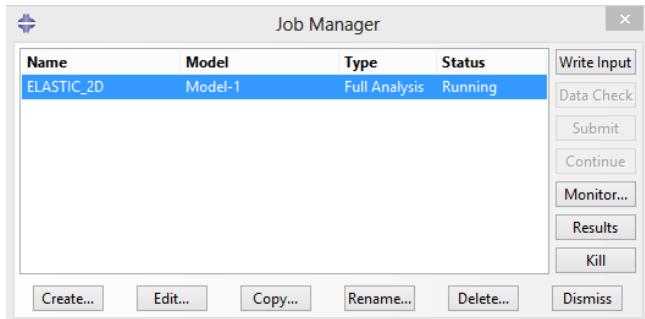


Fig. 5.26: Job manager window

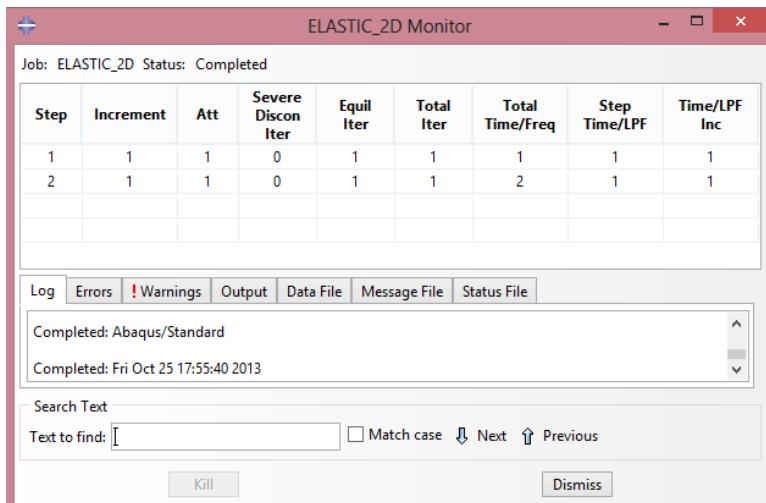


Fig. 5.27: Analysis Monitor window

In order to visualize the result, in the Job Manager window, click the **Results** button. Now, the interface has moved to the post-processing part of the software, as it can be observed under the module section: **Module: Visualization**. Click the SR **Plot Contours on Deformed Shape** and the model will provide both the contours of the variable (default is S Misses) and the deformed shape of the model (default scale is Auto-compute). In order to change the deformed scale, go to RS **Common Options** button, click **LMB**, and the Common Plot Options window will come

forth. Under the Deformation Scale Factor group, check Uniform, and fill 1 for the field. Click **OK**. Now the real scaled deformations are plotted.

Mind that you can navigate through the Steps/Frames using **TR** , which in turn affect the results accordingly. For now we will check the Stresses (Mises) and the settlement (U2) in the second Step named "Load".

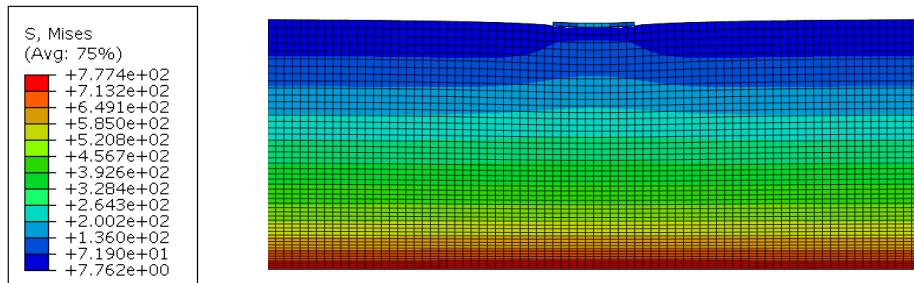


Fig. 5.28: Stresses (Mises) displayed as Contours on Deformed Shape and colour legend

In order to change the variable type that is displayed, go to the **TR** pop-down menus , and from the second menu, choose "U" (Spatial displacement at nodes), instead of "S" (Stresses). In the menu to the right, choose "U2" (Spatial displacement at nodes on the OY direction). "U1" will display the spatial displacement at nodes on the OX direction.

The legend to the left will display the variation of the displacements from minimum to maximum reached values, in the length units conveniently chosen at the beginning of the modelling (meters in this case).

Next, suppose we need to know the settlement of the raft when the 45kPa pressure is applied in the second step, or in other words, the relative settlement between the two steps. Considering that in the first step we loaded the whole model with gravity only, which resulted in an initial settlement that we may disregard (consider it zero), we just need to find out the difference between "U2" in the "Geostatic" step from the "U2" in the "Load" step. In order to do

this, go to **Tools ▶ Create Field Output ▶ From fields...** A pop-up window should appear, like the one in Fig. 5.29.

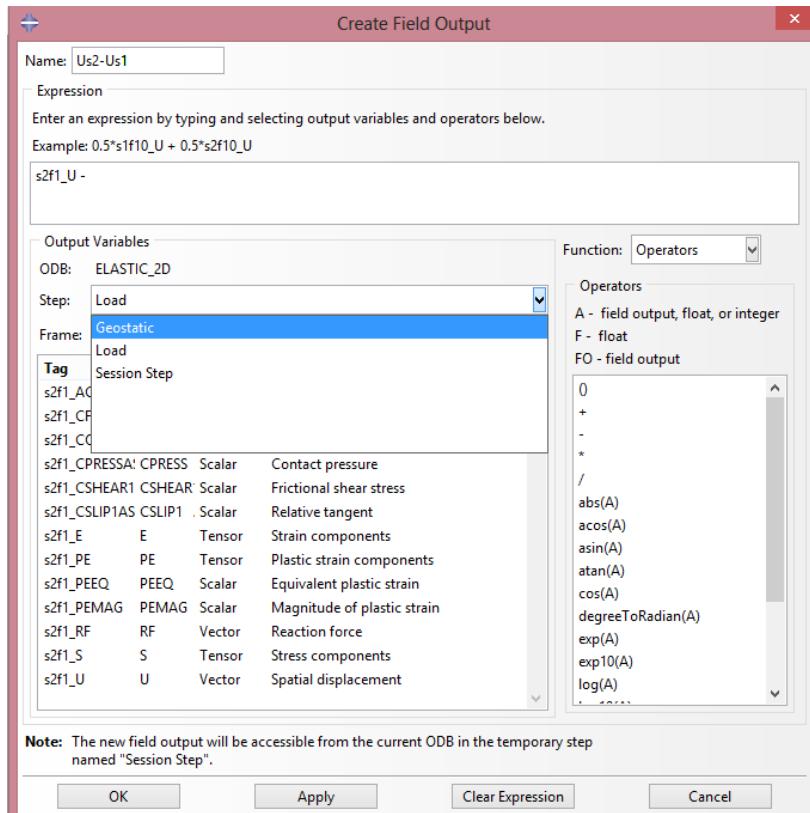


Fig. 5.29: Create Field Output window

Edit the name field if you wish to do so (eg. Us2-Us1). In the Expression field you must write the equation that describes the difference between the displacements at nodes from step "Load" and the displacements at nodes from step "Geostatic". This shall be accomplished as follows:

- **LBM** in the pop-down field named **Step**: that is found in the Output Variables and select **Load** (make sure that the **Frame** pop-down has Step Time = 1.000 which means that is at the end of the calculation for that step);
- **LMB** the **s2f1_U** tag in the field list (which means the spatial displacements vector U in step 2, frame 1);

;

- **LMB** the " - " sign in the Operators field on the right (Operators must be selected from the Function pop-down);
- **LBM** in the pop-down field named **Step**: that is found in the Output Variables and select **Geostatic** (make sure that the **Frame** pop-down has Step Time = 1.000 which means that is at the end of the calculation for that step);
- **LMB** the **s1f1_U** tag in the field list (which means the spatial displacements vector U in step 1 and, 1);
- **LMB** the **Apply** button and then **Cancel**.

Always make sure that you use the operators from the left sub-box in the Create Field Output window and not the keyboard.

In order to recall the field output we have just described, we must go and click the TR  **Field Output Dialog**, LMB on  **Session Frame** and select **Session Step (step for Viewer non-persistent fields)** from the list. Click **OK**. In the subsequent window named **Field Output** (select the field you have just created (Us2-Us1) and from the **Component** window select U2 and click **OK**. Once this actions are performed, you should be able to see the node displacements as in Fig. 5.30.

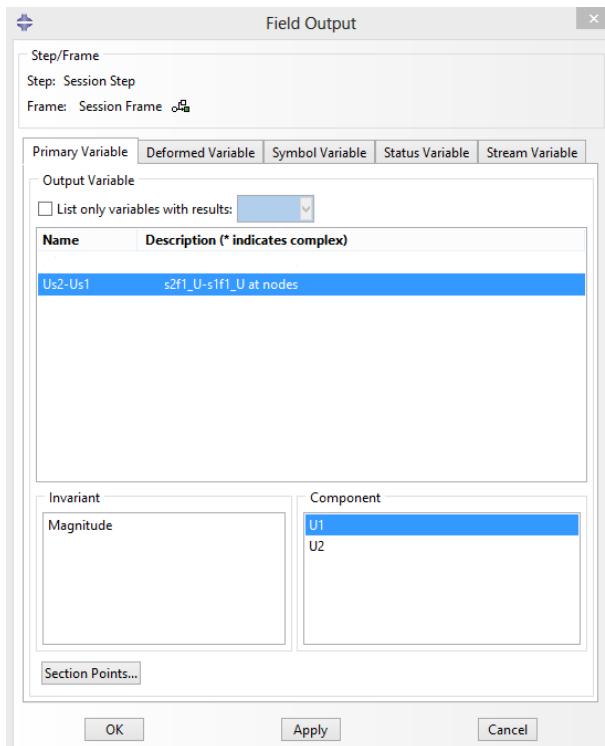


Fig. 5.30: Field Output - component selection

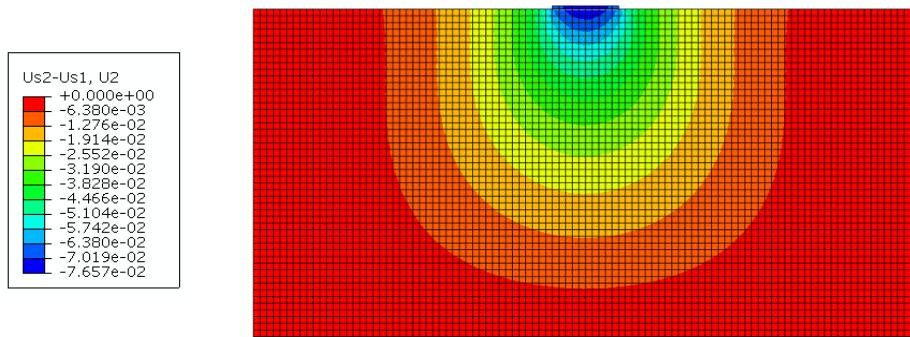


Fig. 5.31: Relative U2 nodal displacements

Notice that the active zone of the foundation is visible now, unlike the field output from step "Loads".

Now, we suppose that one may want the settlement data displayed as a graph or as tabular data. In this case, we must click **View Cut Manager** (the icon next to  button) and in the window that pops-up check **X-Plane** and slide the position cursor (or enter the value directly) to 0. The displayed model in the viewport should look like Fig. 5.32.

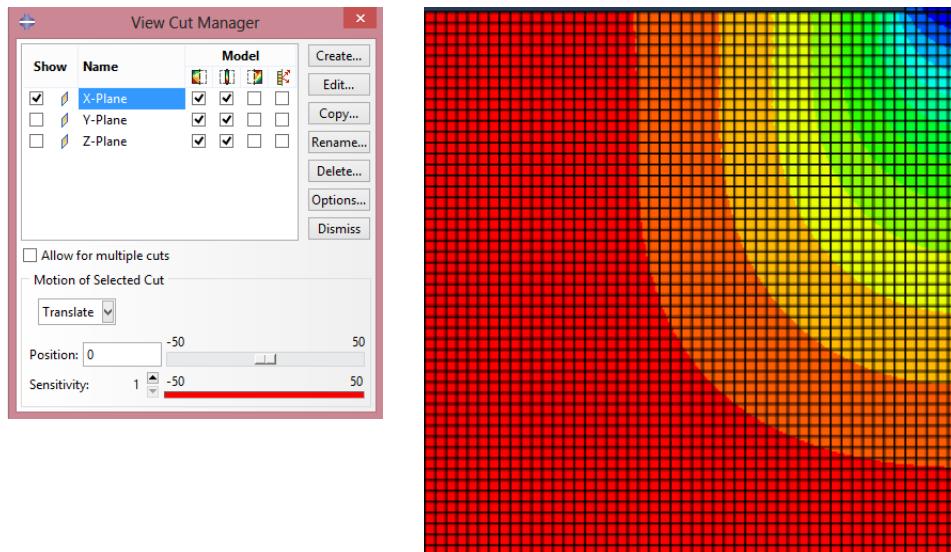


Fig. 5.32: View Cut

Next, go to **Tools** ▶ **Path** ▶ **Create...** check **Node list** and name it (default name is Path-1). Select in the following pop-up window under Part Instance the ELASTIC-2D_SOIL-1 (the part on which we want to take out the data), and click the **Add before** or **Add after** button (it does not really matter), and then, while holding < **shift** >, click in the viewport on the extreme nodes of the symmetry axis against X axis.

In other words, select point 2(0,0) and 1(0,-50). Click **Done** when you finish the selection.

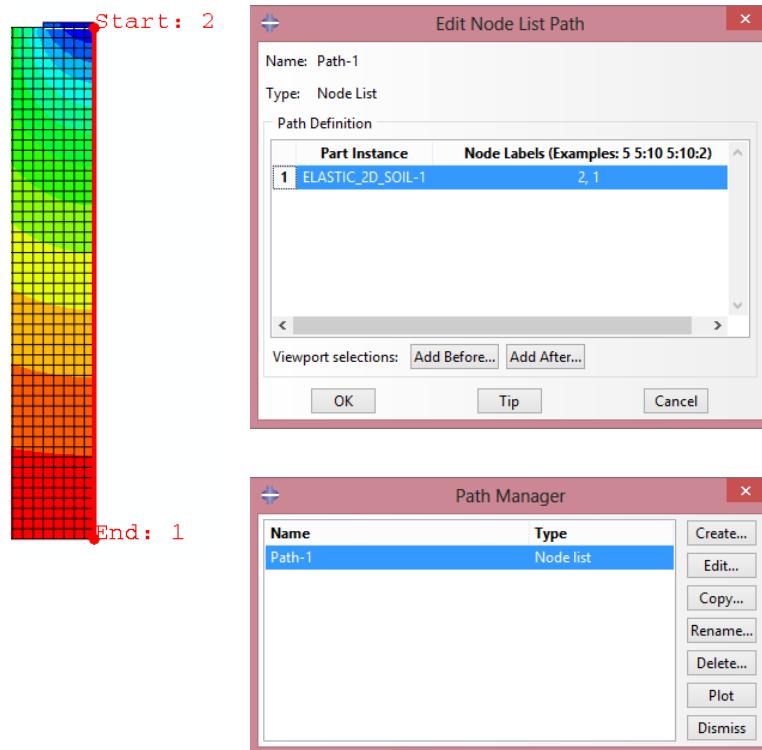


Fig. 5.33: Creating a Node List Path

In the following step, go to **Tools** ▶ **XY Data** ▶ **Create...** check Path and click the **Continue ...** button. In the pop-up window that appears select "Path-1" (default name of node list mentioned above), click Undeformed, click Path point and Include intersections, click True distance. The window should look like Fig. 5.34.

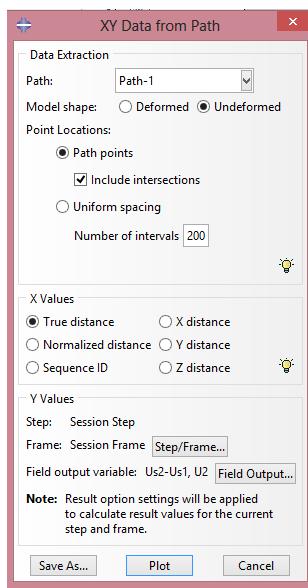


Fig. 5.34: Creating XY Data from path

Note that XY data will always be extracted at points that define a path, and may be extracted at the intersection between the path and the element faces or edges. In other words, Include intersections will improve the XY data density.

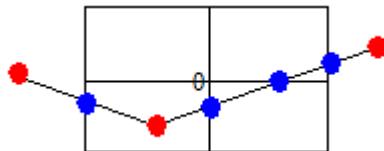


Fig. 5.35: Include intersections

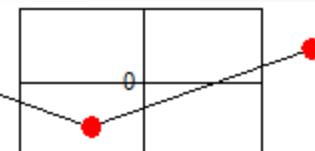
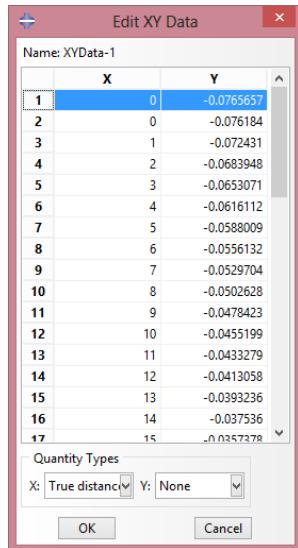


Fig. 5.36: Include intersections

Next you may click **Save As ...** to save the XY data, but only for the current session. The default save name is XYData-1. Once you click **OK** you will find it in the **XY Data Manager**. From here you can **Edit...**, **Copy ...** or **Plot** it. From the Edit XY Data window one may copy by **LMB** dragging or **<shift> + LMB** on the first row of X column and **LMB** on the last row of Y column and press **<Ctrl> + <C>** to store it in the Clipboard.



The dialog box is titled 'Edit XY Data' and has a table header 'Name: XYData-1'. The table contains 17 rows of data with columns 'X' and 'Y'. The first row is highlighted. At the bottom, there are 'Quantity Types' dropdowns set to 'True distance' for X and 'None' for Y, along with 'OK' and 'Cancel' buttons.

	X	Y
1	0	-0.0765657
2	0	-0.076184
3	1	-0.072431
4	2	-0.0683948
5	3	-0.0653071
6	4	-0.0616112
7	5	-0.0588009
8	6	-0.0556132
9	7	-0.0529704
10	8	-0.0502628
11	9	-0.0478423
12	10	-0.0455199
13	11	-0.0433279
14	12	-0.0413058
15	13	-0.0393236
16	14	-0.037536
17	15	-0.0357378

Fig. 5.37:Copying the tabular data

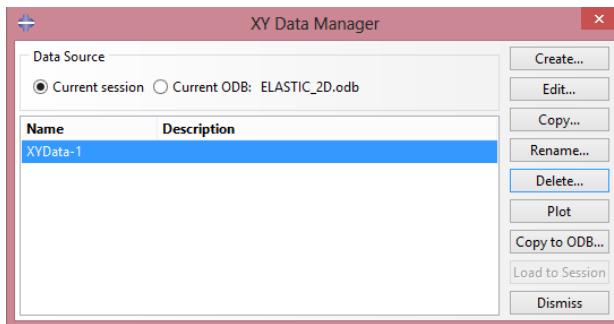


Fig. 5.38: XY Data Manager

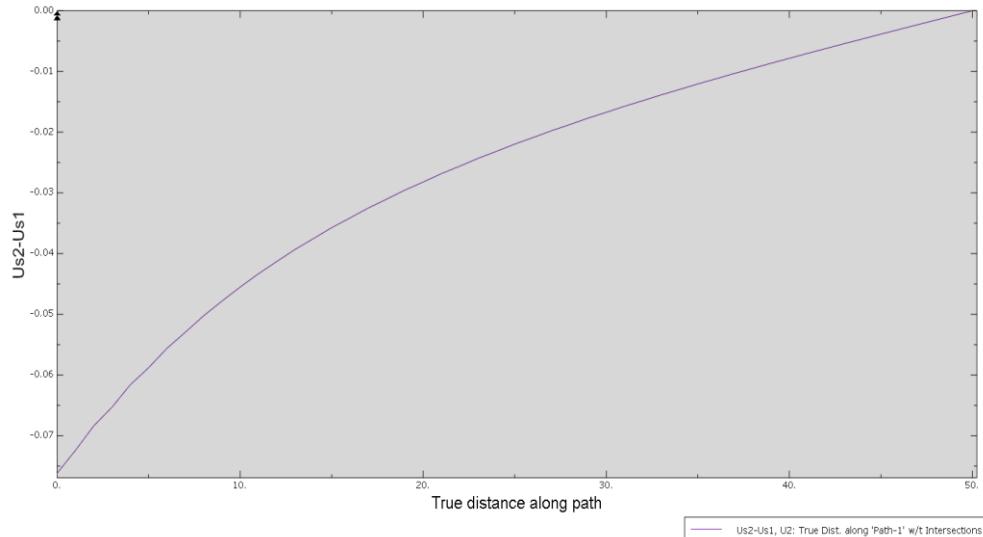


Fig. 5.39: XY Data Plot

In Fig. 5.39 one can see the plotted settlement (the relative node displacement on Y direction between the two steps of the analysis) against Y axis.

5.5 Results interpretation

The user should always bear in mind that the results of different output variables are displayed in their units conveniently chosen at the beginning of the modelling.

User defined Field Outputs are of great importance in post processing, as Fig. 5.31 clearly depicts.

5.6 Things to remember

This model introduces the user to basic 2D modelling and post-processing manipulation.

Also, it introduces the user to defining loads that may be created and propagated through different analysis steps.

Every step presented in this chapter represents a minimal approach to the numerical modelling one can use, further developed during this book.

6. 3D ELASTO-PLASTIC CONTINUOUS PROBLEM

6.1 Keywords

Elasto-plastic material definition, Mohr-Coulomb, contact discretization, element/node sets

6.2 Aims

- Creating an elastic-perfect plastic material model, using Mohr-Coulomb yielding criterion
- Manually inducing the initial stress state of the lithological column
- Manually seeding the soil layers
- Creating element/node sets, usable later, during the post-processing module

6.3 Problem description

The problem of the 3D elasto-plastic model consists of a reinforced concrete raft, loaded with an axial force and moment, which will be transformed into linearly varying pressure. The raft is positioned on a double layer lithology, represented by a cubical soil mass.

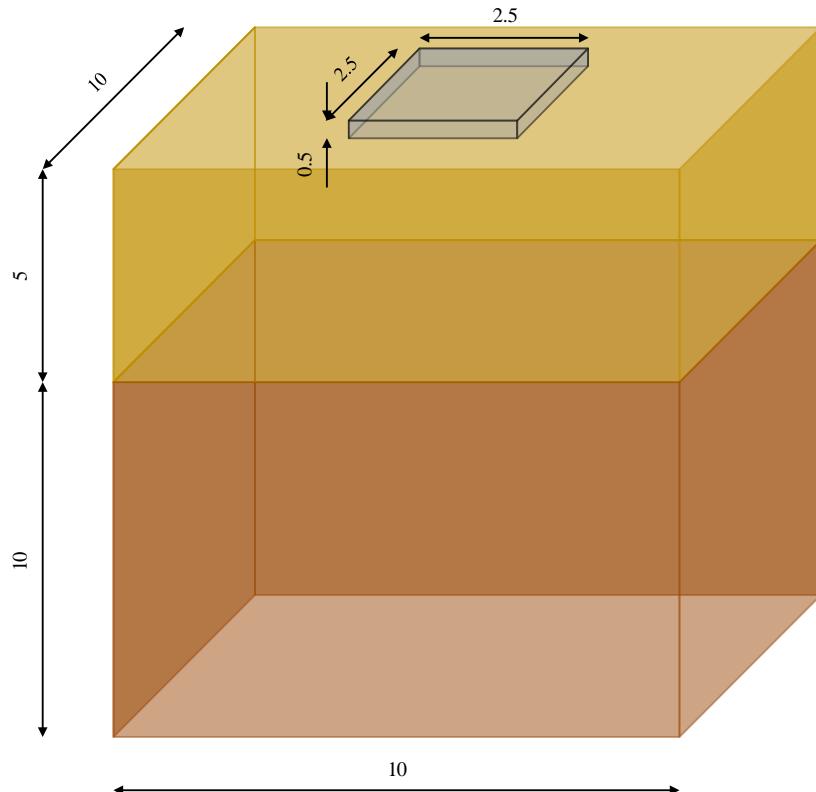


Fig. 6.1: The considered model geometry

The raft is a 2.5 by 2.5m wide slab, having a 0.5m thickness. The two layers are 10 by 10m wide and 5m, respectively 10m thick (see Fig. 6.1). The considered load is composed of an axial load $N=250\text{kN}$ and a bending moment $M=350\text{kN}\cdot\text{m}$.

6.4 Solving steps

First of all, the user has to create the geometry using a CAD software. In order to achieve all the steps the author desires to present, export in the same file part the two cubes representing the soil layers and as a second file – the slab. The two files may be exported as ACIS (*.sat) geometry files type.

Under the **Module: Part**, import the two parts by following the **File ► Import ► Part...** path. For the slab, select the Import all parts, Create individual parts in the Name-Repair tab, while in the Part-Attributes tab, check the 3D and Type Deformable options. Regarding the lithology part, which, as presented before has to contain the two layers in the same file, select Import all parts and check Combine into single part, while putting a tick mark on Retain intersecting boundaries (see Fig. 6.2).

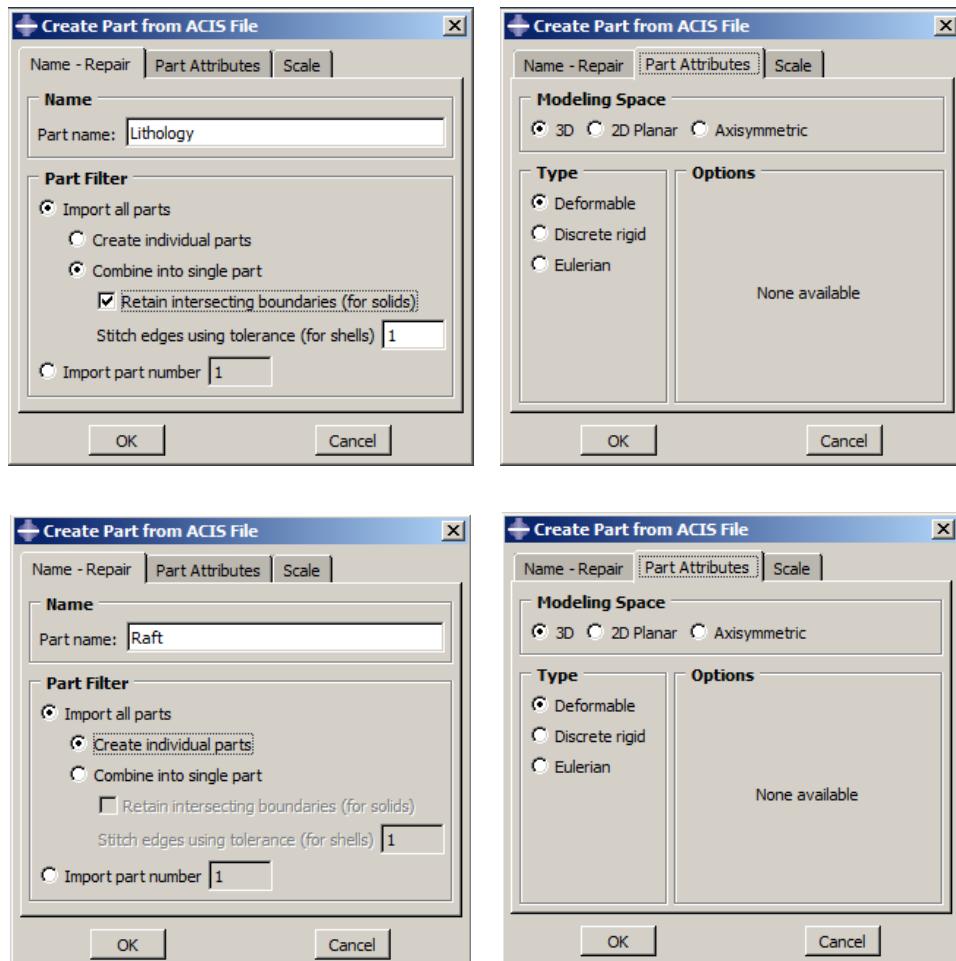


Fig. 6.2: Importing the CAD geometry: top) the two lithological layers; bottom) the raft

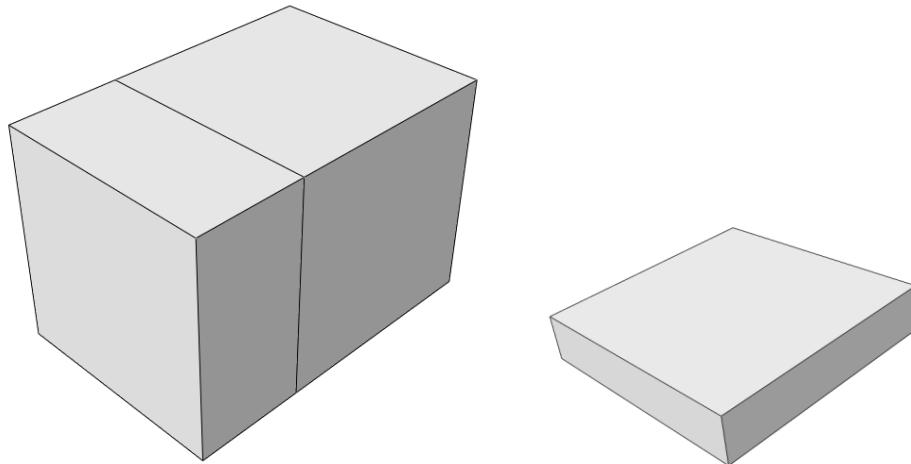


Fig. 6.3: The model's imported geometry: left) the lithology; right) the slab

Moving to the **Module: Property**, the user has to define the material properties, create sections using this material and assign it to the two parts.

In the case of the soil layers, select the following properties in order to obtain a linear elastic-perfect plastic material (also see):

- “Soil Layer 1” (top): input the volumetric mass of 1.8(to/m³), Young Modulus of 12000(kPa), and a Poisson ratio of 0.35; regarding the plastic behaviour, go to **Mechanical ► Plasticity ► Mohr Coulomb Plasticity**, which will create a modified Mohr Coulomb plastic model; under the Plasticity tab, input the value of 15(°) for the Friction Angle and 0(°) for the Dilation Angle; moving to the Cohesion tab, input 45(kPa) for the Cohesion Yield Stress and 0(-) for the Absolute Plastic Strain;
- “Soil Layer 2” (bottom): fill in the values of 2.0(to/m³), Young Modulus of 25000(kPa) and Poisson ratio of 0.27 under the Elastic property tab, while for the Mohr Coulomb plastic use the 27(°) for the Friction Angle, 5(°) for the Dilation Angle, 5(kPa) in the case of the Cohesion Yield Stress and an Absolute Plastic Strain of 0(-).

The slab will be made of reinforced concrete and the problem will consider only a linear elastic behaviour of the material, therefore, fill the following values: 2.5(to/m³), a Young Modulus of 2.7E7(kPa) and a Poisson ratio of 0.2, under the Elastic property.

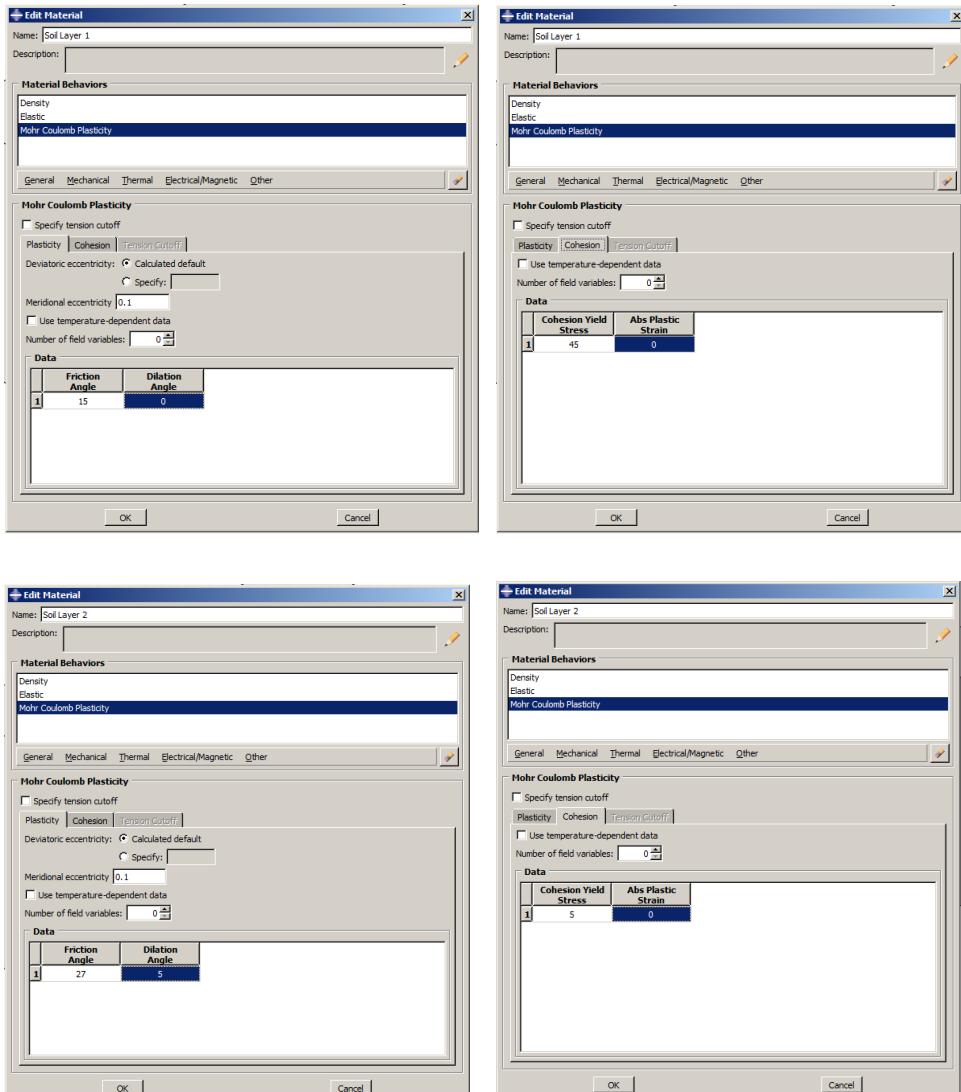


Fig. 6.4: Assigning the Mohr Coulomb plastic properties of the soil layers: top) Soil Layer 1; bottom) Soil Layer 2

Moving to creating the needed sections, all the parts will have assigned a generic section named accordingly to each material considered (“Soil Layer 1”, “Soil Layer 2” and “Concrete”, respectively), 3D homogeneous section. Therefore, go to **SR**  **Create Section** button and select **Solid**, type **Homogeneous**, and click **Continue ...**.

Assign, for each part, their appropriate sections, using **SR**  **Assign Section** button, taking into consideration that for the “Lithology” part, two sections will be assigned, according to the described lithological conditions (see the 6.3 subchapter).

In **Module: Assembly**, import all the parts to the instance by using **RS**  **Instance Part** button, check **Independent** (mesh on instance) and while holding the **Shift** button, click **LMB** on all the parts in order to bring them to the same instance. Finish the procedure by clicking the **OK** button.

The initial assembly should be as the Fig. 6.5 depicts.

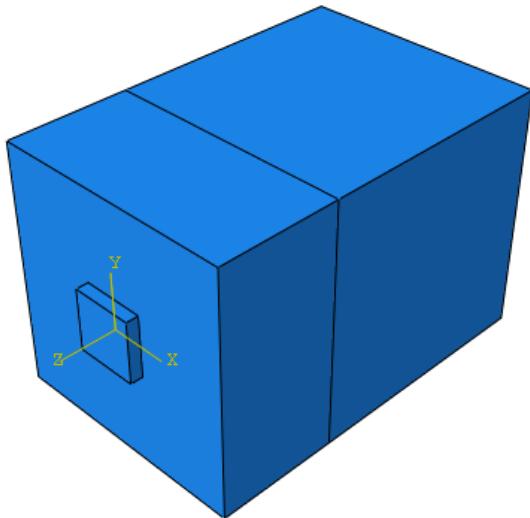


Fig. 6.5: The model's assembly

In order to check the proper material assignment, go to **TR Color Code Dialog**, and in the Color code by drop down menu, select **Materials**, and click **OK**. As the Fig. 6.6 shows, the three sections which have different materials are coloured in order to emphasize their considered material. In the Color Code dialogue window, the user is allowed to change the colour code of the materials, sections or other model characteristics, in order to emphasize the different properties. Or you can just simply click the pop down menu next to the button and select **Materials** (or **Sections**).

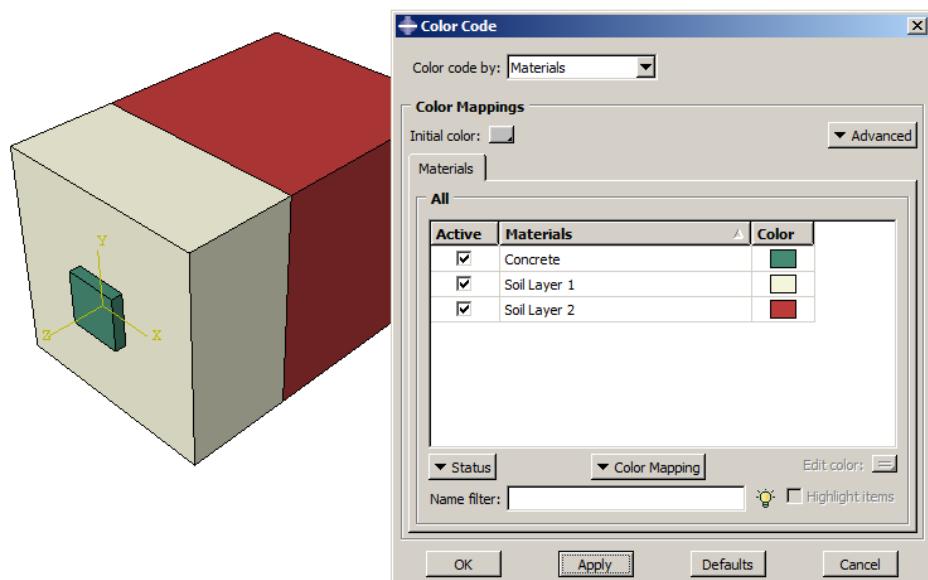


Fig. 6.6: Using the colour code to check the material/section assignment

The analysed problem is one of settlements. In the case of geotechnical engineering it is important to assess and initially induce to our lithology the settlement due to its own weight, therefore a two stage analysis will be considered or the initial stress state may be induced manually. The first method requires that a geostatic step to be taken into account, which will consider that the gravitational acceleration acts only on the soil layers, while disabling the raft. During the second, which is the proper loading of the

foundation, a simplified model in which the foundation has been constructed and immediately loaded will be considered. The second approach allows the user to consider that the initial geometry set (having the CAD assigned length, width etc.) is actually the deformed shape of the solid. In this case, no more settlements under the own weight of the lithology will appear.

Therefore, moving to **Module: Step**, create, using the **SR**  **Create Step** button, a calculation step “Foundation loading”. The default Procedure type is General and the type of calculation step is Static, General. Click the **Continue ...** button, and a new window will appear. In the description field, write “Geostatic step”. The time period will remain 1, as 1s. This implies 100% of the forces acting on the system to be applied to the structure, as the calculation is not time-dependent. Click the **OK** button, in order to complete the creation of the calculation step. Your staged analysis should be planned as presented in Fig. 6.7. This verification can be done by going to the **SR**  **Step Manager**, and in the manager dialogue window, the main characteristics (name, duration, procedure type, taking into account the non-linear geometry of the model’s deformation) are displayed.

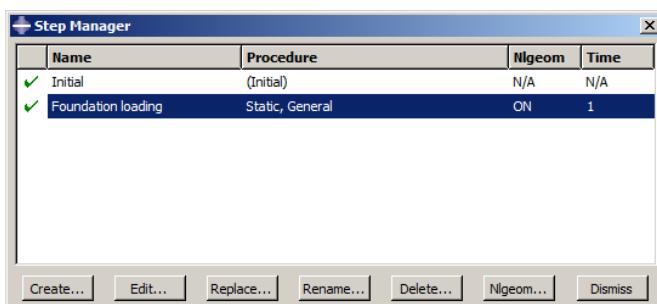


Fig. 6.7: The Step Manager dialogue window

Under the **Module: Interactions**, go to **SR**  **Create Interaction** button, and select the Surface-to surface contact type and, finally, click **Continue ...**. When prompted to select the master surface, click on the Soil Layer’s 1 top surface (in contact with the raft) and click the bottom menu

Done button. In this moment, the software demands the user to provide a method to select the slave surface, and choose Surface. In order to select the bottom surface of the raft, remove from the model space the “Lithology” part. Another method to do this is by following the path **View ▶ Assembly Display Options...**, and under the Instance tab uncheck the Lithology part. Finally, in order to return to the model, click **OK**. Turn the model by holding the **Ctrl + Alt keys + LMB** pressed and drag it in order to look at the rafts lower surface. Select this surface and click **Done** and advance to the Edit Interaction dialogue window.

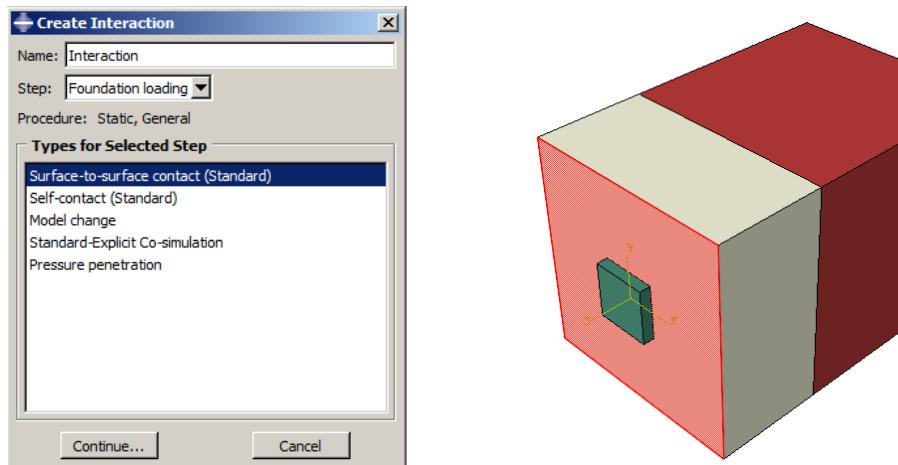


Fig. 6.8: The manual sequence of creating an interaction between the raft and the lithology

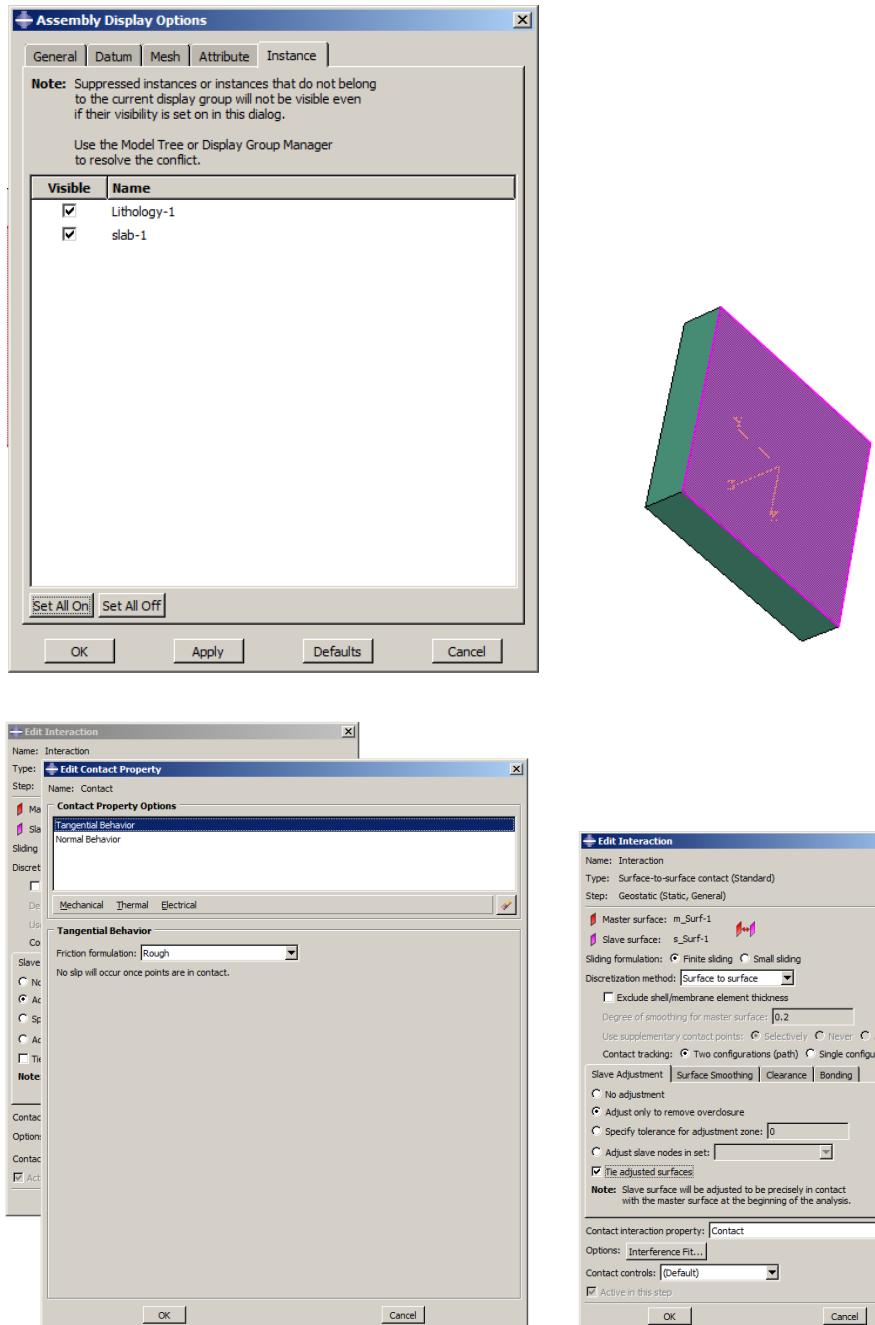


Fig. 6.8: The manual sequence of creating an interaction between the raft and the lithology (picture continued from the previous page))

Name it “Contact”, select the type as Contact and LMB click **Continue ...**. In the Edit Contact Property window, go to **Mechanical ▶ Tangential behaviour** and for the friction formulation choose Rough. This ensures that the two connected surfaces will not slip one with respect to the other. Again, go to **Mechanical ▶ Normal behaviour**, make sure “Hard” Contact is on, which disables any possibility of two parallel surface going one through the other, and uncheck (the default is checked) the Allow separation after contact property. The normal and the tangential behaviour properties indicated above simulate a perfect rigid connection between two adjacent continuous parts. In the Edit Interaction window, check the Adjust only to remove overclosure option under the Slave adjustment tab. Advancing to Contact interaction property, there can be observed that no such property is declared. Therefore click the **Create Interaction Property** button in the right side of the drop down menu and create an interaction having a normal property of “Hard” Contact with a thick on the Allow separation after contact, and as tangential behaviour - Rough.

Leaving this module and heading to **Module: ▶ Load**, impose during the first step a gravitational force that acts on the model – Gravity type, and input a Component 3 value of -9.81 (m/s^2).

Because the problem is using continuous three dimensional elements and the exterior loadings are given in the centre of the raft, concentrated in that exact one point, errors are likely to appear. Therefore it is needed to obtain the variation of the pressure in order to create the exact same effect that the axial force and the bending moment applied to the raft will create. In order to transform the vertical force “N” in pressure, simply divide it with the value of the raft surface’s area. Moving forward to the bending moment, using the Navier stress variation law, compute the absolute maximum value of the stress as ratio between the flexural moment and the rigidity of the raft. In the end, using the two values compute the variation function’s slope and the sum of the two values.

First of all, a local coordinate system is needed, which should belong to the raft. Go to SR  **Create Datum CSYS: 3 Points** button, LMB click it, and name it “raft”. For the first point, which acts as the origin of the system, select any point of the raft; for the second point, which is used in order to obtain the direction of the local 0X axis’ direction, click one parallel to the World Coordinate System’s 0X axis, but still lying on the raft’s top surface. The third point to be chosen must lie on the raft’s surface, in order to determine the XY plane.

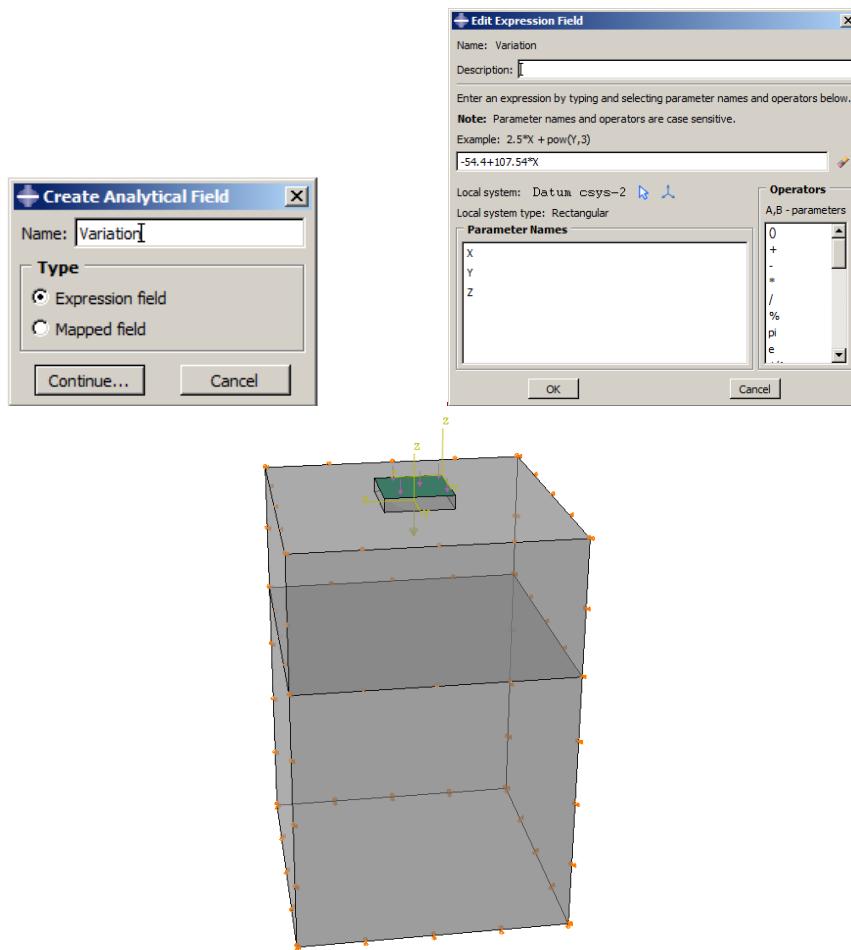


Fig. 6.9: Creating a variation pressure to which the raft is subjected

To implement the variation, go to **SR**  **Create Load**, and choose Pressure and select the top raft surface. In order to induce the pressure variation, for the Magnitude input the value of 1. In order to change the variation, click the **f(x)** Create Analytical Field button. In the Create Analytical Field window, name it “Variation” and check Expression field, and in the fill in window, input the expression of $-54.4+107.52*X$, which implies that the function will vary along the length of the raft from a minimum value of -54.4kPa to a maximum 214.4kPa. In order to finish, go to  Local system and choose the second system type that was previously created. In the end, click **OK**.

If **Loads** criteria is selected from the **TR Color Code** pop down menu, the assembly should be as depicted by Fig. 6.9, which includes also the supporting boundary conditions. Regarding the supporting conditions, pair two parallel faces of the lithology, except for the bottom, an creating simple supporting, using the Displacement/Rotation type, and checking the direction along the axis perpendicular on the aforementioned surfaces.

In order to induce the initial stress state in the soil and not using a geostatic calculation step, go to **SR**  **Create Predefined Field** button, **LMB** click it, name it “Soil Layer 1”, change the Step to “Initial” and select Geostatic stress and click **Continue ...**. Select the first layer and input the value of 0(kPa) for the top point, lying at 0(m) – vertical coordinate. Moving to the second point, input the value of -5(m) for its vertical coordinate and, regarding the stress magnitude, multiply the 5(m) with the gravitational acceleration of $9.81(\text{m/s}^2)$ and the density of the first material, which is $1.8(\text{to/m}^3)$. This will lead to a value of 88.29(kPa), having a minus sign in order to imply it acts from top towards the bottom. The lateral coefficient 1, which is equal to the lateral coefficient 2, is to be computed as $\frac{v}{1-v}$, where v is the material’s Poisson’s ratio, and which represents the K_0 geotechnical coefficient of earth’s thrust at rest. Therefore, we will obtain a 0.539 value. To finish the sequence, click the **OK** button.

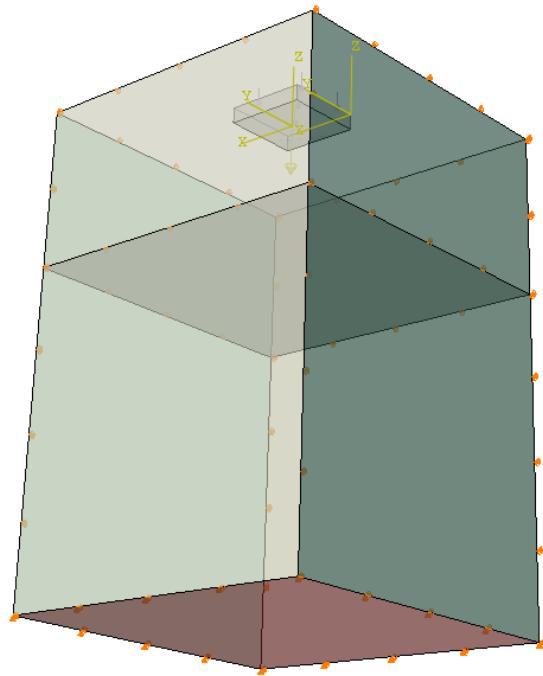


Fig. 6.10: The three different supporting conditions applied: white) on the 0Y (U2) axis; green) on the 0X (U1) axis; brick red) on the 0Z (U3) axis

Moving to the second layer, create another initial condition, introducing the values of -88.29(kPa) and -5(m) for the first point, and -284.49(kPa) and -15(m) regarding the second, lower point. The lateral coefficient in this case will be 0.37.

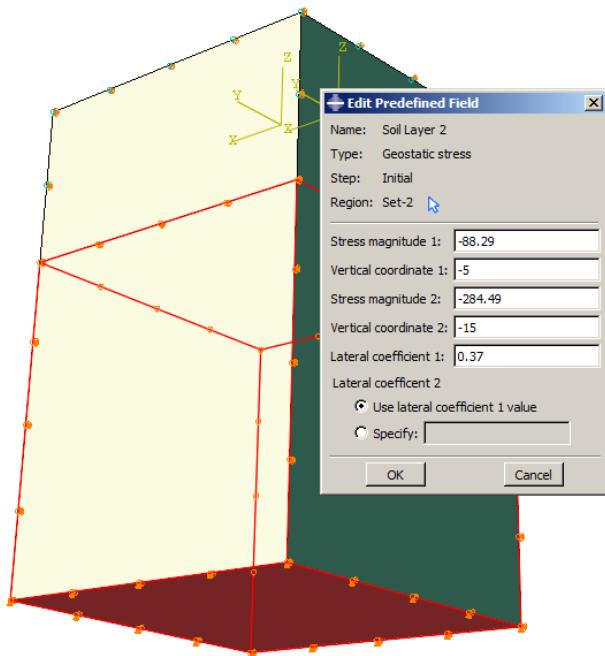


Fig. 6.11: Creating the initial stress state in the lower lithological layer

Advancing to **Module: Mesh**, seed the raft using the **SR Seed Part Instance** at a maximum element size of 25cm (0.025). Regarding the two soil layers, go to **SR Seed Edges** and click the four vertical edges of the first layer, near the superior corner of each (multiple selections may be made by holding the **Shift** key pressed). Click **Done** in the bottom, and, in the Local Seeds choose under Bias **Single**, and input the minimum size of 0.1(m) and maximum of 1(m). In order to verify the seeding, click **Apply**. This technique allows the user to consider smaller elements in the area of interest and lower their number outside of it, so that the model is still offering very good results, but with an enhanced efficiency.

In the case of a faulty direction (the arrow along the edge is not towards the interest area), the user is free to change its direction by the use of the **Flip** bias, where he will select the edges which are not thickened towards the top surface, as Fig. 6.12 depicts.

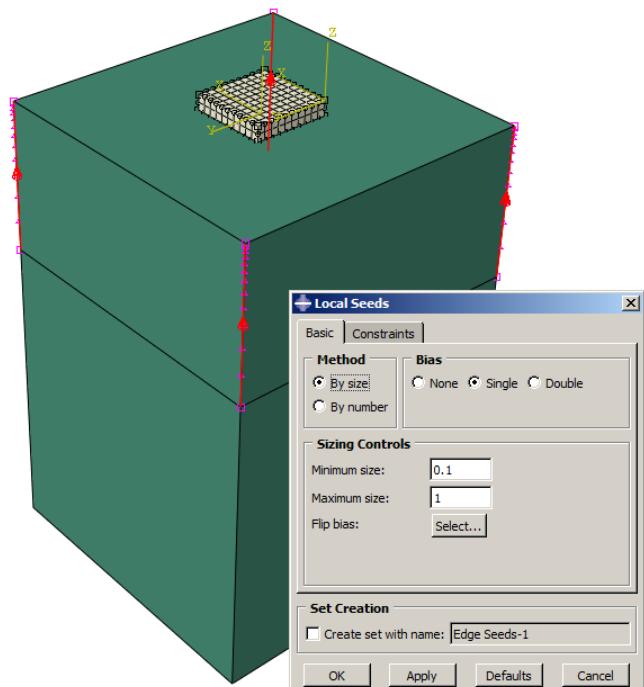


Fig. 6.12: Manually seeding the edges of a continuous part

In the case of the second layer, seed its vertical edges from a minimum size of 1 to 2.5 in the bottom. Adding to the manual seeding, a part seeding is also needed, as we wanted to control only the element size variation on the vertical axis. Therefore, assign using SR **Seed Part Instance** a maximum value of 0.5m. Finally, mesh the parts using SR **Mesh Part Instance**.

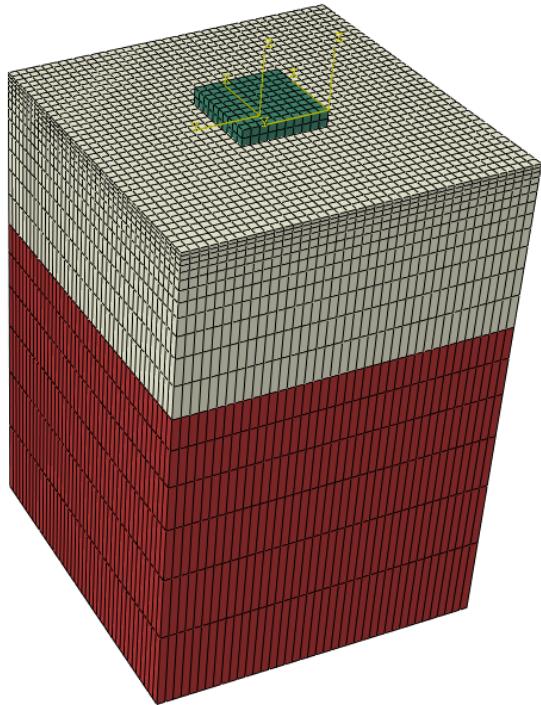


Fig. 6.13: The meshed assembly

In order to submit the model to being evaluated, go to **Module: Job**, open the **SR** **Job Manager**, create a job named “Settlement”, and if the case, under the parallelization tab increase the number of CPU cores that will participate at the calculation process. Finally, submit the job to being executed. When prompted that the job in progress is completed, click on the results, in order to advance to **Module: Visualization**.

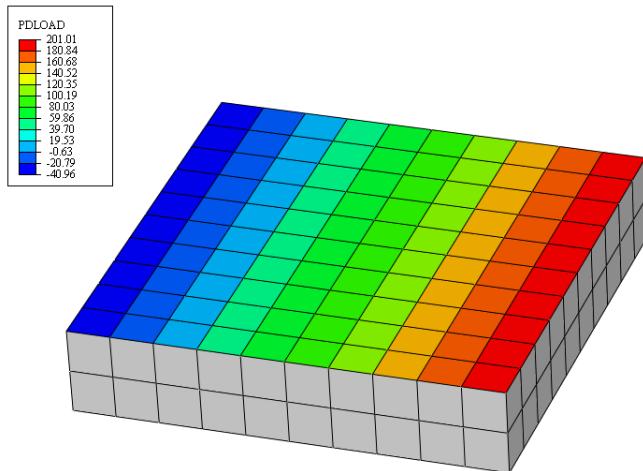


Fig. 6.14: The applied pressure variation

As displayed in Fig. 6.14, the applied pressure varies from 201.008kPa to -40.958kPa. This can be explained as the deformed form of the raft does not extend as much as the initial geometry on which the stress distribution was calculated. Sustaining this fact, the manual calculation which was computed in the beginning, was using the theory of an infinitely rigid plate subjected to bending moment and axial force.

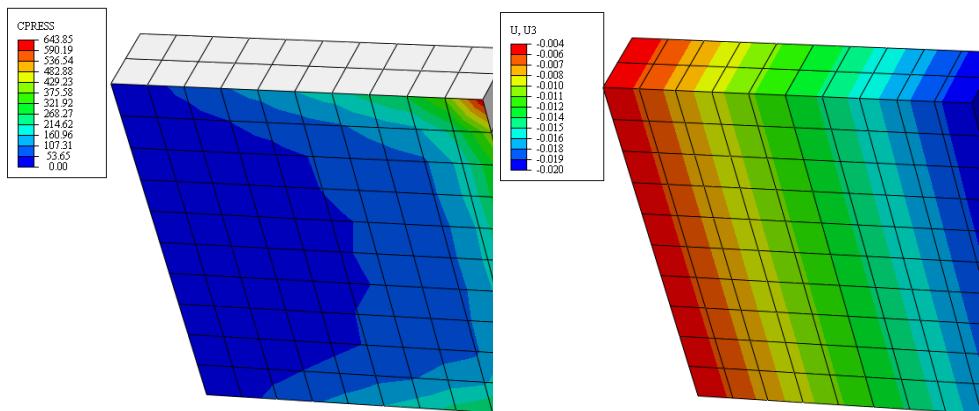


Fig. 6.15: Contact pressure variation and vertical displacement variation

In order to obtain the variation of the subgrade modulus on the raft's footing, extract the data along two paths, situated one in the middle and one on the edge of the raft's footing, both perpendicular to the bending moment's rotational axis. (see Fig. 6.16). This can be done by following the **Tools** ► **XY Data** ► **Create ...**, and in the Create XY Data dialogue window, check Path and click **Continue ...**. In XY Data from Path window, select Undeformed, True distance, and thick Include intersections. For each path export the vertical displacement and the contact pressure. Save each of them using the **Save As ...** button, after naming them "Path_X_U3" or "Path_X_CPRESS", where X is the path's number. In the end, export the data to an usable data file, go to **Report** ► **XY...** and select the four data sets created, under the XY Data tab, and in the Setup tab name the exporting file and finally click the **OK** button. Using any table calculation software, for example Microsoft Excel, process the data, by dividing the contact pressures to the displacements. It is of great importance to determine the line from which the raft is lifting, using the condition that the contact pressure will have a null value. In this case, no subgrade modulus may be calculated.

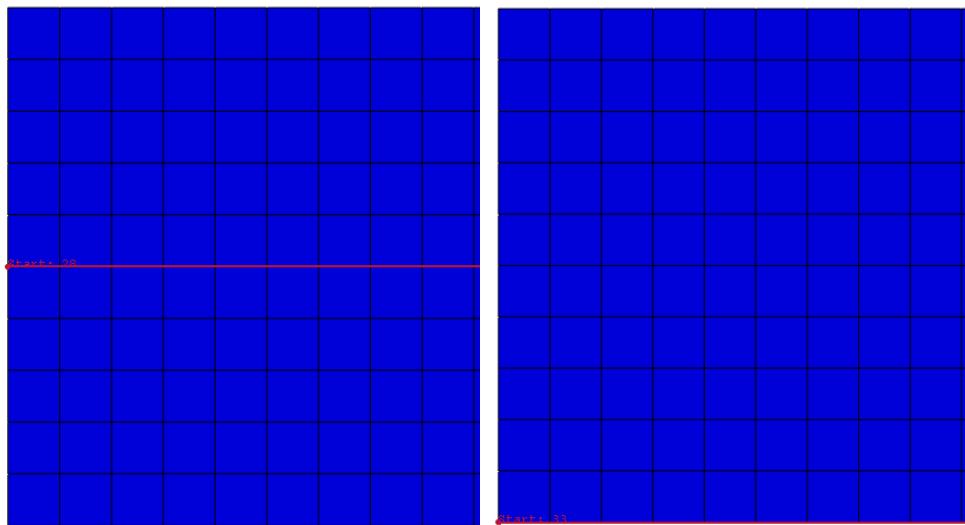


Fig. 6.16: Paths along the raft's footing

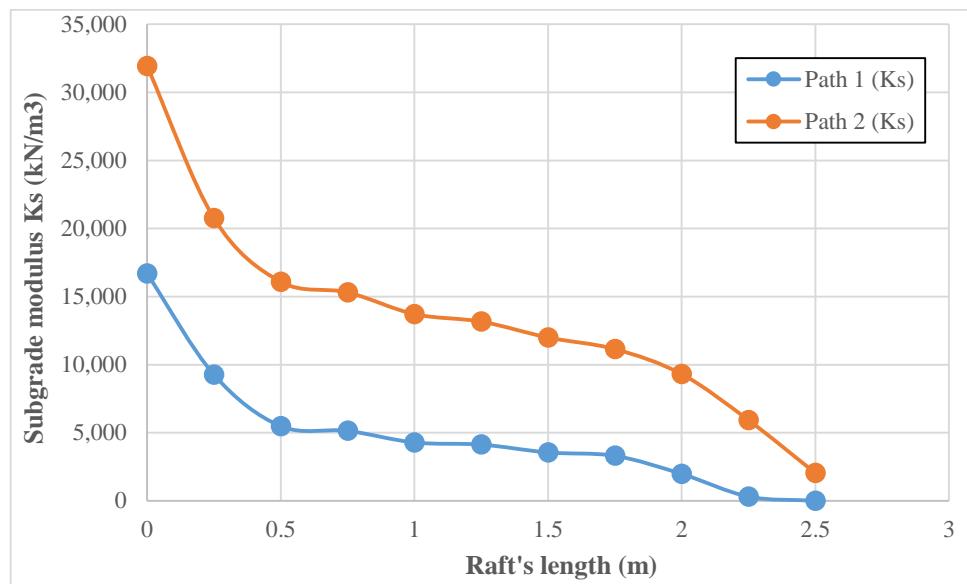


Fig. 6.17: The subgrade modulus variation along the length of the raft: blue) Centre path; orange) Edge path

As depicted in Fig. 6.17, the two functions tend to a value of 0(kN/m³), as the raft tends to lift and have no contact with the foundation soil. Moreover, the theorem of having larger stresses along the edge of a raft, rather than inside, is being also displayed in the case of this model – larger subgrade modulus values (orange line in Fig. 6.17) and larger contact pressures, accordingly to Fig. 6.15.

Adding to these aspects, another valuable one has been emphasized: the subgrade modulus is nor a lithological parameter, nor a constant value: it depends on many factors, such as: raft's geometry, loads' positions and values, soil parameters etc.

6.5 Results interpretation

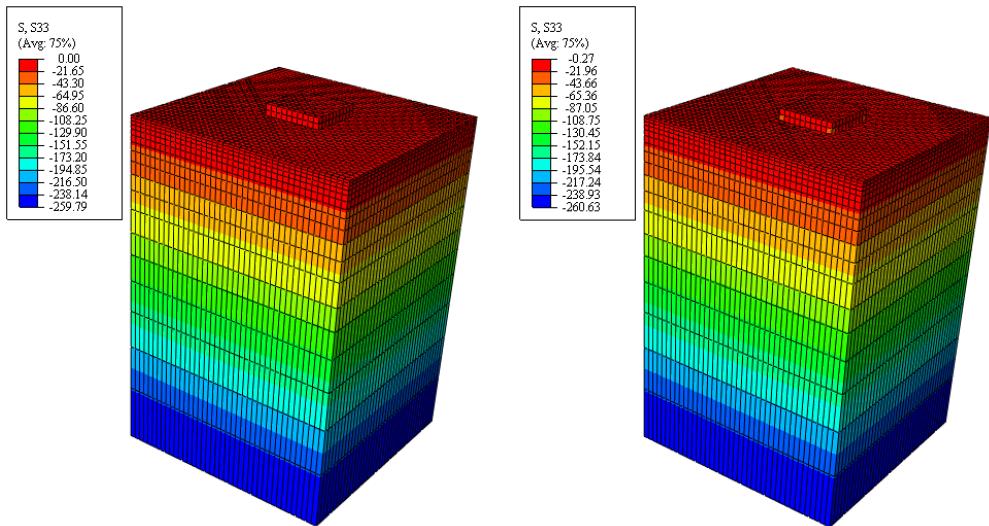


Fig. 6.18: Vertical stresses: left) initially declared; right) computed from the own weight and raft's load

Using the manually induced geostatic stresses allows the user to consider the geometry under its deformed shape, no more altering deformations appearing as part of the consolidation process. Still, the user has to take into account the fact that this technique may be applied only to a horizontally delimited lithology, otherwise swelling deformations may appear for the lower sides of the slope.

Obtaining a subgrade modulus variation under a raft or direct isolated foundation is not a straight forward process, as the stresses in the elements of the superstructure are to change following the changes in the support's rigidity. Therefore, an iterative process should be taken into consideration, down to a difference of 10% or less between two iterations.

6.6 Things to remember

The user has been accustomed to the usage of elastic-perfect plastic materials, such as the soil, creating and using surface interactions between two or more three dimensional parts, and manually inducing the geostatic stresses in order to avoid the geometry deformations in the case of considering a consolidation analysis.

This problem also displayed the process of determining, using the Finite Element Analysis, the subgrade modulus for the direct foundations, and provided guidelines regarding considering the obtained values further during a structural analysis where the foundation soil will be discretized as springs, by the usage of Winkler elastic model.

7. MECANICAL COUPLING OF SOLID 3D PARTS WITH THEIR SIMPLIFIED 1D OR 2D GEOMETRY

7.1 Keywords

3D continuum elements, 1D linear elements, 2D surface elements, Mechanical coupling

7.2 Aims

- Creating interactions and constraints between linear or surface parts and continuum three dimensional elements
- Creating interactions between continuum three dimensional elements
- Integrating the stresses of three dimensional elements in order to obtain sectional forces

7.3 Problem description

The problem to be treated consists of a concrete column linked with a beam to the nearby wall. The column will be modelled using both 1D (red line) and 3D elements, the beam only with 3D elements, while the wall consists of a shell (red surface) embedded in a 3D element.

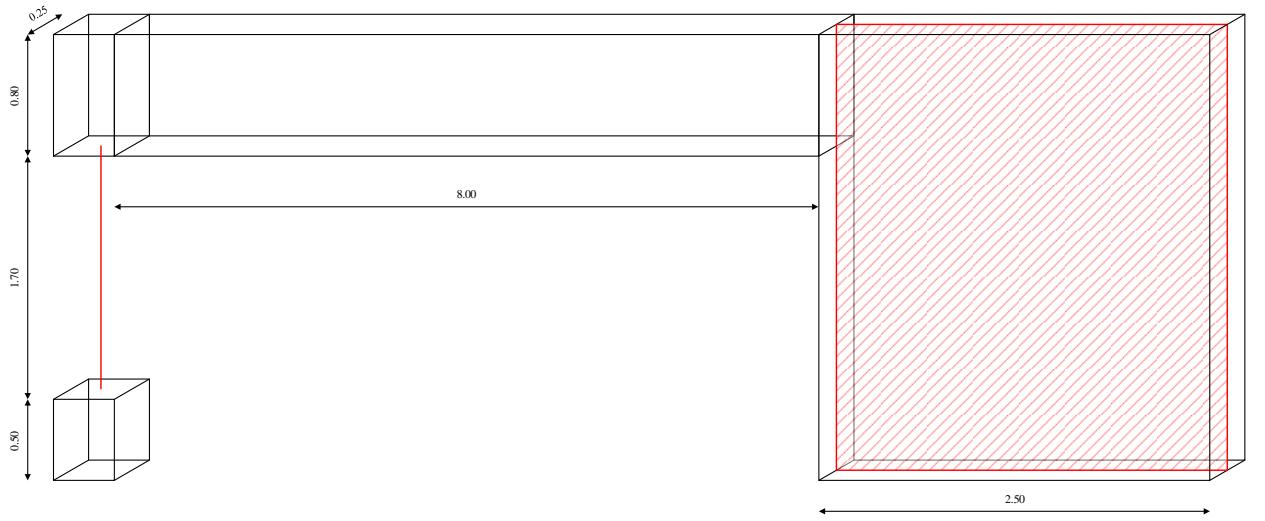


Fig. 7.1: The considered model geometry

The entire structure is subjected to a lateral pressure of 100kPa acting on the top part of the column. The own weight of the structure will not be taken into account.

7.4 Solving steps

In order to obtain the geometry, the user will draw and export the parts using a CAD software, as it follows:

- the two continuous three dimensional parts of the column, the beam and the wall, each saved separately, as ACIS SAT (*.sat) type;
- the linear segment of the column and the shell will be exported as IGES (*.igs) type.

To import each of these parts to ABAQUS, the following steps have to be followed, under the **Module: Part**. Using the already known path **File ▶ Import ▶ Part...**, select the beam and in the Create Part from ACIS File window, fill in the name “Beam”, and under the Part Attributes tab, make sure to select **③D** and Type **④Deformable**, as displayed in Fig. 7.2.

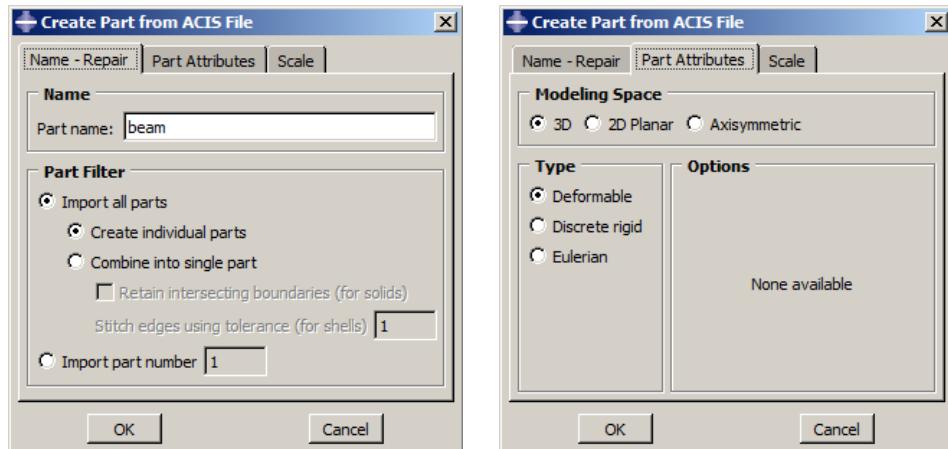


Fig. 7.2: The Create part from ACIS File in the case of the beam: left) Name-Repair tab; right) Part Attributes tab

For the wall part, the same algorithm will be used, with the only remark that the name of the part will be “Wall”, as it can be seen in Fig. 7.3.

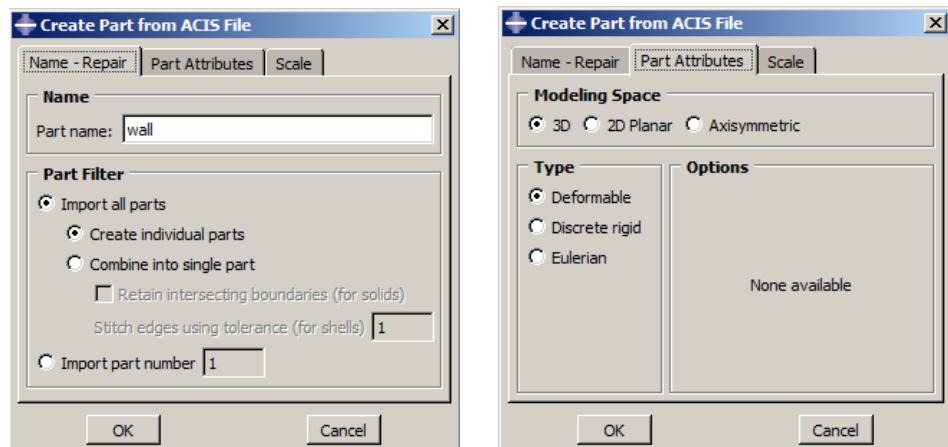


Fig. 7.3: The Create part from ACIS File in the case of the wall: left) Name-Repair tab; right) Part Attributes tab

Regarding the continuous segments of the column, the user has to cover the same command sequence, again, with the remarks that the name will be “Column” and the radio button checks shall be placed on the Combine into single part, Retain intersecting boundaries (for solids), as displayed in Fig. 7.4.

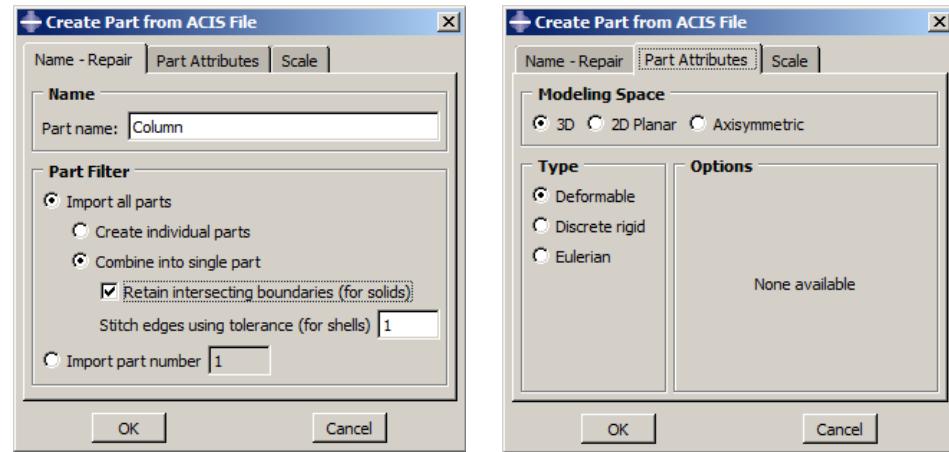


Fig. 7.4: The Create part from ACIS File in the case of the column: left) Name-Repair tab; right) Part Attributes tab

In order to import the linear segment of the column, **File** ► **Import** ► **Path...**, change the file type to IGES and select the column file. In the Create Part from IGES File window, under the Name-Repair tab, change the part name to “Linear column”, and in the Part Attributes, we leave the default checks on ②3D and Type ②Deformable, as displayed in Fig. 7.5. This is because we will integrate the linear element in a three dimensional structure, and allow the element to deform freely, according to the boundary and loading conditions.

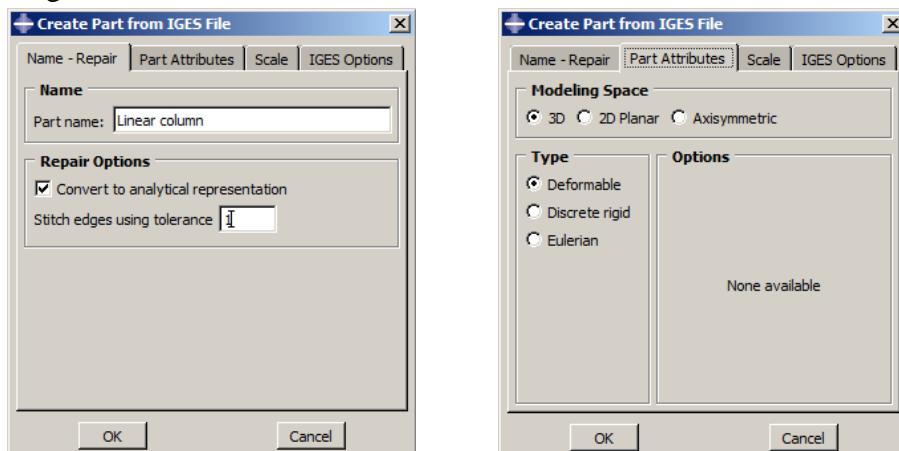


Fig. 7.5: The Create Part from IGES File in the case of the linear segment of the column: left) Name-Repair tab; right) Part Attributes tab

The last part to import in our model is the shell that will encastred in the wall part. The sequence to be followed is the one presented above for the linear column segment, with the remark that the name of the part will be “Wall shell”, as presented in Fig. 7.6.

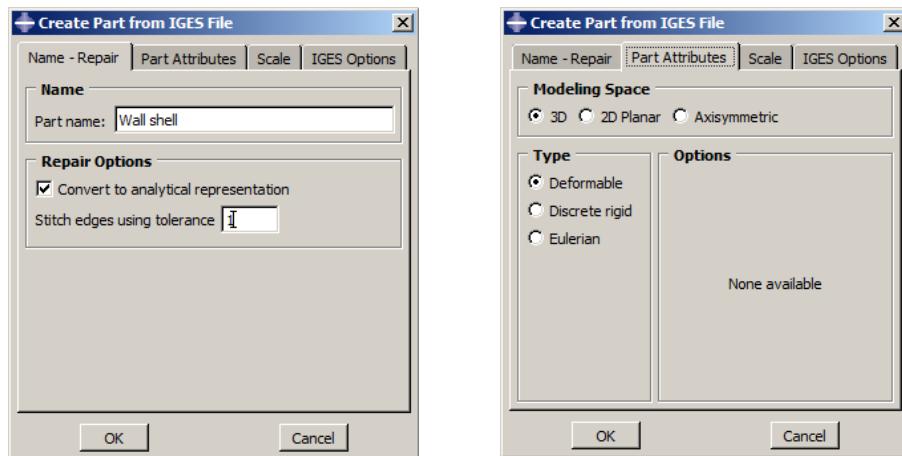


Fig. 7.6: The Create Part from IGES File in the case of the linear segment of the column:
left) Name-Repair tab; right) Part Attributes tab

Moving to the **Module: Property**, we have to create the material – concrete in our case, the sections, and assign them to the existing parts.

From the creation of the material point of view, the concrete will be considered as linear elastic type. Taking into account that there are two different situations, from the static point of view, the user is obliged to create two different materials:

- for the case of the column (both linear and continuum parts) and the beam, the concrete will have the following properties: volumetric mass of 2.5(to/m^3), Young Modulus of 2.7E7(kPa) and a Poisson’s ratio of 0.2 (Fig. 7.7);
- for the case of the wall (both shell and continuous part), because the shell is embedded in the three dimensional element and they equally share the external loads, we will consider half of the mass and of the

material's rigidity; therefore, the following values are to be considered: volumetric mass of 1.25(to/m^3), Young Modulus of 1.35E7 (kPa) and Poisson's ratio of 0.2, according to Fig. 7.8.

Therefore the first type of concrete, with full mass and rigidity will be referred to as "Concrete Full", while the second type, will be denoted as "Concrete Half". These are the material names that shall be used during this exercise.

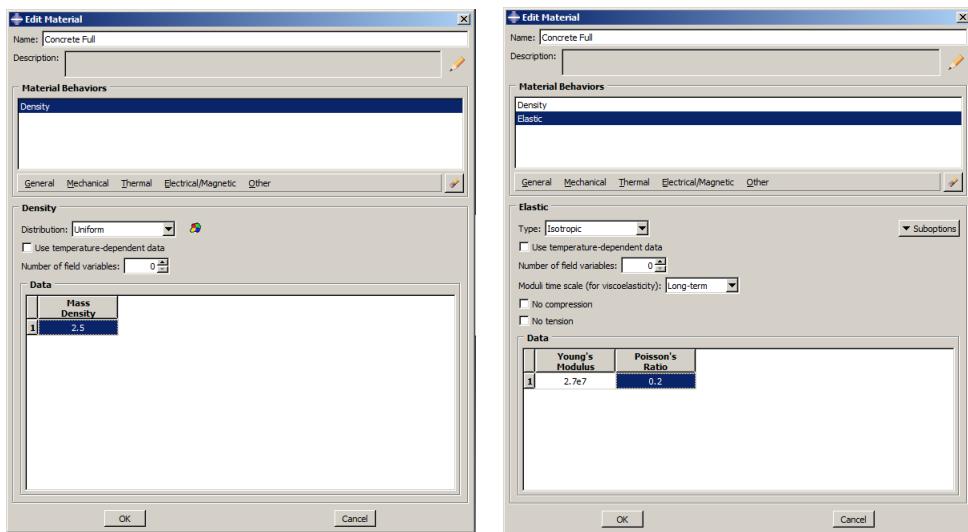


Fig. 7.7: Edit Material window while creating "Concrete Full"

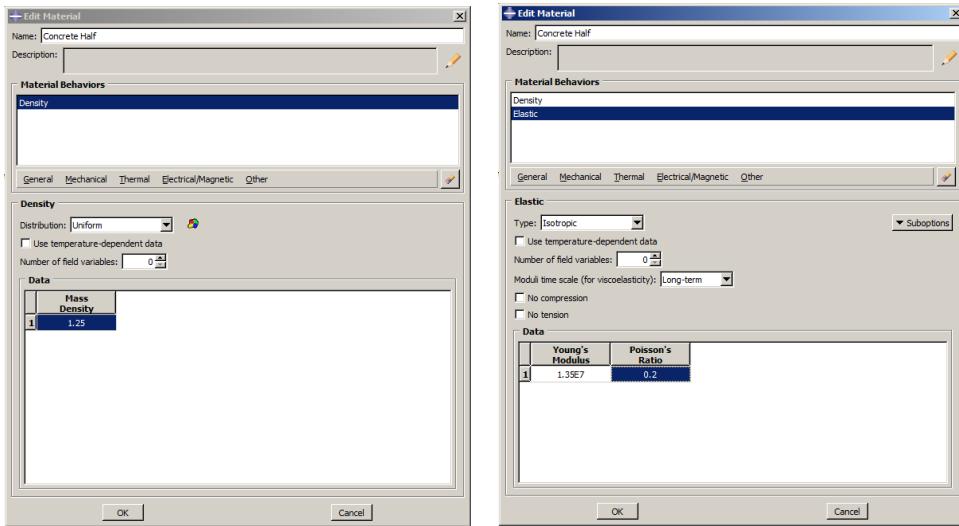


Fig. 7.8: Edit Material window while creating the "Concrete Half"

Moving to creating the needed sections, all the continuous parts except for the wall, will have assigned only a generic “Concrete Full” named, 3D homogeneous section. Therefore, go to **SR** **Create Section** button and select **ⒶSolid**, type Homogeneous, and click **Continue ...**. The material assigned to this section will be “Concrete Full”.

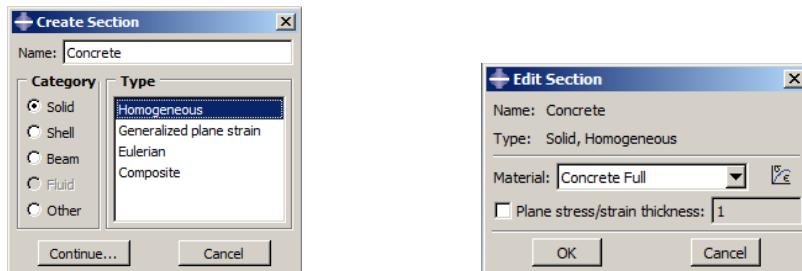


Fig. 7.9: Creating the 3D homogeneous section "Concrete Full"

The same command sequence will be applied in order to create the “Concrete Half” material, needed for the wall three dimensional element, with the remark that instead of choosing “Concrete Full” as material, the user will pick “Concrete Half”.

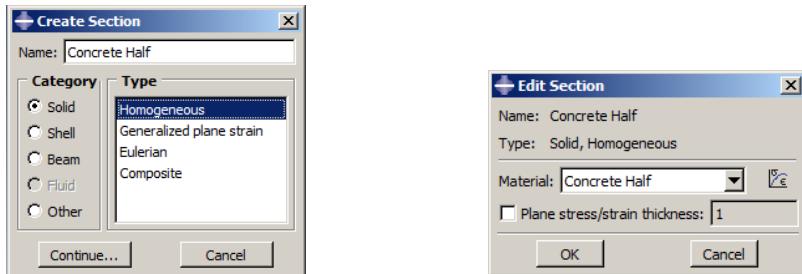


Fig. 7.10: Creating the 3D homogeneous section "Concrete Half"

Regarding the linear segment of the column, the first step in creating a section is by constructing a 0.25×0.25 m rectangular profile. Therefore, go to **SR** **Create Beam Profile** button and select under the **Create Profile** window the Rectangular type and name it "Column". Click the **Continue ...** button and in the **a:** and **b:** windows, insert the value of 0.25(m).

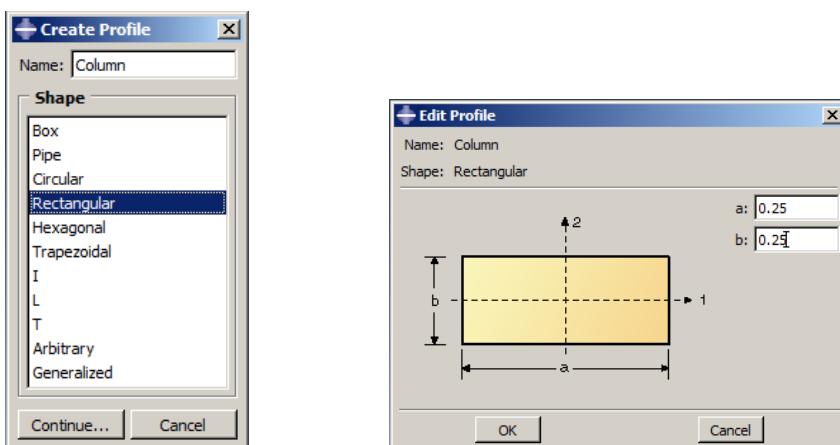


Fig. 7.11: Creating the rectangular column profile

Now, based on the newly created profile, go to the same **SR** **Create Section** button, name it "Column linear", check the **Beam**, type Beam, and click **Continue ...**. The material assigned to this section will be "Concrete Full". In the Edit Beam Section window, make sure the Profile name corresponds to the created profile ("Column"), select for the Material name

“Concrete full” and fill in the field next to Section Poisson’s ratio the value 0.2. In the end, click **OK**.

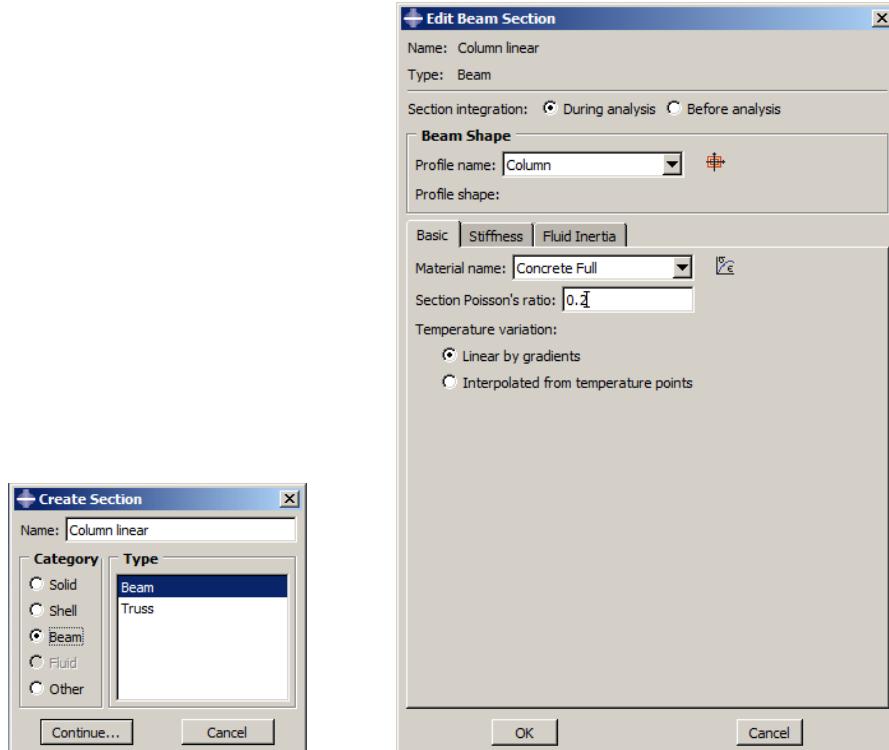


Fig. 7.12: Creating the linear column section

The last section to be created is the one needed for the shell. Following the same command – click on the SR  **Create Section**, in the Create Section dialogue window, name it “Wall Shell” and check Shell, type Homogeneous. In the Edit Section window, fill in the Value field 0.25(m), select “Concrete Half” as the material to be considered and change to 11 the Thickness integration points. Click the **OK** button in order to close the window and save your modifications.

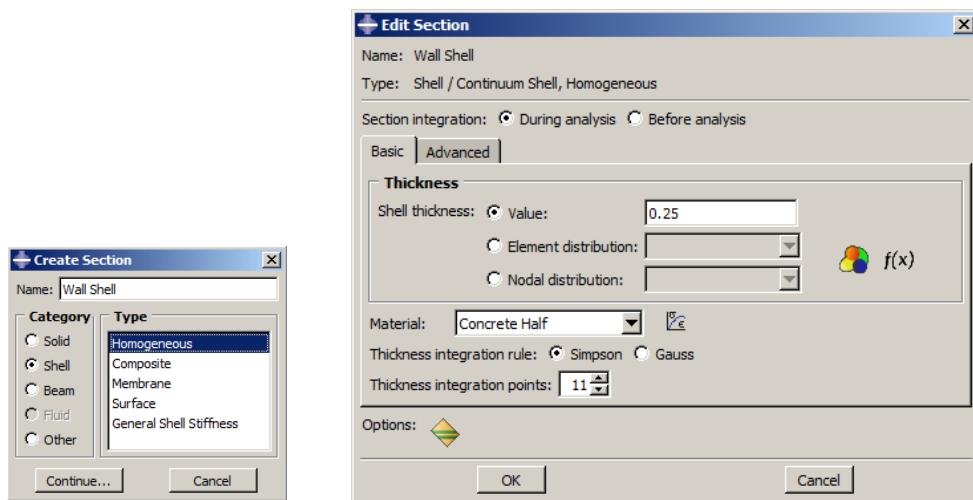


Fig. 7.13: Creating the shell section

Assign, for each part, their appropriate sections, using **SR** **Assign Section** button.

In **Module: Assembly**, import all the parts to the instance by using **RS** **Instance Part** button, check **Independent** (mesh on instance) and while holding the **Shift** button, click **LMB** on all the parts in order to bring them to the same instance. Finish the procedure by clicking the **OK** button. The assembly should look as presented in Fig. 7.14.

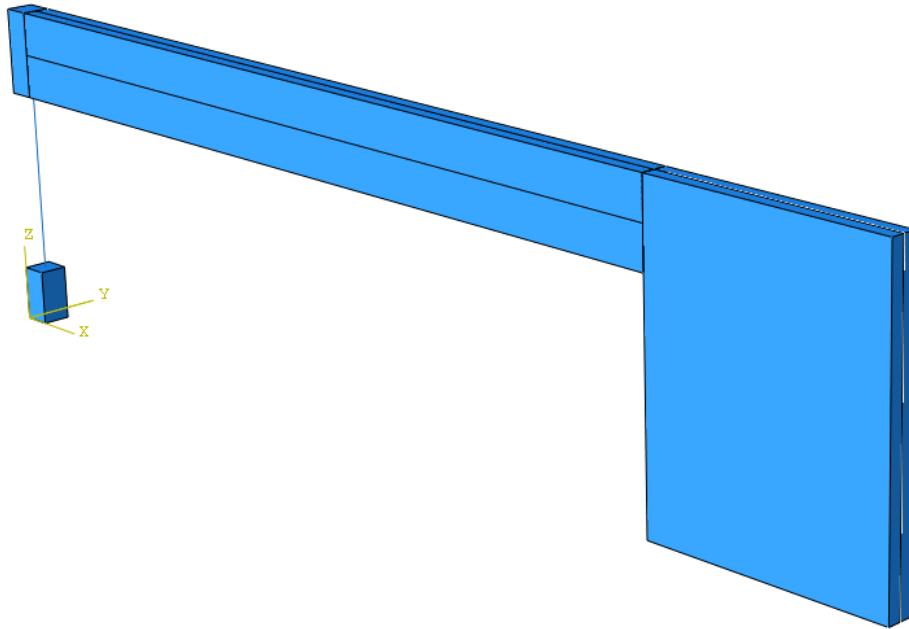


Fig. 7.14: The assembly created, presenting the position of the considered parts

Moving to **Module: Step**, create, using the SR **Create Step** button, a calculation step “Load”. The default Procedure type is General and the type of calculation step is Static, General. Click the **Continue ...** button, and a new window will appear. In the description field, write “Loading step”. The time period will remain 1, as 1s. This implies 100% of the forces acting on the system to be applied to the structure, as the calculation is not time-dependent. Click the **OK** button, in order to complete the creation of the calculation step.

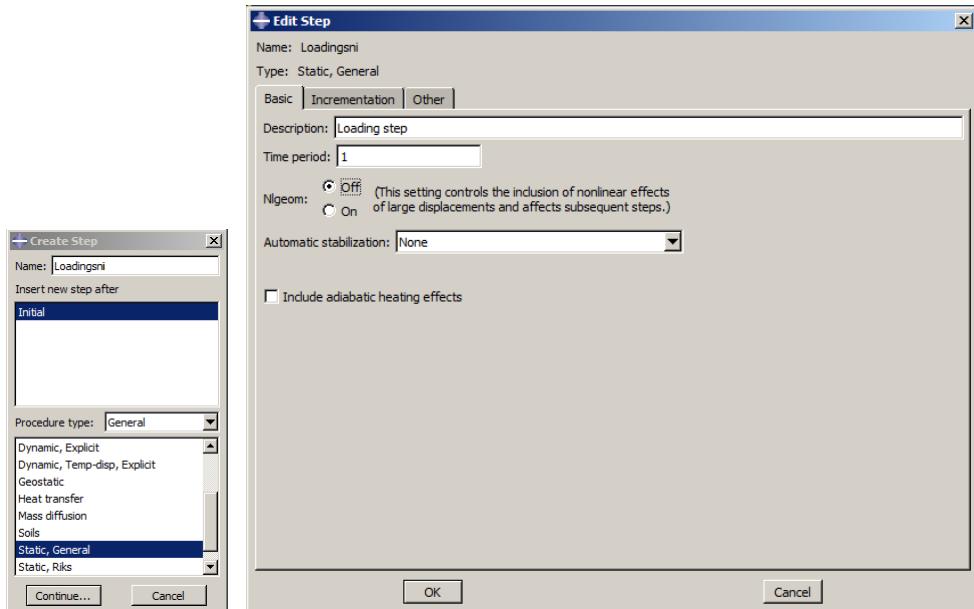


Fig. 7.15: Creating the loading calculation step

Because the structure has been modelled also with linear elements and shell elements, for the results of the calculations one expects to obtain forces (axial, shear and bending moments). They are not covered with the default calculation demands, the user will go to **Output ▶ Field Output Requests ▶ Manager...** and a window appears, containing the existing field output. Clicking the **Edit...** button brings forth the Edit Field Output Request window, where the categories of Output Variables are displayed. Full check Forces/Reactions group (not grey, as default but black check).

Under the **Module: Interactions**, go to **SR**  **Find Contact Pairs** button, and in the Find Contact Pairs window, leave all the fields with their default values and go to the bottom of it and LMB click on the **Find Contact Pairs** button. Two pairs will be found, between the column and one end of the beam, and the other end and the three dimensional continuous wall element. Up to this moment, only a classic way to create the Interaction Properties has been presented. Now, without leaving the Find Contact Pairs window, go to

Property column head and double click LMB on it. This will bring forth the Create Interaction Property window. Name it “Contact”, select the type as Contact and LMB click **Continue ...**. In the Edit Contact Property window, go to **Mechanical ▶ Tangential behaviour** and for the friction formulation choose Rough. This ensures that the two connected surfaces will not slip one with respect to the other. Again, go to **Mechanical ▶ Normal behaviour**, make sure “Hard” Contact is on, which disables any possibility of two parallel surface going one through the other, and uncheck (the default is checked) the Allow separation after contact property. The normal and the tangential behaviour properties indicated above simulate a perfect rigid connection between two adjacent continuous parts.

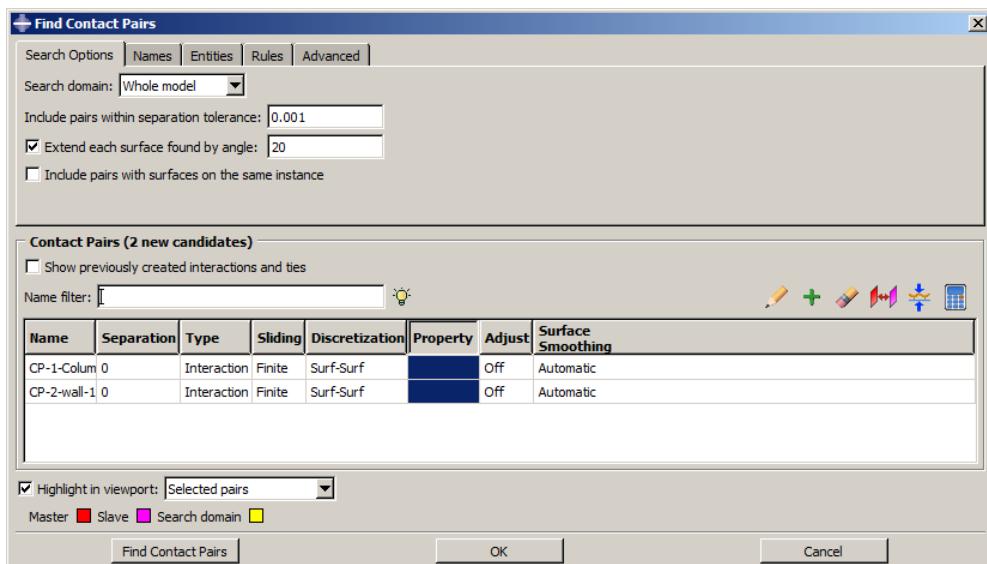


Fig. 7.16: The Find Contact Pairs window displaying the found contacts

In order to advance, click on the **OK** button and the software will return to the previous window. As it can be seen, in the highlighted Property column, the contact appears. Moving forward, double **LMB** click on the **Adjust** column head and the Slave Node/Surface Adjustment Options window will come forth. Check **Adjust** only to remove overclosure and click the **OK** button in order to end this command. Finally, click **OK** of the main window.

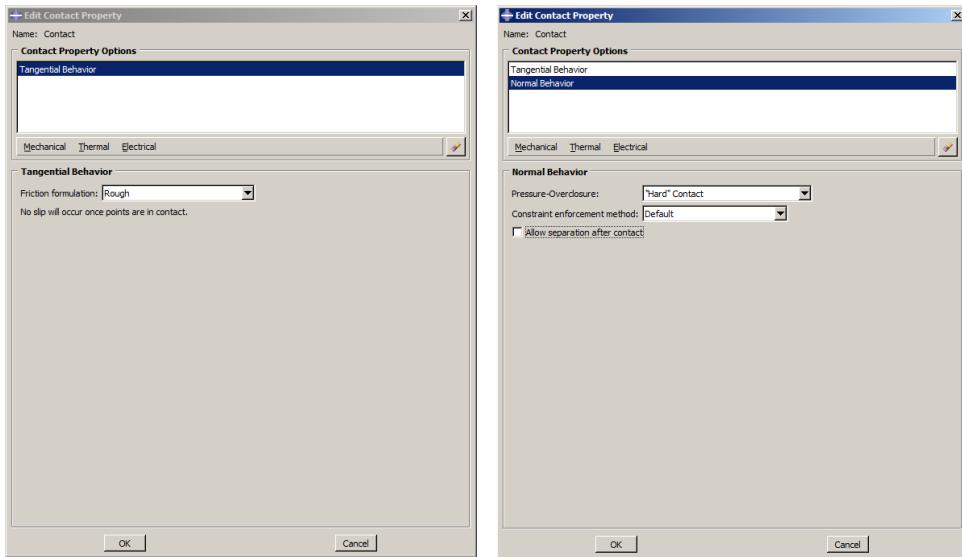


Fig. 7.17: Creating the Interaction Property

Moving to the connection it has to be done between the linear and the continuous parts of the column, go to SR **Create Constraint** button, and name it “Lower column”, select Coupling and click the **Continue ...** button. Select the lower point of the line representing the column, click **Done**, and then, when prompted to Select the constraint region, select Surface and click on the contact surface between the line and the lower continuous end of the column. Again, click **Done**, and the Edit Constraint window will appear. Check Kinematic, and make sure all the displacements (denoted with U) and the rotations (UR) are checked. Moreover, check Adjust control points to lie on surface. In the end, click **OK** to end the creation of the constraint.

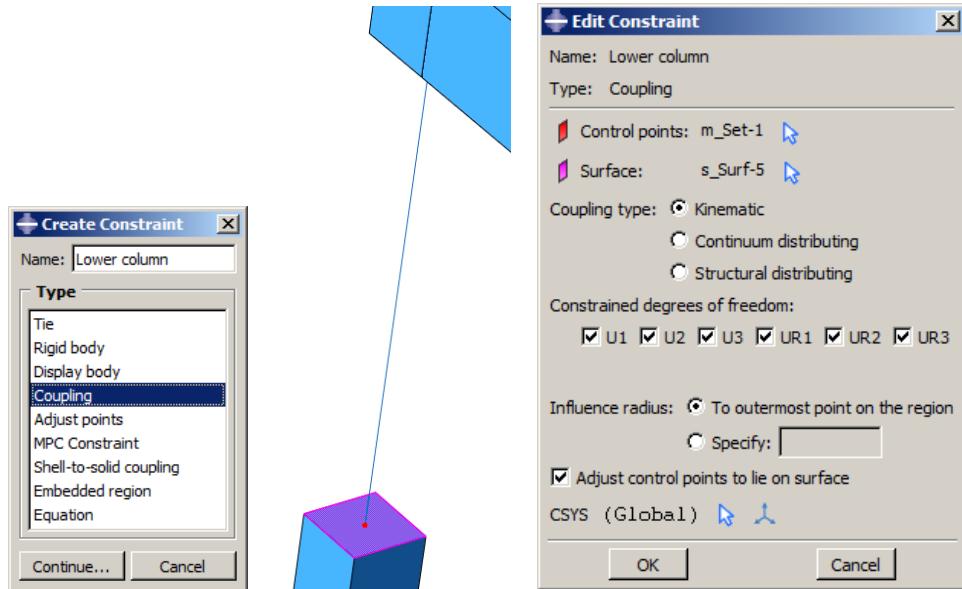


Fig. 7.18: Creating the Coupling: left) The Create Constraint window; centre) Selecting the Coupling Constraint point and surfaces; right) The Edit Constraint window

The aforementioned sequence regarding the creation of a constraint is to be repeated for the upper part of the connection between the second continuous part of the column and the wire in the middle. This one shall be named “Upper column”.

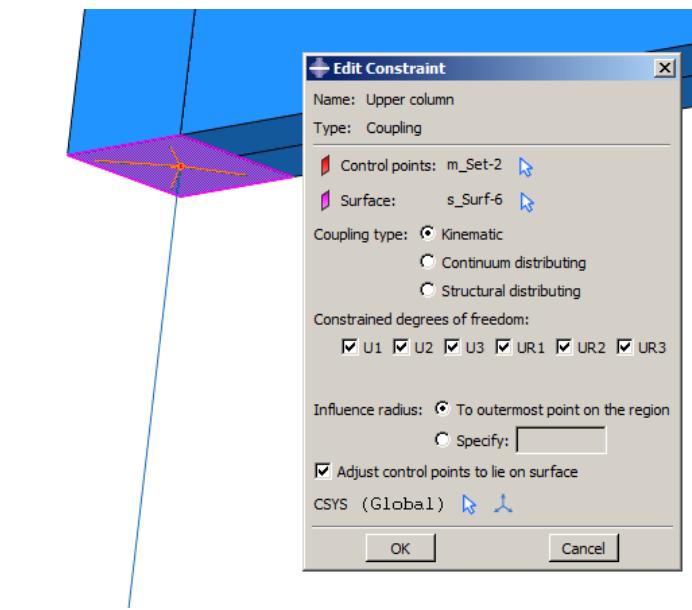


Fig. 7.19: Creating the second constraint

At this moment, only one necessary connection has to be done: between the two parts of the wall – shell and three dimensional continuous. Again, go to **SR** **Create Constraint** button, and name it “Wall”, select Embedded region and click the **Continue ...** button. The software will demand to pick the embedded region, in our case the shell. Select one of its visible edges and click **Done**. Next, after highlighting it (red colour), it is asked to provide the host region: click on Select Region, following which select the 3D wall element, and to finalize the command sequence, click **Done**. In the Edit Constraint window leave all the defaults values and checks as they already are, and simply click **OK**.

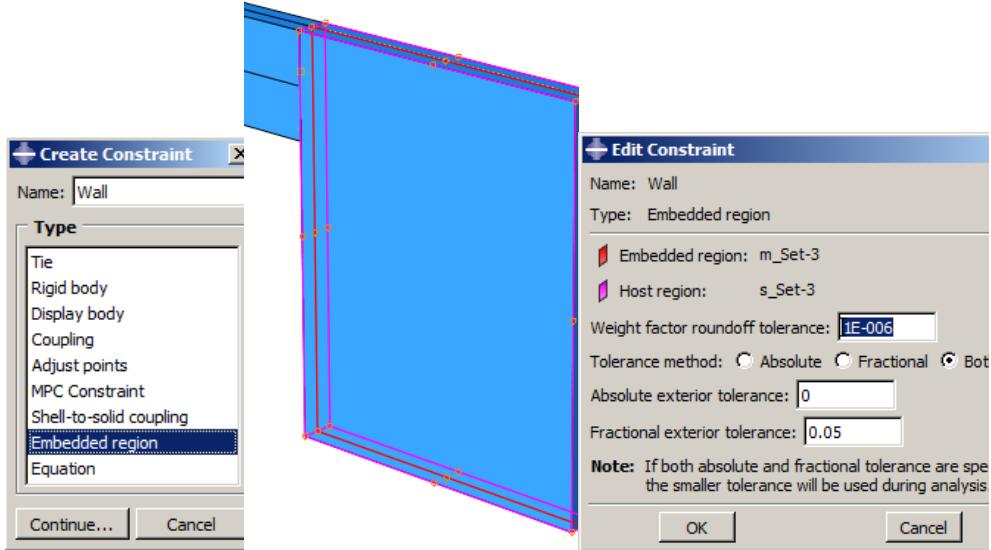


Fig. 7.20: Creating the Embedded constraint: left) The Create Constraint window; centre) The two regions selected; right) The Edit Constraint window

Advancing to **Module: Load**, encastre the lower faces of the continuous elements of the column and wall, using the SR **Create Boundary Condition** button (Fig. 7.21). Name this condition “Encastre”.

For the external loads to which the structure is subjected, apply on the upper part of the column a pressure of 100(kPa), which pushes the column towards the wall, along the 0X direction, as displayed in Fig. 7.21. This is done by accessing the SR **Create Load** button.

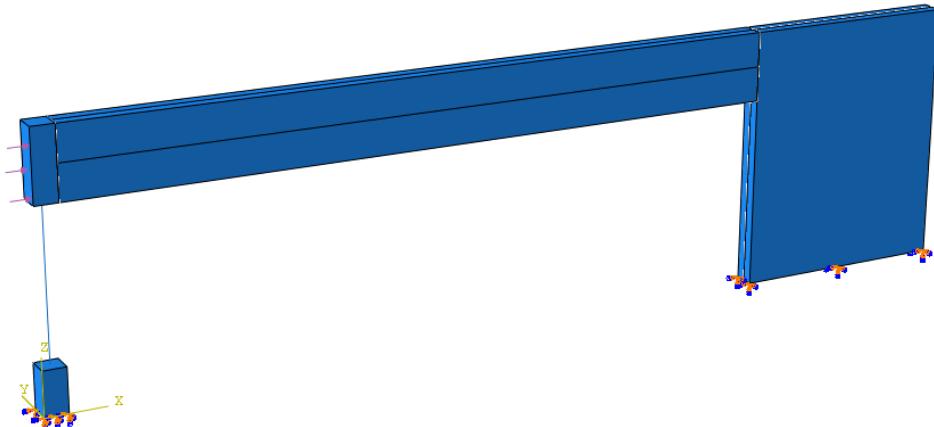


Fig. 7.21: The encasted assembly, subjected to lateral load

Moving forward, to **Module: Mesh**, seed all the parts with a maximum element size of 0.1(m), using the SR **Seed Part Instance** button, and mesh it by clicking LMB on the SR **Mesh Part Instance**.

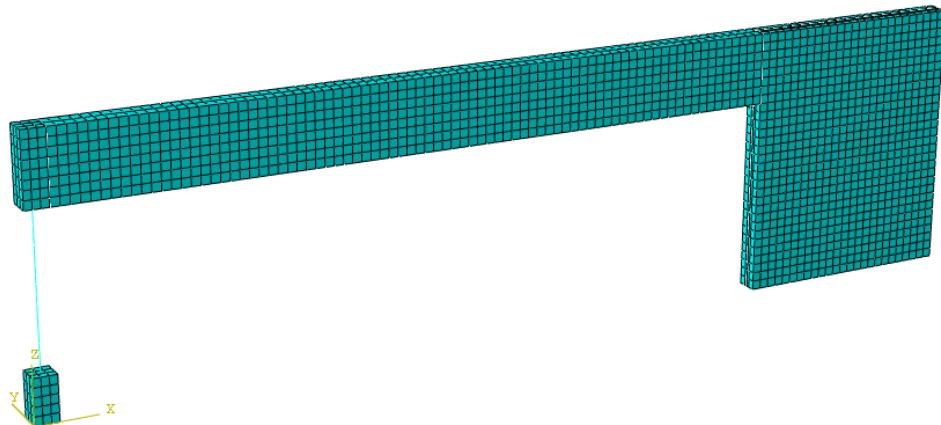


Fig. 7.22: The Meshed assembly

In order to submit the model to being evaluated, go to **Module: Job**, open the SR **Job Manager**, create a job named “Coupled_structure”, and if the

case, under the parallelization tab increase the number of CPU cores that will participate at the calculation process. Finally, submit the job to being executed.

When prompted that the job in progress is completed, click on the results, in order to advance to **Module: Vizualization**.

In order to display any of the computed output variables, go to **Result ▶ Field Output...** and select either by knowing the acronym, wither by following the variable description. Moreover, in order to accurately observe the variation the variables especially on the linear element, but also on the shell, go to **View ▶ ODB Display options...** and check Render beam profiles and Render shell thickness.

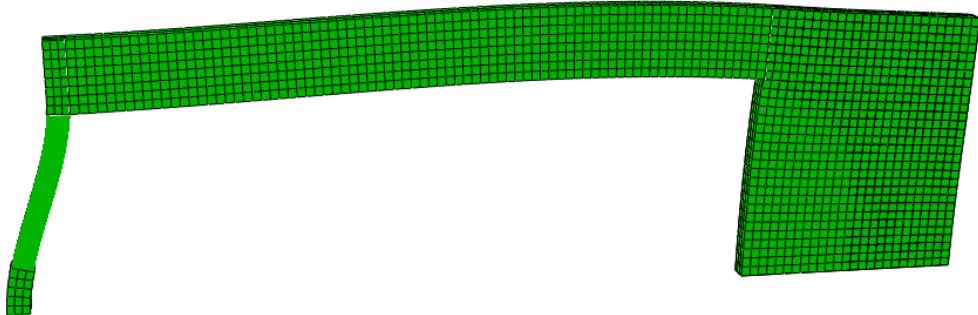


Fig. 7.23: Overview of the assembly's deformed aspect

For example, if we consider only the column, in the case of plotting displacements along the 0X axis, the answer obtained is ranging from 0 to 0.0539mm.

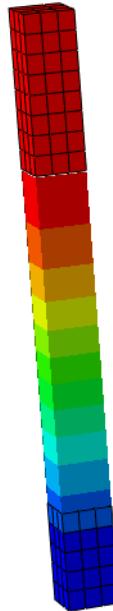


Fig. 7.24: Displacements along the 0X axis on the column

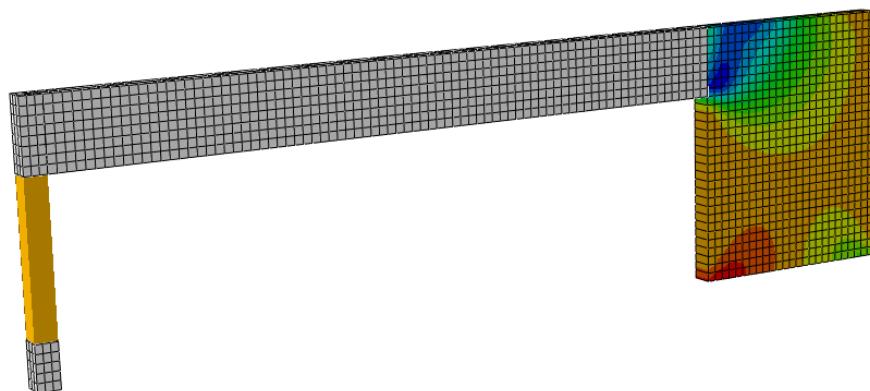


Fig. 7.25: Sectional Forces along 0X axis being calculated only for the 1D or 2D elements

In order to integrate automatically the efforts of a continuous' element, during the post-processing phase, the user must go to the **SR**  **Create Free Body Cut** and LMB click. Here, in the Create Free Body Cut window, name the cut

“Beam” and check 3D element faces, click **Continue ...** and in the Free Body Cross-Section window, under Item type select Surface, Method – Surface sets and while having checked Highlight items in viewport, look for the set obtained during the creation of the contact between the column and the beam (Fig. 7.26). Finally, click **OK**. In the Edit Free Body Cut, make sure that for the summation Point is checked Centroid of the cut, and for the Component Resolution Normal and tangential. Click **OK**. In order to choose from what is displayed (forces, moments or both) and under which form (resultant or components on the direction of the World Coordinate System), go to **SR**  **Free Body Cut Manager** and at options, under the different tabs, for the cut created.

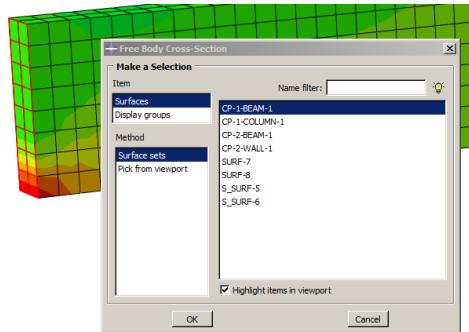


Fig. 7.26: Searching the lateral face existent set

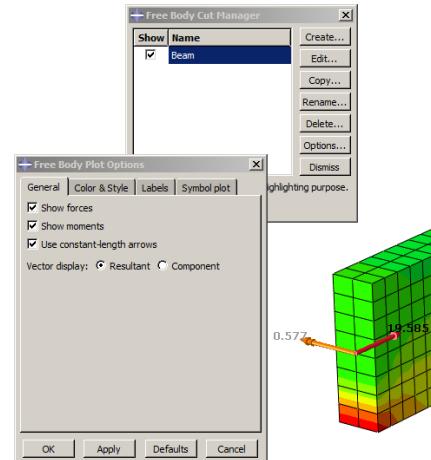


Fig. 7.27: Plotting the resultant moment (orange) and force (red) of the section

It is strongly encouraged that previously plotting the forces and moments, the user changes the colours under which the efforts are displayed (for example orange for the moments and red for the forces, see Fig. 7.27).

7.5 Results interpretation

Taking a closer look at Fig. 7.23, as displayed in Fig. 7.28, the well-known of double encasted column subjected to lateral loads at one side has been obtained, verifying the validity of the coupling constraint created – even if the vertical element is comprised of three individual parts (two continuous and one being one-dimensional), due to the interaction, the assembly behaves as one continuous element.

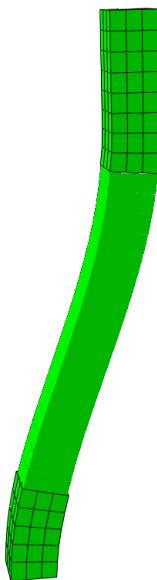


Fig. 7.28: Deformed aspect of the column

Moreover, if we consider the coupling between the shell and the wall, we observe that they present the same deformations, but we must take into account that their rigidities are altered to the half. Therefore, the stresses that each of the two parts support are reduced to half of the real value, which is the sum of the two (Fig. 7.29).

This two techniques may represent simple tools in order to obtain fast integration on the area of the element.

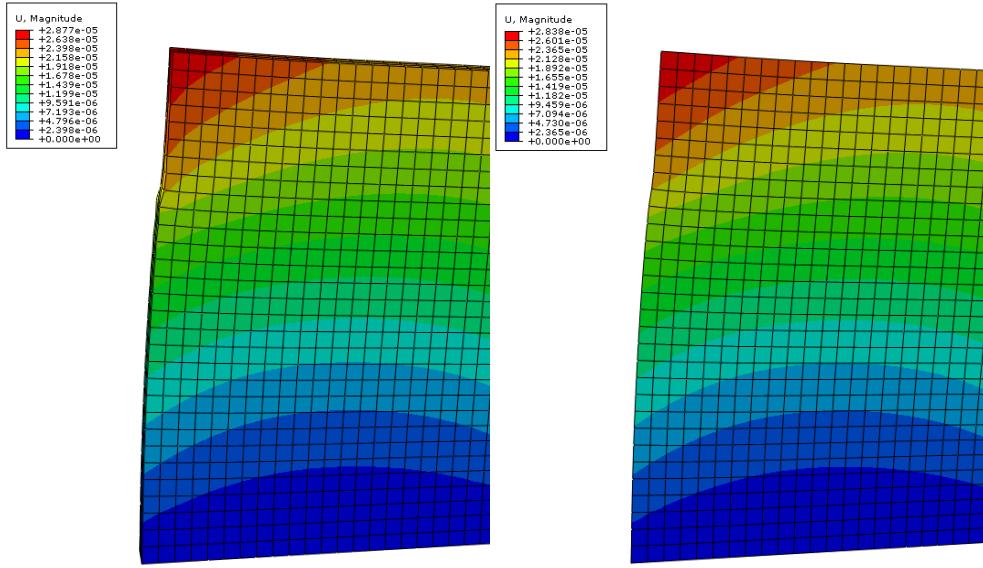


Fig. 7.29: Comparison between the displacement obtained on the coupled wall: left) 3D element; right) shell

7.6 Things to remember

This application introduced notions and techniques used to model either the interactions between non-interest areas a structure, simply modelled as linear and surface elements, while the other are continuous three dimensional, either fast integrate stresses in order to convert them to sectional forces.

8. 2D ANALYSIS OF AN ELASTIC-PERFECT PLASTIC RECTANGULAR SAMPLE SUBJECTED TO TRIAXIAL COMPRESSION

8.1 Keywords

Axisymmetric problem, elastic-perfect plastic material, dynamic explicit, history output

8.2 Aims

- Creating a axisymmetric problem
- Creating a dynamic explicit calculation step
- Determining the needed mesh quality as function of the obtained results

8.3 Problem description

A cylindrical soil sample, of 50 by 100mm (diameter and height) is subjected to triaxial compression by the means of imposed top platen advancing with a constant speed of 2mm/s (see Fig. 8.1). The top and bottom faces and the lateral edge of the sample are horizontally supported, adding to which the lower part is also vertically supported.

It is of great interest to observe the different results the different meshing techniques provide in order to understand their impact on the obtained data.

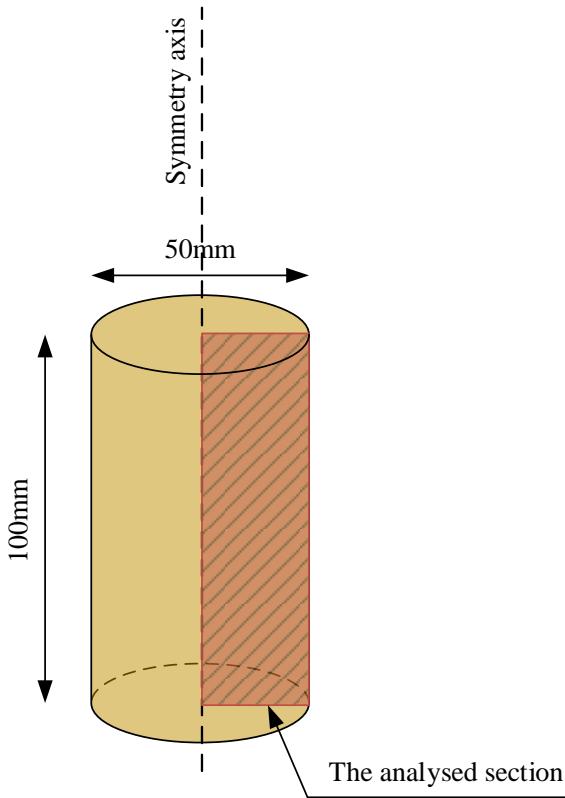


Fig. 8.1: The model's geometry

8.4 Solving steps

The shell is to be drawn, at user's preference, either under ABAQUS's sketching module, either using any CAD software and exported as ACIS (*.sat) file. Either way, the part shall be created as Fig. 8.2 displays, by checking Axisymmetric, Deformable and Shell. All these aspects and properties will be provided in the Create Part dialogue window, as displayed in Fig. 8.2.

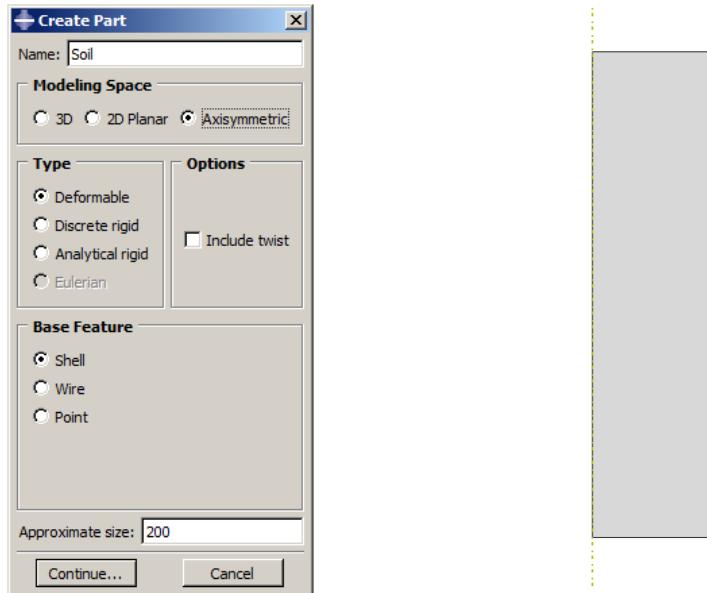


Fig. 8.2: Importing the modelled geometry: left) Create Part window; right) the geometry of the part

Following this command sequence the needed part has been created, also imposing the axisymmetric problem type. Therefore, advance to **Module: Property**, and, taking into consideration a stiff clayey silt, create a new material using **SR** **Create Material**, which will be named “Soil”. Here, in the Create Material dialogue window, input the Density of 1.7(to/m³), under **General** properties, Young’s Modulus of 12500(kPa) and a Poisson’s Ratio value of 0.35, under **Mechanical ▶ Elasticity ▶ Elastic** and, finally, in order to assign a perfect plastic behaviour, go to **Mechanical ▶ Plasticity ▶ Mohr Coulomb Plasticity**, and input a 16(°) value for the Friction Angle, 0.1(°) Dilation Angle, 40(kPa) Cohesion Yield Stress and 0(-) Absolute Plastic Strain (see Fig. 8.3).

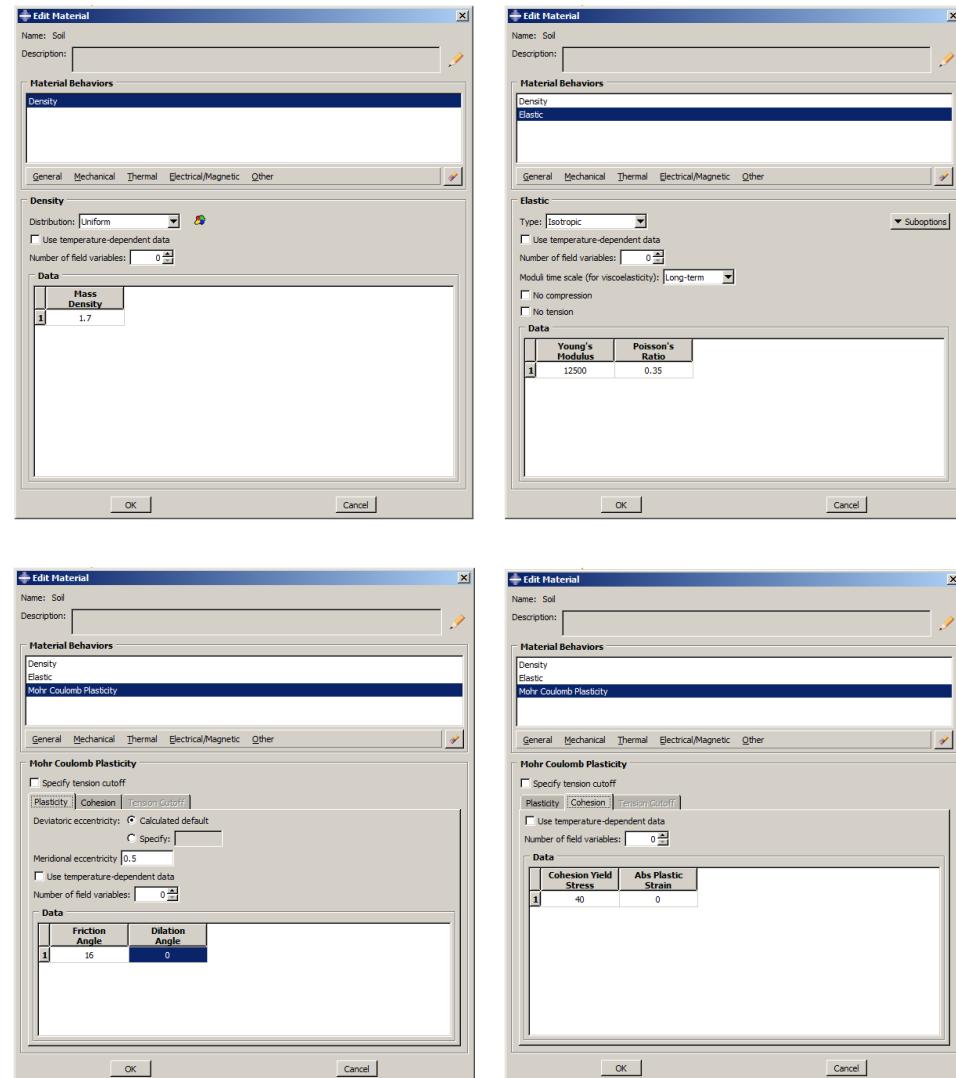


Fig. 8.3: Creating the material: left-top) density property; right-top) elastic property; bottom) Mohr Coulomb plasticity properties

Following the creation of the material, create a “Soil” section, having **⊖Solid**, **Homogeneous** type. End the sequence by **LMB** clicking the **Continue ...** button and make sure the previously created material is selected.

Assign the section to the part by **LMB** clicking the **SR**  **Assign Section** button.

Moving to the **Module: Assembly**, in order to insert the two components into the whole system, **LMB** click the **SR**  **Instance Part** button and check Independent (mesh on instance). Finish the procedure by clicking the **OK** button.

Advancing to **Module: Step**, click on the **SR**  **Create Step** button, and name the calculation step “Load”. The desired Procedure type is General and the type of calculation step is Dynamic, Explicit. Click the **Continue ...** button, and a new window will appear. In the description field, write “Load”. A value of 10s is required so that the advancing top platen will reach at the end a value of axial strain of 20%. At this value, the shearing process of a soil sample is considered successfully completed, either by overpassing the peak value of the deviator stress, either by touching a plateau of it. Set the nonlinear geometry calculation On. Click the **OK** button, in order to complete the creation of the calculation step.

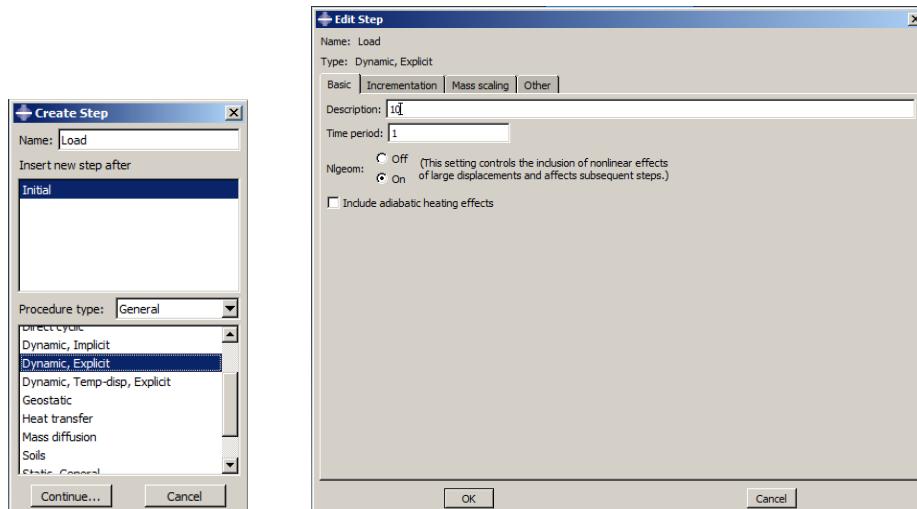


Fig. 8.4: Creating a dynamic step

As there is only one part, there is no need to go to **Module: Interaction**, therefore jump directly to **Module: Load** and click the **SR**  **Create Boundary Condition** button. Here, name it “Lateral support”, make sure the **Mechanical Category** is selected and choose Displacement/ Rotation. Finally, click **Continue ...** in order to advance. Choose the top, bottom and vertical edge overlapping the symmetry axis, holding the **Shift** key and clicking the **LMB** on them. Advance to the Edit Boundary Condition dialogue window, where the displacement on the OX direction has to be checked U1 and a 0(m) value filled in. Click the **OK** button to leave the dialogue window.

Adding to this, a vertical support is needed, to which the lower edge will be subjected. Using the same command sequence as before, select for the lower edge of the sample U2.

In order to impose the platen movement, also under Boundary Conditions, select the top edge and, under Velocity/Angular velocity, check V2 and fill in the -0.002(m/s) value.

Adding to this, a confining pressure of 200(kPa) has to be applied. Taking into account the real test conditions, apply it to the upper edge and the one opposed to the symmetry axis. This can be done either by using the **SR**  **Create Load** button, either by following the **Load ► Create...** path. The assembly containing the boundary conditions and external loads should look as Fig. 8.4 displays.

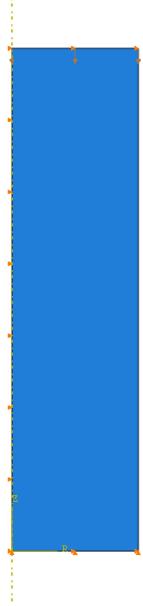


Fig. 8.5: The boundary conditions and external loads applied to the model

Advancing to **Module: Mesh**, select a global seeding of 5mm (0.005), by using the **SR  Seed Part Instance** button. In order to emphasize the efficiency of different meshing techniques, a quad and a tri-based mesh, and, for further explaining, a finer quad mesh. The tutorial will present the possibility of choosing the type of mesh element to be used, and finally the results obtained, during the 8.5 subchapter.

In order to impose the software the type of meshing element to consider, go either to the **SR  Assign Mesh Controls**, either along the **Mesh ► Controls...** path. In the Mesh Controls dialogue window, check, at your will one of the three available Element Shapes, and select also a Technique and, if desirable, an Algorithm Option.

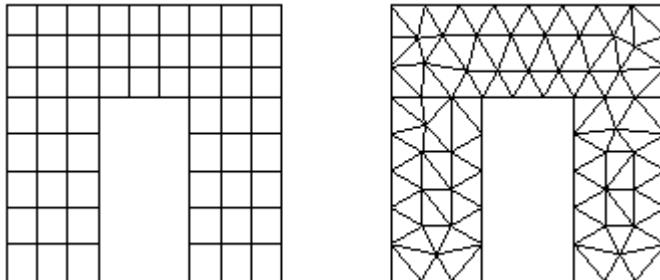


Fig. 8.6: Example of quad and tri-based meshing of the same geometry as presented in ABAQUS User's manual

As Fig. 8.6 presents, different meshing techniques provide a different number of elements and, in the case of special geometry of the assembly's parts, may allow or not the possibility of discretization.

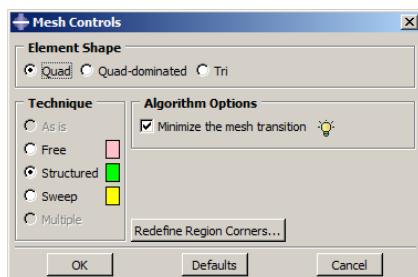


Fig. 8.7: The Mesh Controls dialogue window

In the end, by using the **SR** **Mesh Part Instance**, end the discretization phase. At this moment, the model is ready to be submitted to calculation. Therefore, we advance to the **Module: Job**. Go to **SR** **Create Job**. A Create Job window will appear in which the name shall be changed from the default “Job-1” to “Triaxial compression”. According to your system configuration, you may choose to accelerate the computation phase by selecting a parallel CPU calculation (based on the cores number) and a GPU acceleration.

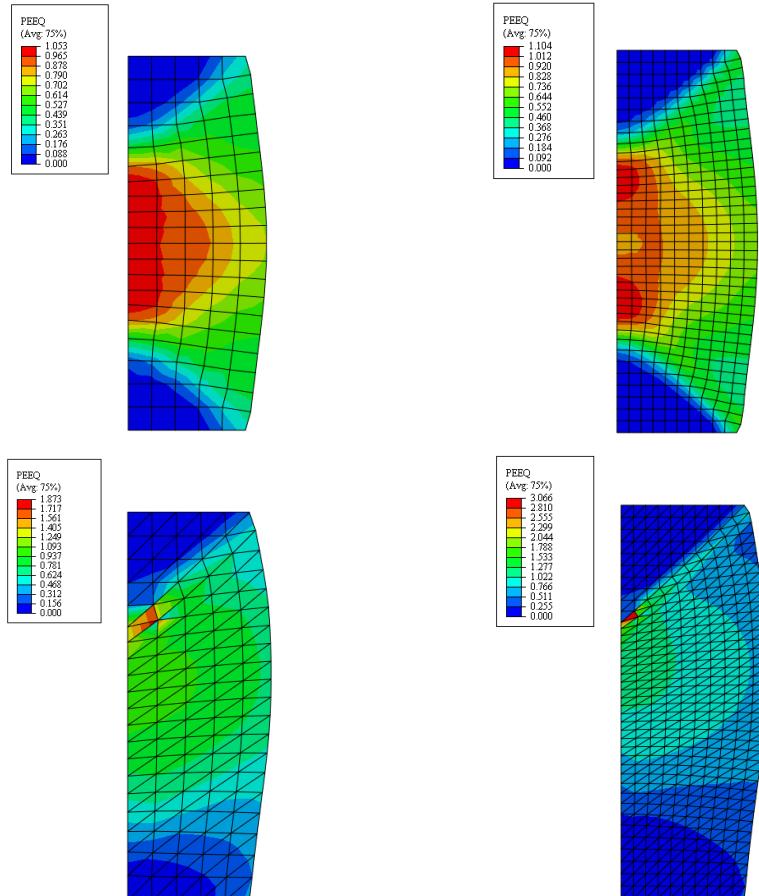


Fig. 8.8: The equivalent plastic strain variation (PEEQ): top-left) quad mesh&seed of 5mm; top-right) quad mesh&seed of 2.5mm; bottom-left) tri mesh&seed of 5mm; bottom-right) tri mesh&seed of 2.5mm

Every calculated increment offers information regarding the model's stress state and geometrical deformations. In order to emphasize the changes, with respect to the original geometry, **LMB** click the **SR** **Allow Multiple Plot States** button, which gives the user the possibility of choosing different states of the model to be plotted one against the other.

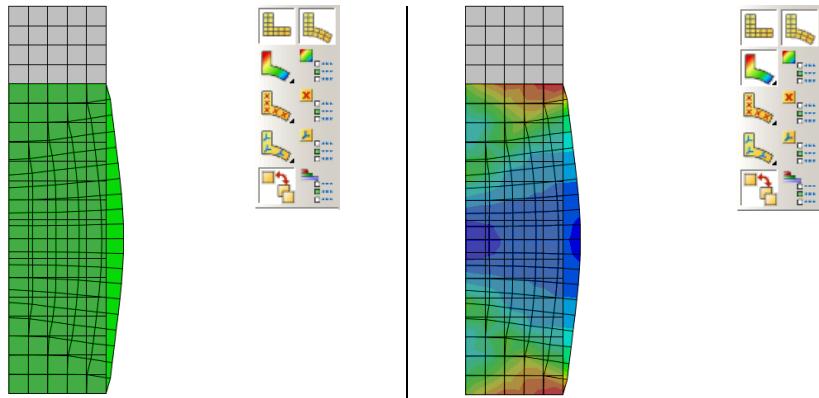


Fig. 8.9: Plotting the geometrical variation with respect to the original situation: left) the deformed and undeformed shapes and the SR's selected buttons; right) the pressure state of the deformed shape overlapped on the initial geometry

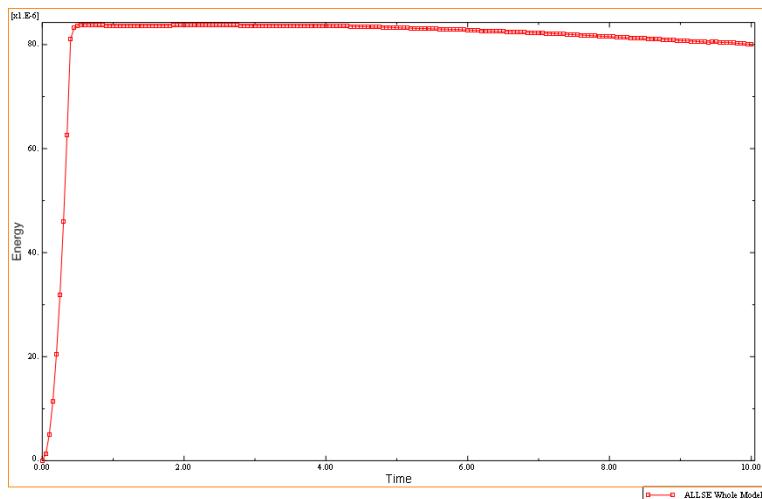


Fig. 8.10: Strain energy variation

As the step was dynamic, a history regarding the entire model was saved, providing different parameter variations, such as External work, Frictional dissipation, Plastic dissipation, or Strain energy (see Fig. 8.10). These can be accessed by going along the **Result ▶ History Output...** path.

In order to get a feeling of the actual three dimensional symmetric problem, there are possibilities to rebuild the geometry. Therefore, go to **View ▶ ODB Display Options...** and, under the Sweep/Extrude tab, check Sweep

elements, and change the maximum angle to 270°. Adding to that, in order to obtain a more accurate version of the real object, use as many segments as possible (maximum number allowed per angle unit is 1). In order to visualize the obtained three dimensional recomposed geometry, click the **Apply** button. This will create an almost precise matching three dimensional solid.

If only a simpler symmetrical overall figure is desired to be obtained, under the Mirror/Pattern tab, select the mirroring plane and finally click the **Apply** button. Another way of viewing the results is by using a Circular pattern, which will offer a similar to the sweep method, the only difference being that the user may observe internal features, as the calculate section is rotated around the symmetry axis.

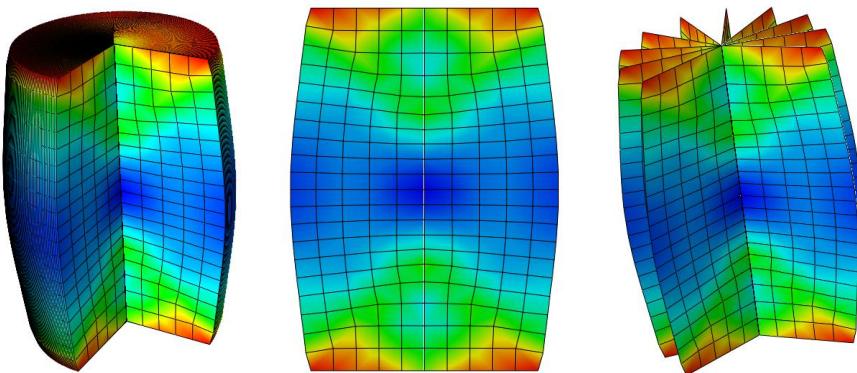


Fig. 8.11: Different ODB display options: left) sweep; centre) mirror; right) circular pattern

8.5 Results interpretation

As Fig. 8.8 displays, there are important differences between different techniques and, even if using the same meshing method, differences of approximation may appear: for example, by decreasing the element size to 50% of the initial value, a difference of 5-10% may appear. Adding to this, if a meshing technique is not appropriate to the model's or part's geometry, may

provide wrong answers, both in terms of output variable variation, and values obtained.

Another aspect that the user must take into account is to maintain an equilibrium between the refinement degree of the meshing and the calculation power available.

8.6 Things to remember

This problem presented the pros and cons of considering different meshing techniques and element sizes, with respect to the average calculation power, and needed precision.

Moreover, the sequence of stages and steps to be covered in order to obtain an axisymmetric problem has been presented. This allows the user to simplify a model's geometry and reduce its computational costs, in term of power and time.

9. MECHANICAL–PORE PRESSURE (CONSOLIDATION) ANALYSIS OF A 3D SAMPLE

9.1 Keywords

Soil consolidation, Hydro-mechanical coupling, Time-dependent analysis

9.2 Aims

- Creating a material that has both hydraulic and mechanical properties
- Subjecting the assembly to both hydraulic and mechanical boundary conditions and loads
- Creating a time-dependent calculation step, both with automatic and user imposed time stepping
- Processing the data obtained in time-output variable coordinate system

9.3 Problem description

This problem is related to a consolidation process of a saturated sample (Fig. 9.1) that is subjected to an effective isotropic stress of 200(kPa). The top and bottom surface of the sample are having as boundary condition a hydraulic pressure of 100(kPa), which is usually imposed with the aid of a pressure cell that maintains this value.

Because it is needed to obtain a 200kPa effective value of the isotropic stress while maintaining a 180kPa Back Pressure, according to Terzaghi's law of effective stresses, the sample has to be subjected to a value of 300kPa, the sum of the two. The base of the cylinder will be considered encastered.

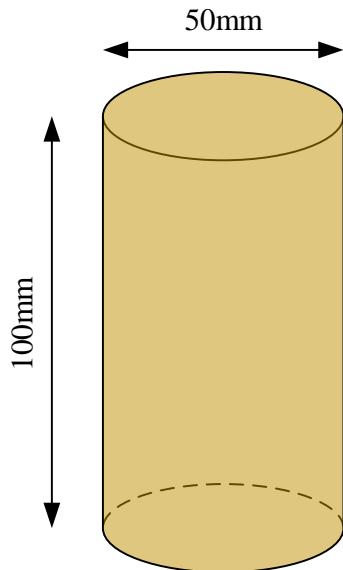


Fig. 9.1: The model geometry

9.4 Solving steps

Using either a CAD software, either the pre-processing interface of ABAQUS, build the geometry displayed in Fig. 9.1: a cylinder of 50mm diameter and 100mm height.

If the case of using an exported part, go to **Module: Part**, name it “Sample” and import it as **③D ④Deformable**.

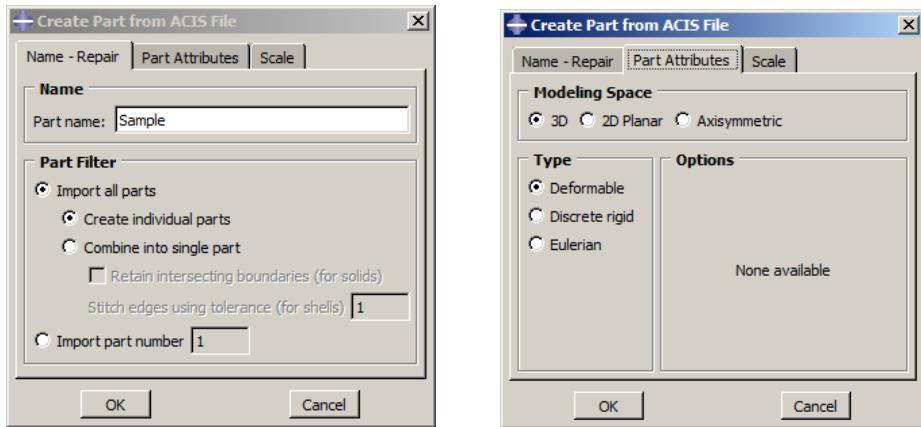


Fig. 9.2: Importing the geometry of the sample

Moving to **Module: Part**, LMB click on the SR **Create material** button, name it “Soil” and under **General ▶ Density**, fill the value of 1.8(to/m³). Moving to **Mechanical ▶ Elasticity ▶ Elastic**, input the values of 8000(kPa) for the Young’s Modulus and a Poissoin’s Ratio of 0.35. As it has already been presented in chapters 8 and **Error! Reference source not found.**, the soil behaviour may be modelled as linear elastic-perfect plastic material. Therefore, advance to **Mechanical ▶ Plasticity ▶ Mohr Coulomb Plasticity** and input the following values in the Plasticity tab: for the friction angle use a value of 17($^{\circ}$), while in case of the dilation angle consider the value 0($^{\circ}$). Moving to the Cohesion tab, use a value of 75(kPa) for the Cohesion Yield Stress and 0(%) for the Absolute Plastic Strain.

So far all the aforementioned aspects of the material have covered only the mechanical part of the problem. Moving to the hydraulic field, advance to **Other ▶ Pore Fluid ▶ Permeability**. In the newly appeared dialogue part of the Edit Material window, a table having the column heads of k (permeability) and Void Ratio appeared. The values to be filled in are presented in Tab. 9.1. Moreover, at the Type, check that Isotropic is chosen, and input for the Specific weight of wetting liquid the value of 10(kN/m³). In the end, LMB click **OK** in order to finish the material definition sequence.

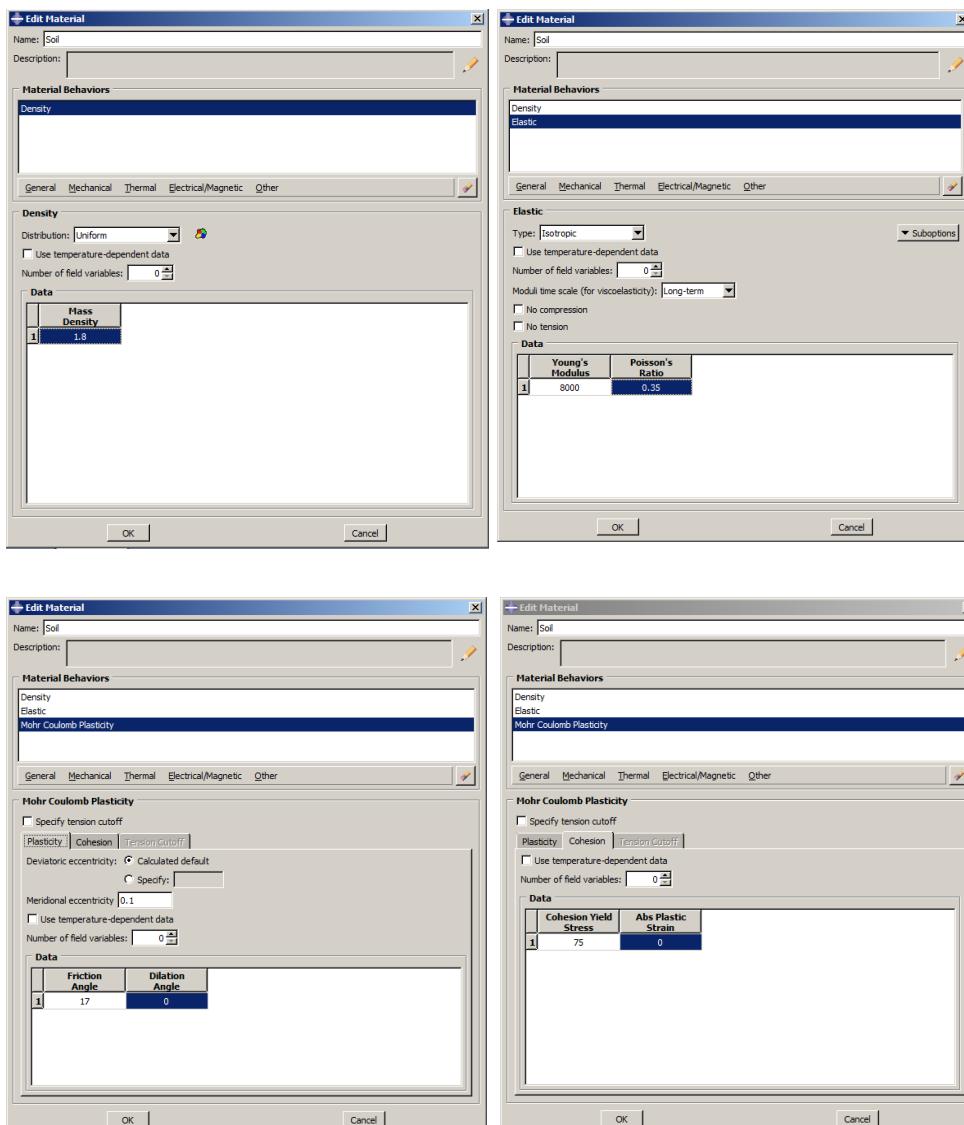


Fig. 9.3: Creating the mechanical properties of the material: top-left) Density assigning; top-right) Linear elastic properties; bottom) Plastic Mohr Coulomb properties

Tab. 9.1: Permeability as function of the Void Ratio

k (m/s)	Void ratio (-)
1E-0010	0.3
8E-0010	0.5
2.5E-009	0.7
1E-008	0.8

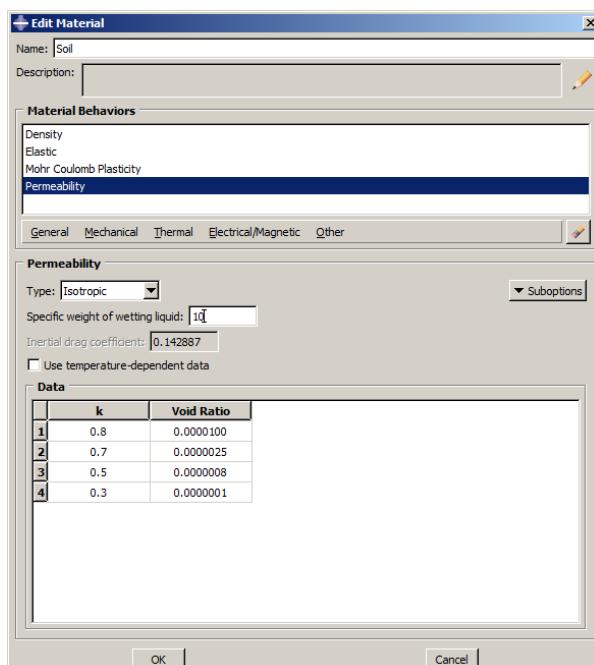


Fig. 9.4: Creating the hydraulic properties of the material

Further, create a section using **SR** **Assign Section**, named “Sample”, making sure that the **○Solid** is checked and select the Type as Homogeneous. Clicking **Continue ...** brings forth the Edit Section window, where the material “Soil” is to be chosen. End the sequence by clicking **OK**.

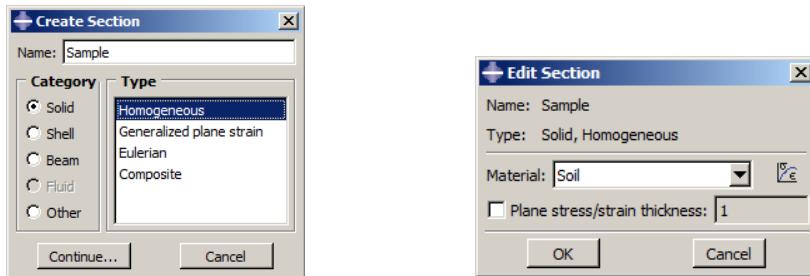


Fig. 9.5: Creating the section and assigning the material

Advancing to **Module: Assembly**, LMB click the **SR** Instance Part button, check Independent (mesh on instance) and finish the procedure by clicking the **OK** button

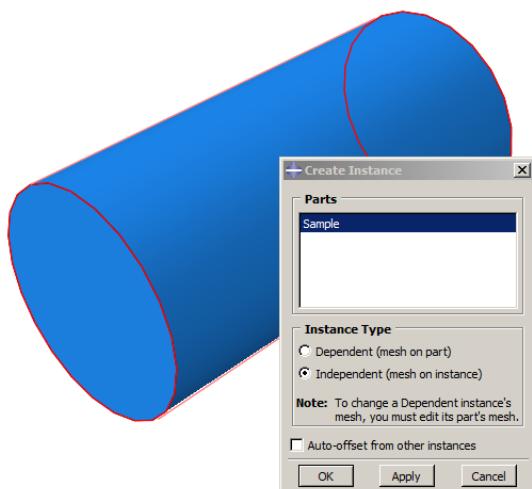


Fig. 9.6: Importing the part to the assembly

In the **Module: Step**, click the **SR** Create Step button, and name the calculation step “Consolidation”. The default Procedure type is General and the type of calculation step is Static, General. Change to Procedure type Soils and Click the **Continue ...** button, and a new window will appear. In the description field, write “Consolidation stage”. Change the time period from the default value 1, to 86400, meaning a full 24hours time (usually a sample

will be monitored just during this period). In order to ensure the ease of calculation, at the Automatic stabilization option, choose Specify dissipated energy fraction, leaving the default values. Leaving the Basic tab, go to Incrementation tab. Here, the user may impose the stepping of the calculation and the maximum pore pressure change per increment. In the case of the increment size, change the displayed values to: initial 1(s), the minimum to 0.25(s) and the maximum to 3600(s). It is also highly advisable to increase the maximum number of increments. A value of 250 will be used. Uncheck the Creep/swelling/viscoelastic strain tolerance and input a value of 500(kPa) to the maximum pore pressure change per increment. Although theoretically one expects a maximum value of 200(kPa), due to the Mandel-Cryer effect (the rise of the pore pressure above the value determined as the difference between the isotropic stress and the back pressure applied to the sample). Click the **OK** button, in order to complete the creation of the calculation step.

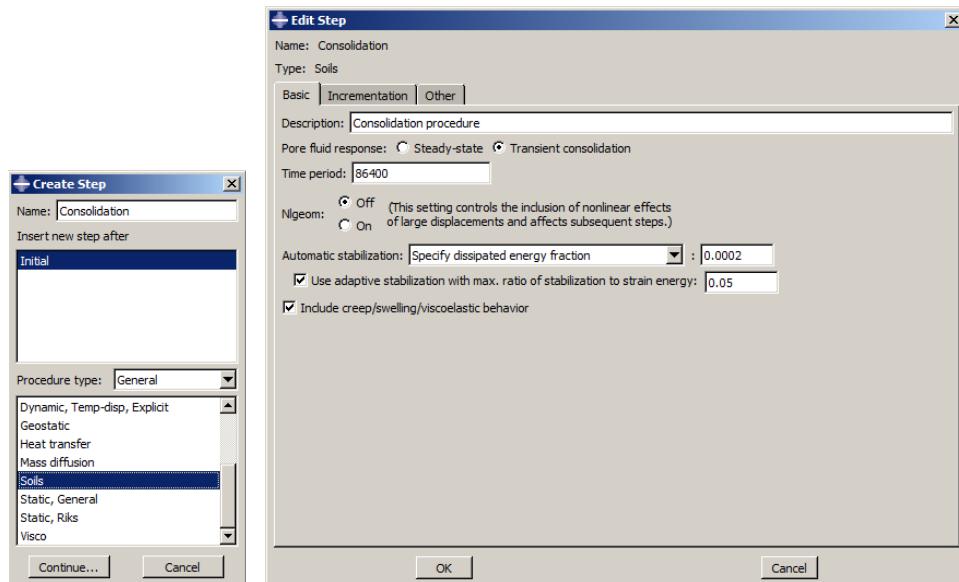


Fig. 9.7: Creating the consolidation step

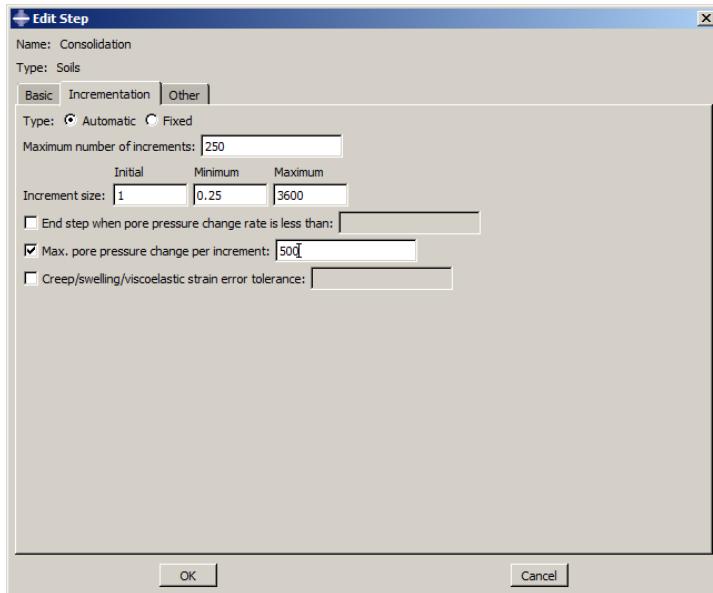


Fig. 9.7: Creating the consolidation step (continued from the previous page)

One of the aims of this example is to monitor the variation of the pore pressure in the middle of the sample. Therefore, go to SR **Create Datum Plane: Offset From Principal Plane** button, LMB click it and when prompted to choose which the reference plane to offset is, choose XY Plane. The offset you have to input is 0.05(m), as it represents half of the height of the considered soil sample. Now, go to SR **Partition Cell: Define Cutting Plane** and LMB click and hold until the Partition Cell menu appears (Fig. 9.8) and choose **Partition Cell: Use Datum Plane**. Select the plane previously created and finish the sequence by LMB clicking the bottom menu's **Create Partition** button.



Fig. 9.8: Partition Cell menu

Switch again to **Partition Cell: Define Cutting Plane**, choose the two halves of the sample, and select the 3 Points technique. Pick any available

three points that forms a plane parallel to ZY. Select all the pieces once more and, using the same technique, create a perpendicular cutting plane.

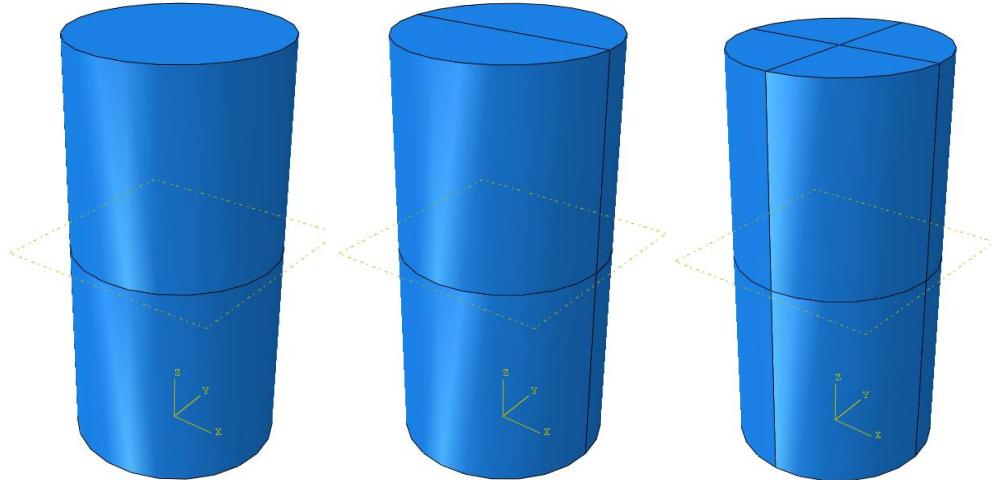


Fig. 9.9: Partitioning the sample

Go to **Tools ▶ Set ▶ Create...**, name it “Centre”, click **Continue ...**, and when prompted for the geometry of the set, hide the top half of the sample in order to choose the middle, top and bottom points.

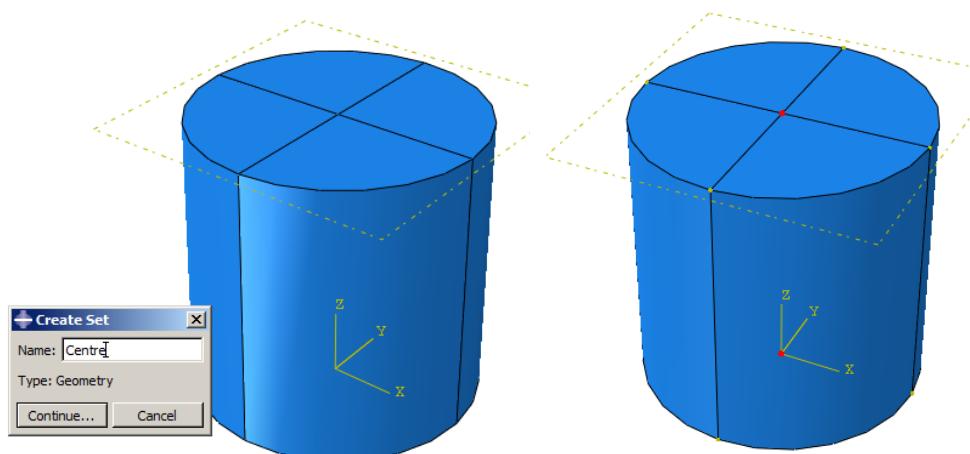


Fig. 9.10: Creating the Set: left) Create Set window; right) Selecting the interest point

Now, go to **Output ▶ History Output Requests ▶ Manager...** and click the **Edit ...** button. For the domain, select Set, and choose the set previously created “Centre”. The frequency will remain default at every 1 increment. For our points it is of interest to monitor the pore water pressure, effective stress, and the possible displacements. Therefore, check Stresses, Strains, Displacement/Velocity/Acceleration and Porous media/Fluids groups of output variables (). Click the **OK** button in order to save the modifications.

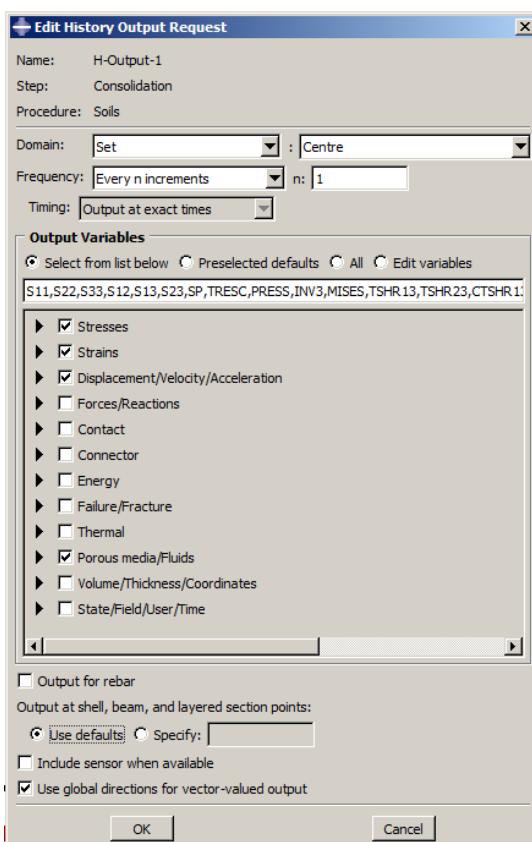


Fig. 9.11: The Edit History Output Request window

Further, we go to the **Module: Load**, and impose the encastre condition on the bottom of the sample. Go to the **SR** **Create Boundary Condition** and the supporting condition “Encastre”, select Mechanical and

Symmetry/Antisymmetry/Encastre type and click **Continue ...**. Choose the bottom of the sample and check **Encastre**. Finally click **OK**.

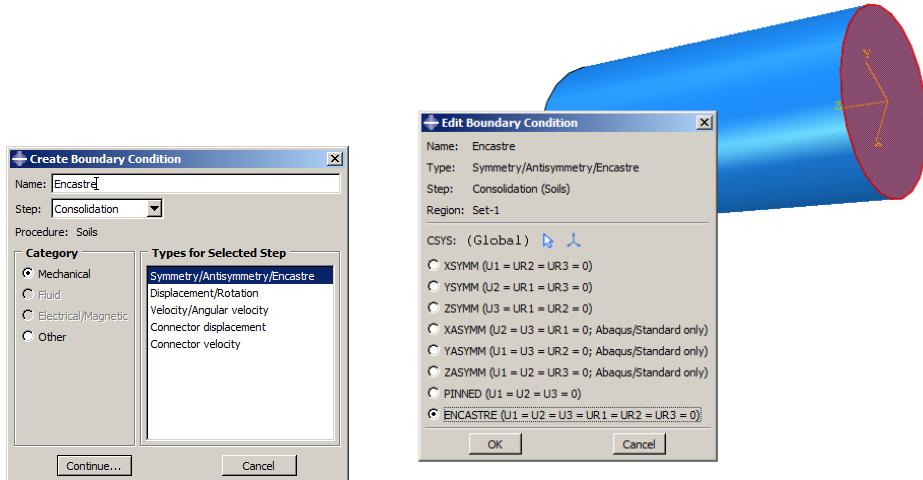


Fig. 9.12: Creating the mechanical boundary condition

In order to assign hydraulic boundary conditions, go to **SR** **Create Boundary Condition** and choose Other, selecting afterwards Pore pressure as type. Name it “Back Pressure”. Next, after clicking **Continue ...**, choose the top and bottom surfaces of the sample, where the hydraulic gradient is going to be imposed, click **Done** and finally, at Magnitude, fill the value of 100(kPa).

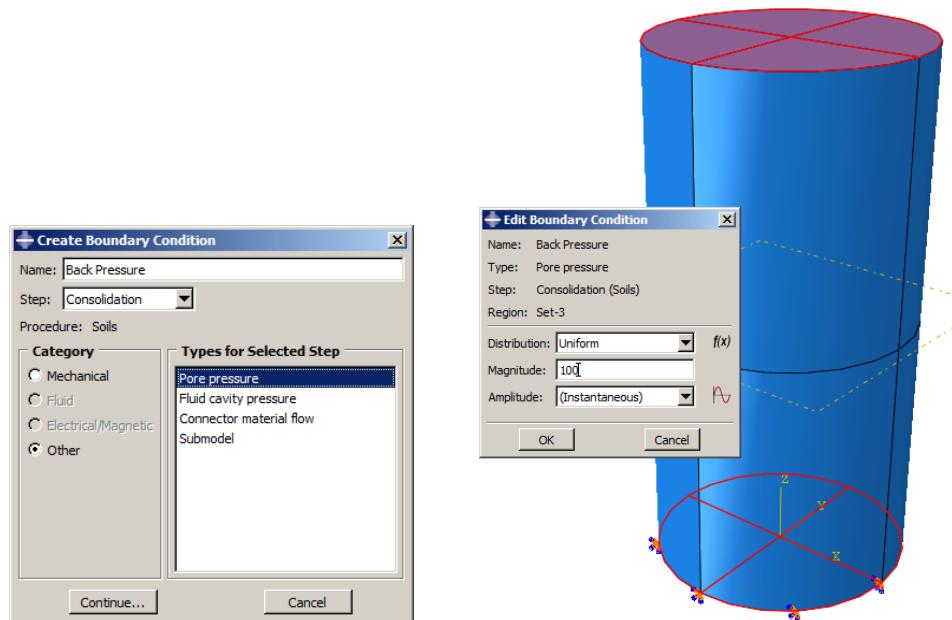


Fig. 9.13: Creating the hydraulic boundary condition

Arriving to the external loads, it has been shown in the 9.3 that an isotropic stress of 300(kPa) must be applied to the sample. Therefore, go to **SR**  **Create Load**, and select all the faces of the sample, except for the lower encasted one, and apply the pressure. This side will be automatically subjected to the same value of stress, as it is supported and the reaction will have that exact value. Name the load “Cell Pressure”.

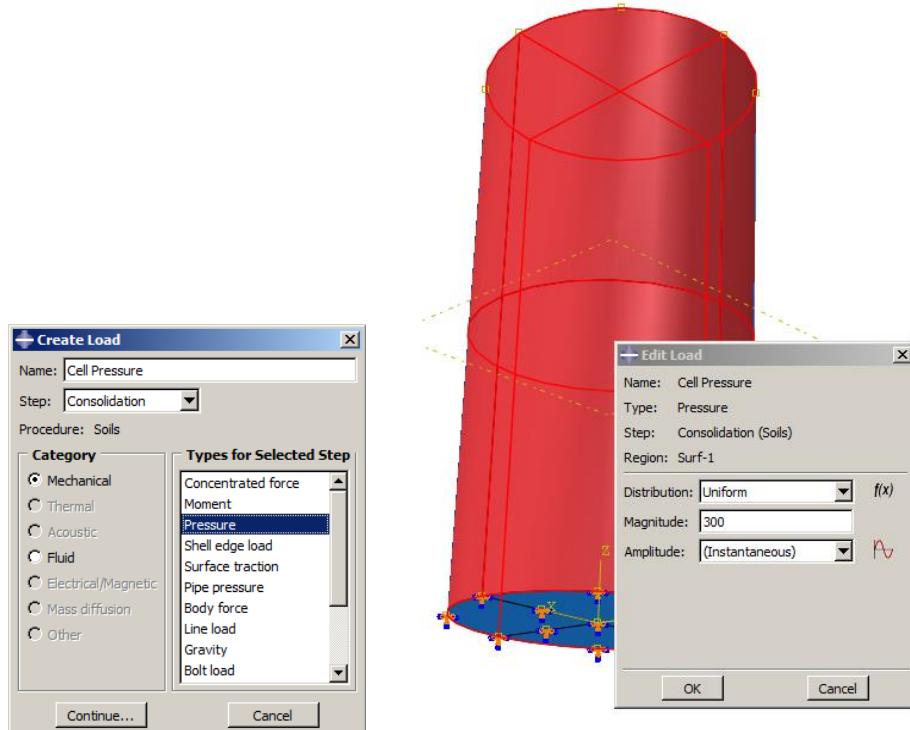


Fig. 9.14: Creating the Cell Pressure stress

As it can be observed, during the material creation, a tabular form of permeability as function of the void ratio. In order to impose the sample its initial void ratio value, go to **Predefined Field** ► **Create...**, name it “Void Ratio”, change the step to “Initial”, and under the category Other, select Void Ratio. Click **Continue ...**, select all the sample, and in the Voids ratio impose the value of 0.78 (Fig. 9.15).

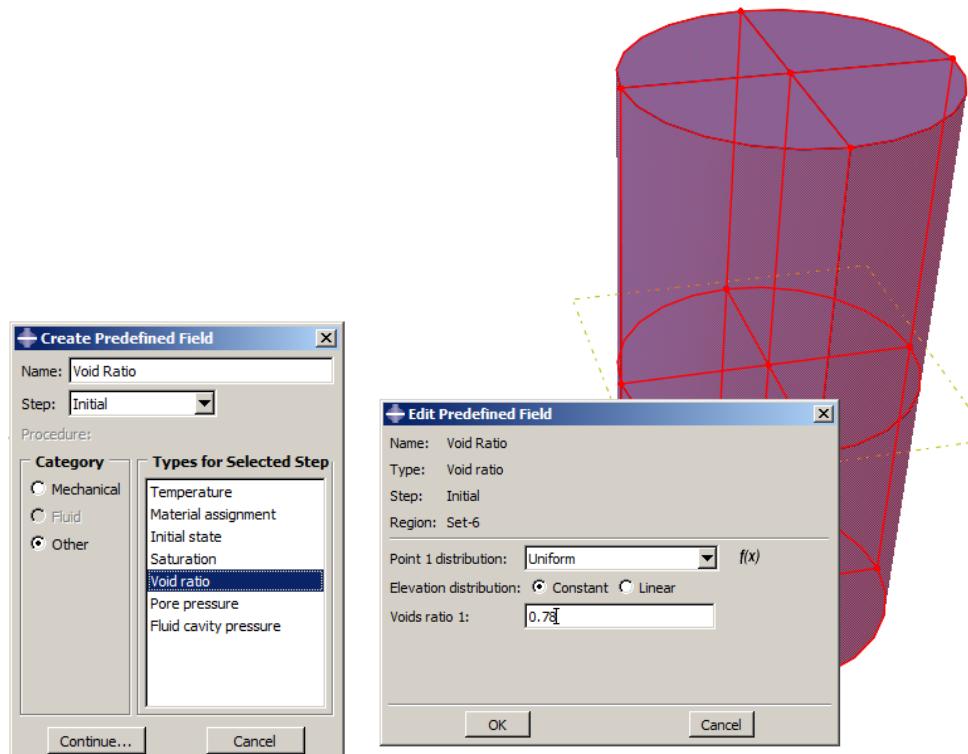


Fig. 9.15: Defining the initial void ratio

Advancing to **Module: Mesh**, seed the soil sample using the **SR Seed Part Instance** at a maximum element size of 5mm (0.005). Mesh the part using **SR Mesh Part Instance**.

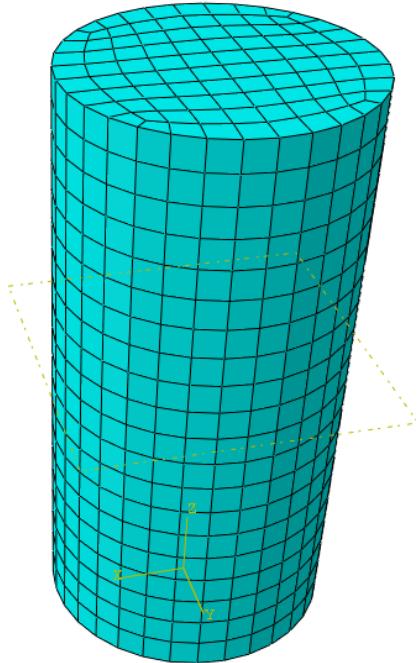


Fig. 9.16: The meshed soil sample

Up to this moment all the previously presented problems and exercises were simple mechanical ones and no other degree of freedom was presented. In this case, as it has been mentioned since the Aims chapter, the coupled hydro-mechanical phenomena of consolidation is to be modelled. Therefore, the Element Type cannot be the default 3D Stress, but changed to Pore Fluid/Stress. Go to SR Assign Element Type, select the sample, and choose Pore Fluid/Stress Family. In the lower part of the Element Type window, the code of the element changed from C3D8R (Continuous 3D element, 8 node brick, Reduced integration) to C3D8P (Continuous 3D element, 8 node brick, Pore pressure). Finally, click **OK**.

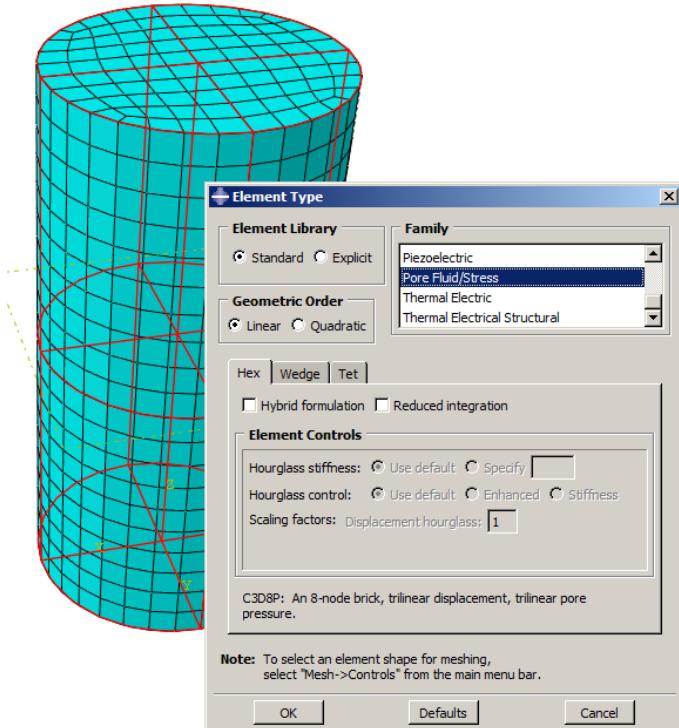


Fig. 9.17: Changing the element type

Under the **Module: Job**, create a job named “Consolidation” and submit it to the calculation process.

Following the job execution, in the “Consolidation” Monitor window, accessed from the Job Manager window by LMB clicking the **[Monitor ...]** (Fig. 9.18), it can be observed that for the column Time/LPF Inc (marked with red), each calculation increment has a period starting from 1(s), the initial condition given during the calculation step creation, growing up to the 3600(s) maximum limit. The last increment covers a period of time smaller than the maximum limit, but with a value calculated by the software so that the total time (marked with orange) reaches the final value of 86400(s).

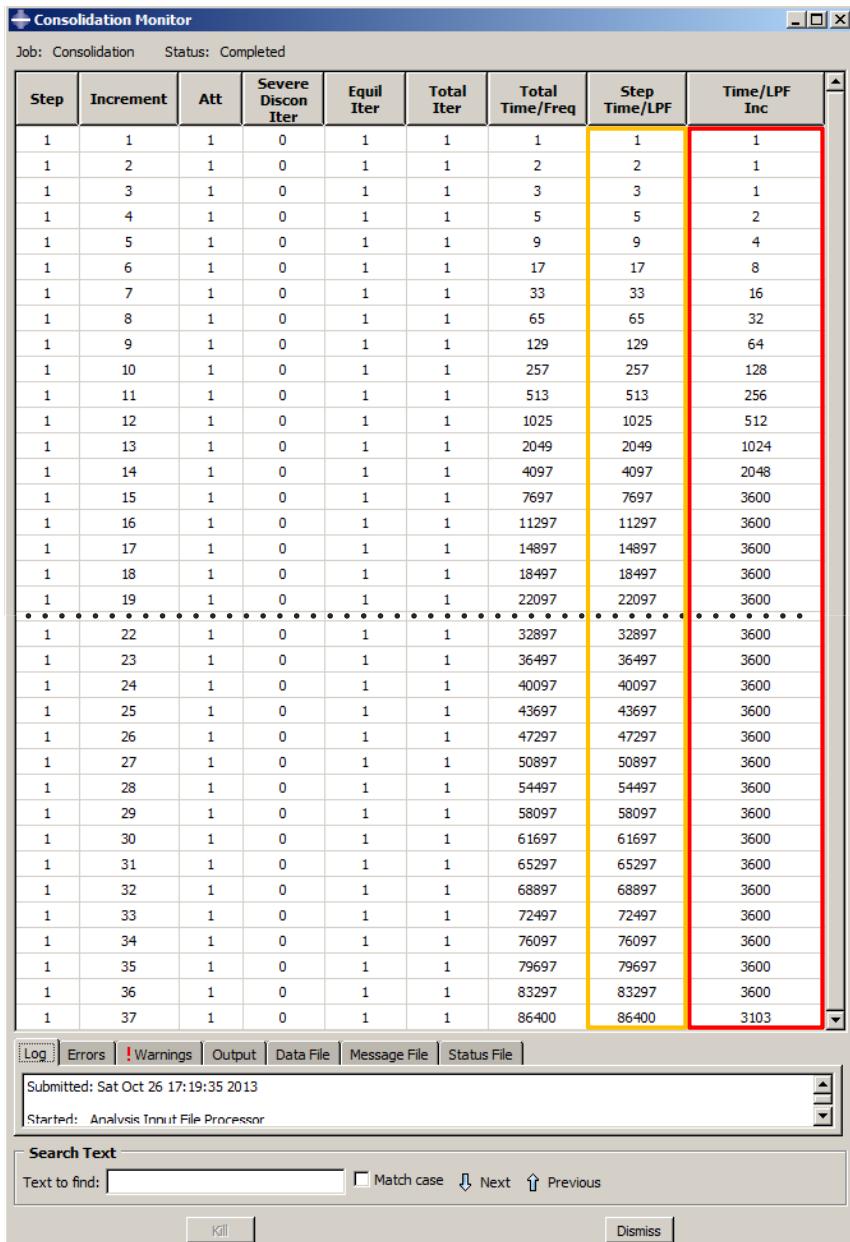


Fig. 9.18: The Monitor window

Moving to the **Module: Visualization**, it is of great interest to observe the variation of the pore pressure inside the soil sample, taking into account also

the drainage surface, and the deformations at top. In order to obtain this data, demanded during the creation of the calculation step, by means of History Output Requests, go to **Tools** ► **XY Data** ► **Create**, and check ODB history output, click **Continue ...**, and, if the user wishes to save the data, click on the **Save as ...**, but if it is desired to plot the variation, select the output variable and the point for which it was saved (multiple outputs and points may be drawn at the same time) and click **Plot**. For example, if we consider the pore water pressure and the centre of the sample, one obtains the following graph, displayed in Fig. 9.23.

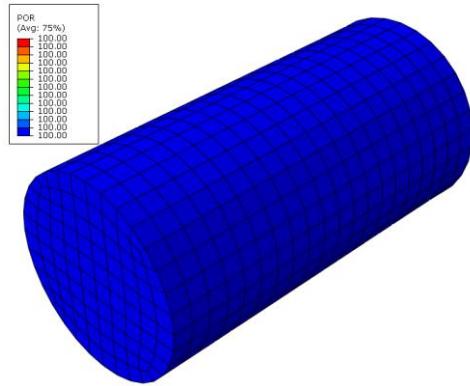


Fig. 9.19: Pore pressures at the end of the calculation step

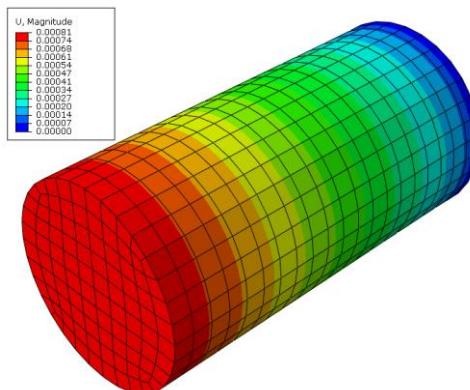


Fig. 9.20: Deformations at the end of the consolidation step

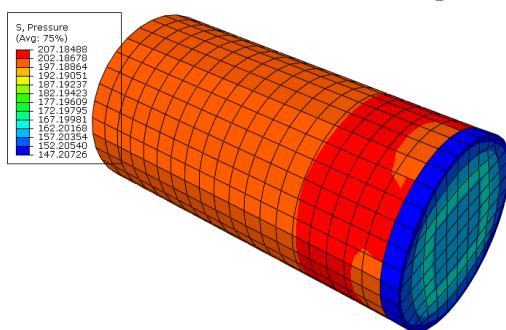


Fig. 9.21: The effective pressure acting on the sample

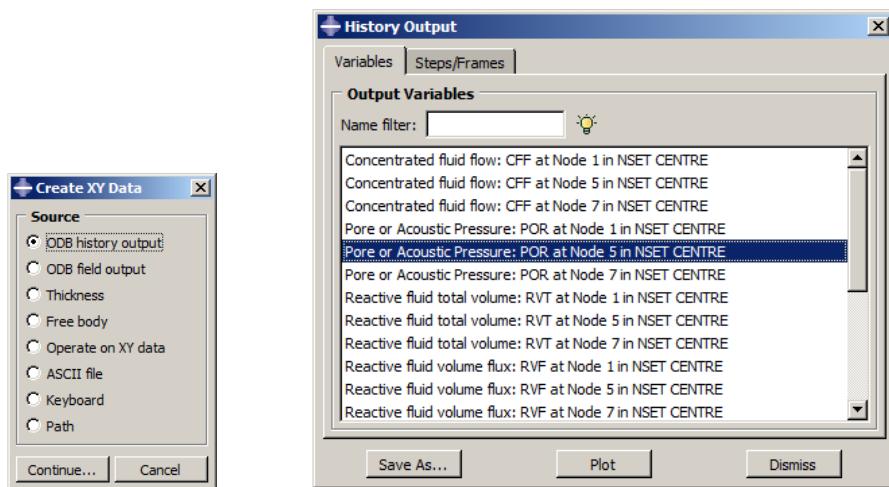


Fig. 9.22: Obtaining the history data output

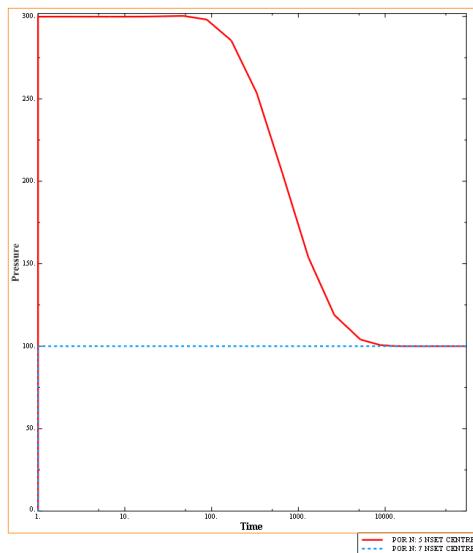


Fig. 9.23: Pore pressure variation of the middle and top of the sample

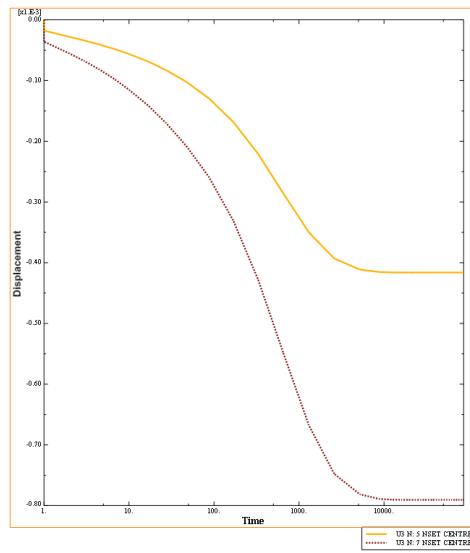


Fig. 9.24: Vertical displacement variation of the middle and top of the sample

9.5 Results interpretation

The consolidation model has been accurately described, the overall form of the displacements graphs and pore pressure variations displaying the theoretical forms. An overall stabilization has been determined at around 10500(s).

If we select the Pore pressure (POR acronym), for the end of the consolidation step we will obtain an uniform value of 100(kPa) (Fig. 9.19), equal to the applied Back Pressure. Observing the first calculation iteration, at 1(s), the Mandel-Cryer effect created an added 100(kPa) pore pressure (see Fig. 9.23).

From the point of view of the displacements variation (Fig. 9.24), it is easily observable that the top point (marked with purple) has double strain than the one at the middle (denoted with yellow), which corresponds to the initial distances between themselves and the supporting point.

9.6 Things to remember

This problem treated the subject of the hydro-mechanical coupling, using as exemplification phenomena the transient soils' consolidation. A simple, Lagrangian approach has been used, appropriate to daily use, but in the case of advancing towards more complex models, either Euler modelling or even CFD (Computational Fluid Dynamics) may be taken into consideration.

Moreover, the results have been extracted either graphically, using the available plot technique, either saved as loadable data files.

10. CRUSHABLE PARTICLES PROBLEM

10.1 Keywords

SPH (Smoothed Particles Hydrodynamics), Cut-off limit, Crushable material

10.2 Aims

- Imposing and using the cut-off limits of materials
- Usage of the SPH technique to overcome large strains FEM problem

10.3 Problem description

A concrete sphere of 2.0m diameter is subjected to a compression test. Two infinitely rigid planes of 4.0 by 4.0m are positioned at the top and bottom of the sample. The one on top will have a constant speed motion towards the other one, having a speed of 2m/s, subjecting the concrete to compression.

In the case of this problem, as a difference from the other already presented, a comparison between the model using the SPH option and the same without SPH, in order to emphasize its role.

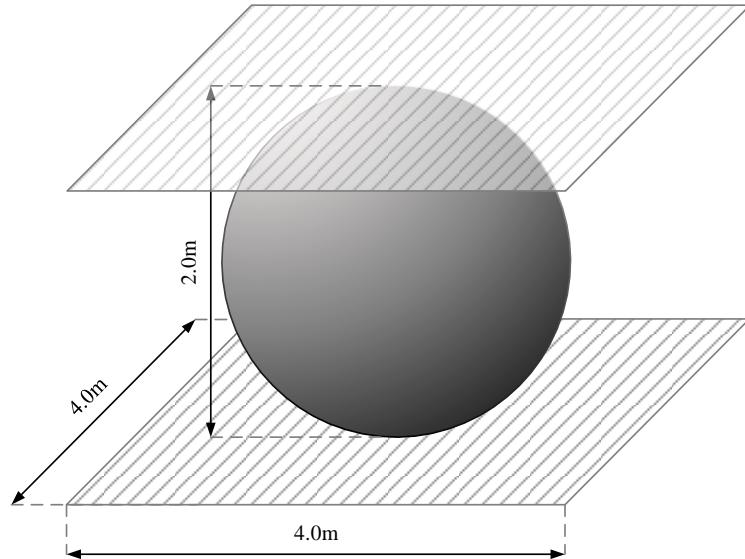


Fig. 10.1: Model's geometry

10.4 Solving steps

The geometry of the cylinder and the two planes will be obtained using, at user's choice, either a CAD software, either ABAQUS Sketch preprocessing module.

Import the parts to ABAQUS, under the **Module: Part**, by following the path **File ▶ Import ▶ Part...**, and name them “Top Plane”, “Bottom Plane” and “Sample” (Fig. 10.2). All the parts are to be declared $\odot 3D$ and $\odot Deformable$.



Fig. 10.2: The three considered parts: left) Bottom Plane; centre) Sample; right) Top Plane

Moving to **Module: Property**, create the material using the SR  **Create material** button. Under the name “Concrete”, go to:

- **General:** **Density** and input the value of 2.5(to/m³);
- **Mechanical:** **Elasticity ► Elastic** and use the 2.1E7(kPa) for the Young’s Modulus and 0.2 for the Poisson’s Ratio;
- **Mechanical:** **Plasticity ► Concrete Damaged Plasticity** and insert the values of 38(°) for the Dilation Angle, 1(%) for Eccentricity, 1.12 for the f_{b0}/f_{c0} , 0.1 for K and 0.666 the Viscosity Parameter; moving to Compressive behaviour, insert the tabular data presented in Tab. 10.1, and for the Tensile behaviour the one displayed in Tab. 10.2;
- Under the Compressive behaviour tab, click **Suboptions ► Compression Damage** and input the data displayed in Tab. 10.3;
- For the Tensile behaviour tab, click **Suboptions ► Tension Damage** and fill in the table with the data from the Tab. 10.4.

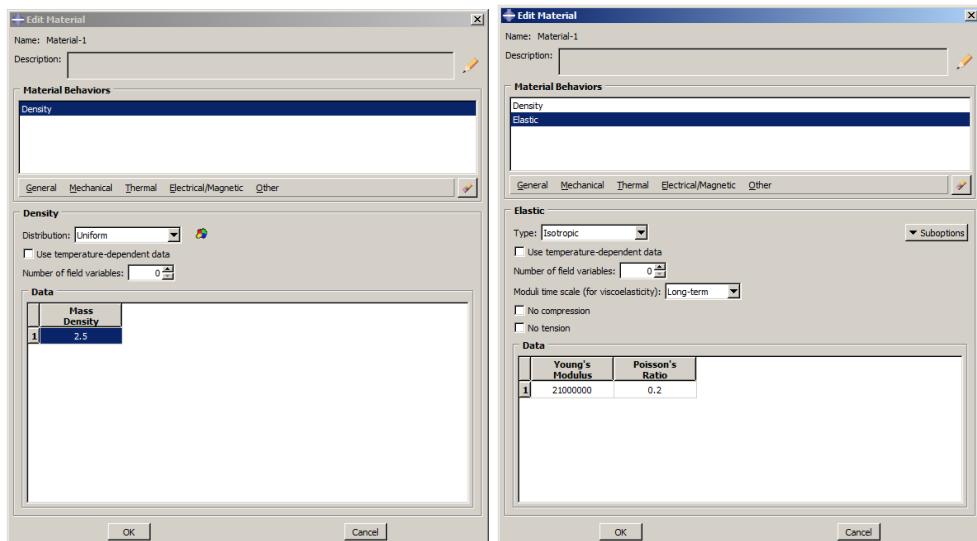


Fig. 10.3: Creating the material

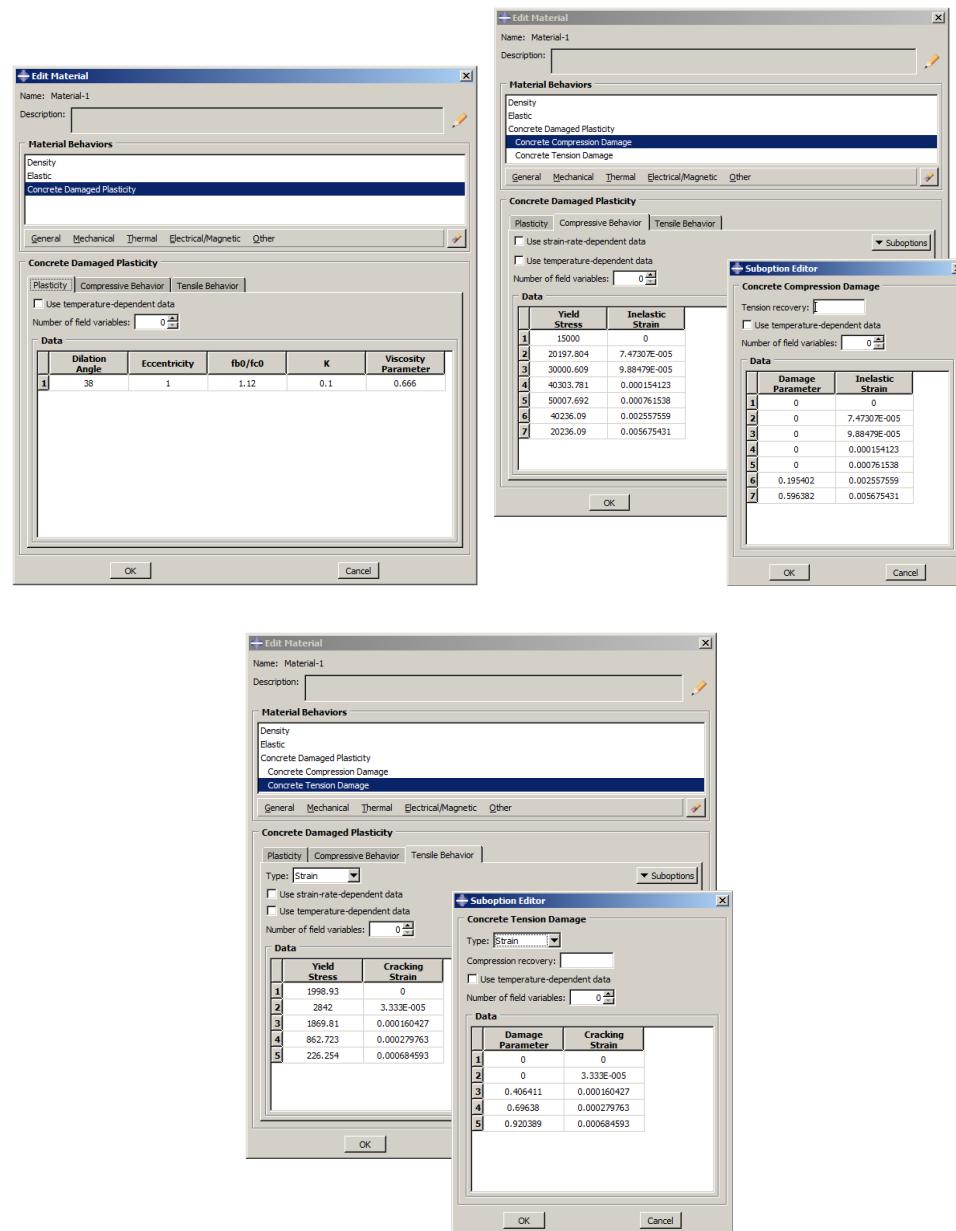


Fig. 10.3: Creating the material (continued from the previous page)

Tab. 10.1: Concrete Compressive behaviour test data

Yield Stress (kPa)	Inelastic Strain (-)
15000.000	0
20197.804	0.000074730
30000.609	0.000098847
40303.781	0.000154123
50007.692	0.000761538
40236.090	0.002557559
20236.090	0.005675431

Tab. 10.2: Concrete Tensile behaviour test data

Yield Stress (kPa)	Cracking Strain (-)
1998.93	0
2842	0.00003333
1869.81	0.000160427
862.723	0.000279763
226.254	0.000684593

Tab. 10.3: Concrete Compression Damage test data

Damage Parameter (-)	Inelastic Strain (-)
0	0
0	0.0000747307
0	0.0000988479
0	0.000154123
0	0.000761538
0.195402	0.002557559
0.596382	0.005675431

Tab. 10.4: Concrete Tension Damage test data

Damage Parameter (-)	Cracking Strain (-)
1998.93	0
2842	0.000033330
1869.81	0.000160427
862.723	0.000279763
226.254	0.000684593

End the action by clicking the **OK** button. In order to assign the material to the sample, create a section by going to **SR**  **Create Section**, name it “Concrete” also, and check **⊖Solid**, type Homogeneous. Finally assign the section to the sphere.

For the two rectangles, create a section having a linear elastic material with the properties of steel: Young Modulus of 210E6(kPa) and Poisson’s Ratio of 0.3. The section you need to create – “Steel Sheet” is a **⊖Shell**, type Homogeneous, having a thickness of 0.01(m) and 11 integration points.

Advancing to **Module: Assembly**, import the three parts by using **SR**  **Instance Part** button. Check the **⊖Independent** and click **OK** (see **Error! reference source not found.**).

Cut the two shells in four equal squares, as Fig. 10.4 displays, using the **SR**  **Partition Face: Use Shortest Path Between 2 Points** tool. Go **Tools ► Reference point...** and select the two centre points created for each shell. **LMB** click the  **Cancel** button in the bottom menu.

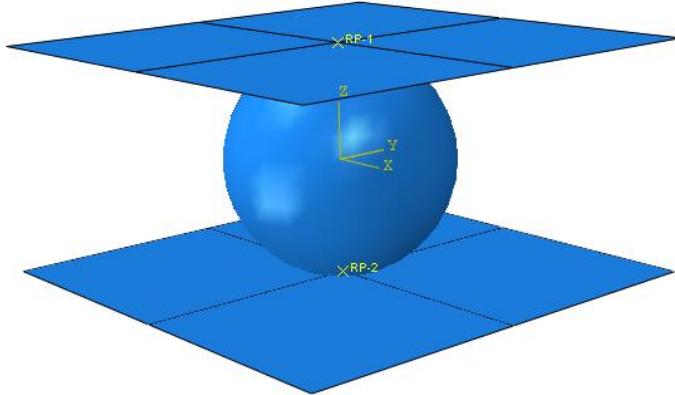


Fig. 10.4: The final state of the assembly

Advance to **Module: Step**, and click the **SR** **Create Step** button. Create a calculation step named “Crush”. In this case select Dynamic, Explicit, as displayed in Fig. 10.5 , having a period of 0.02(s).

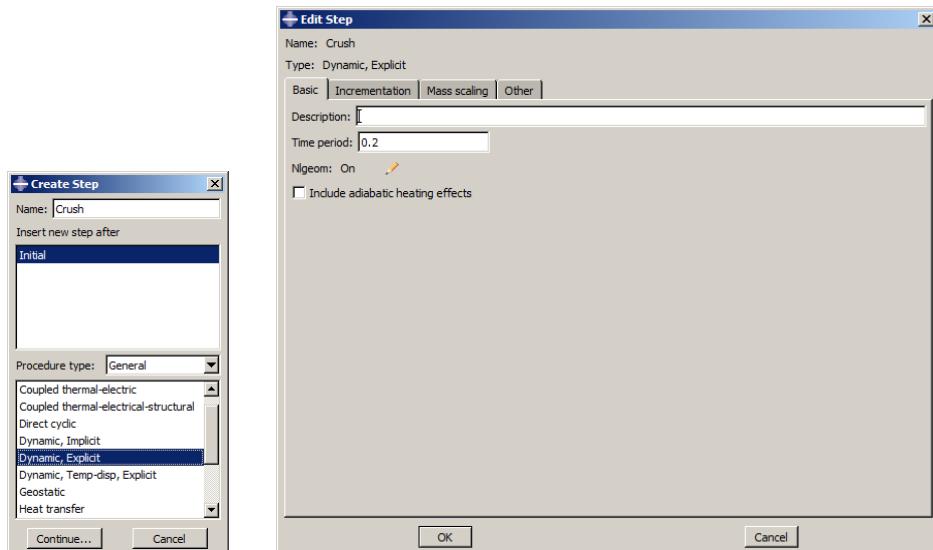


Fig. 10.5: Creating the calculation step

Moving to **Module: Interaction**, go to **SR Create Interaction Property**, name it “Contact”, and along the path **Mechanical ► Tangential behaviour**, choose the Penalty friction formulation and input the 0.3 friction coefficient. Going to **Mechanical ► Normal behaviour**, select “Hard” Contact, and check the Allow separation after contact option. This option models the platen (steel) – concrete interaction that acts during an actual compression test.

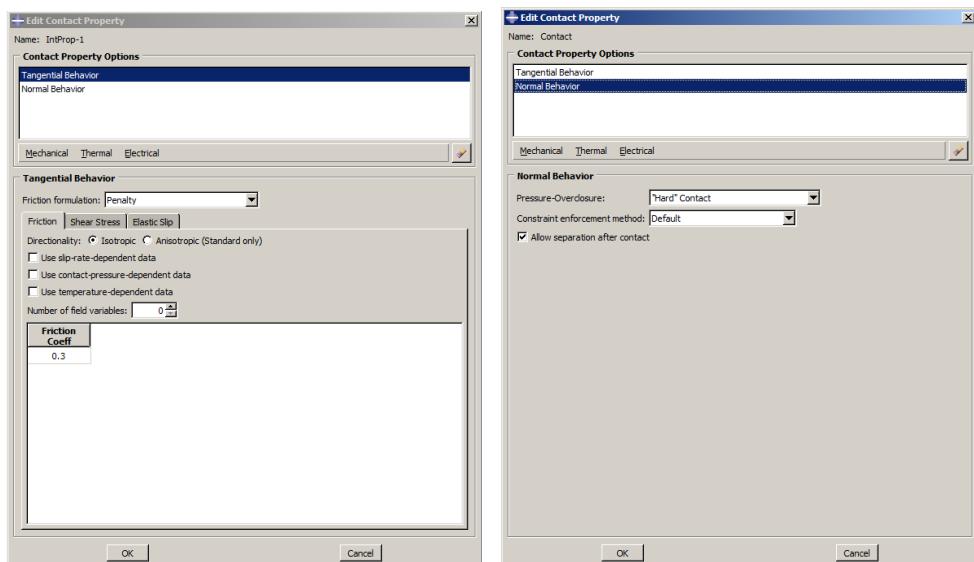


Fig. 10.6: The contact property definition

Now go to **SR Create Interaction** and, in the Create Interaction window, name it “Interaction” and select for the Step – Initial and General Contact (Explicit). In the Edit Interaction dialogue window, please observe the check on the option All with self. This enables the situation that the interaction property created (“Contact”) is enabled whenever two parts of the assembly will reach contact situation. In order to end this phase, click **OK**.

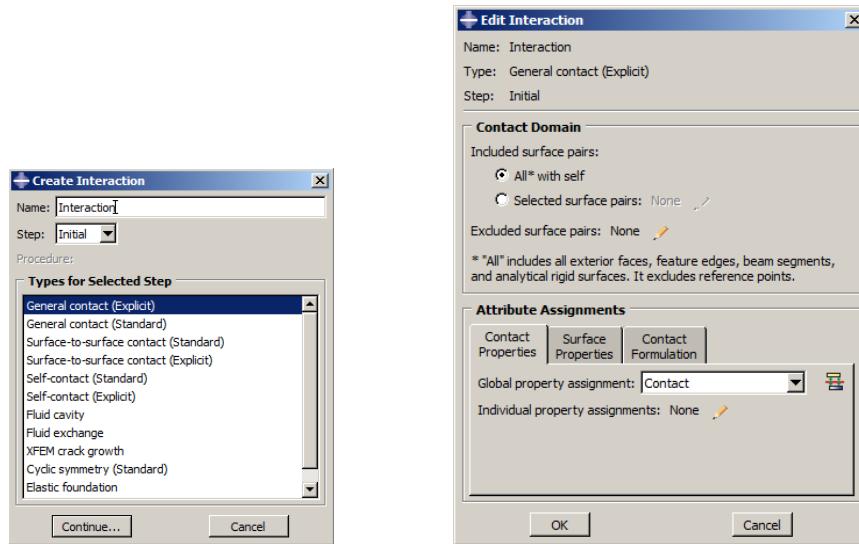


Fig. 10.7: Creating the interaction

Go to the SR **Create Constraint** button, LMB click it, and select the Rigid body type of constraint, naming it “Bottom platen”. Click **Continue ...**, go to the Region type menu, select Body (Elements), click on the cursor and select the four squares that form the lower platen and finish the sequence by clicking the **Done** button in the bottom menu. In the end, for the Reference Point, choose the reference point situated in the middle of it. Check the Adjust point to the centre of mass at start of analysis and finally click **OK** (see Fig. 10.8).

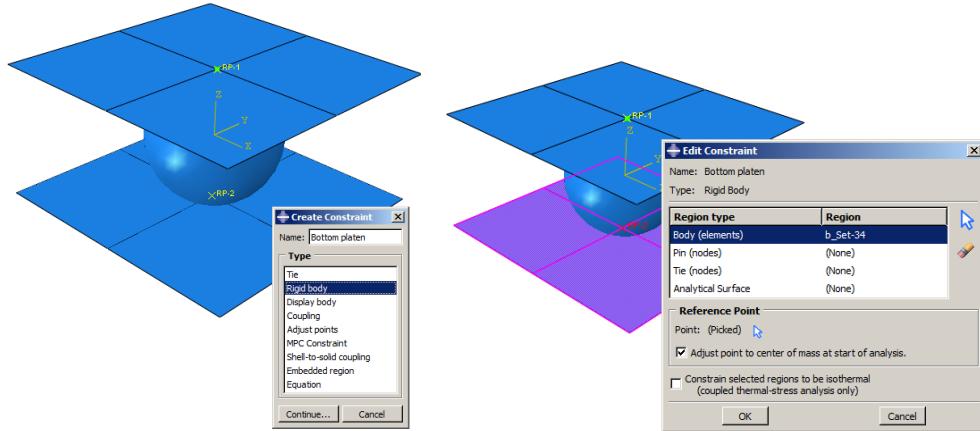


Fig. 10.8: Creating the Rigid Body constraint

Repeat the sequence in order to create a second rigid body for the top platen.

Leaving this module and heading to **Module: Load**, regarding the top platen, for the first step, support it by using Displacement/Rotation type and check U1, U2, UR1, UR2 and UR3. Go again to the **SR** **Create Boundary Condition**, name it “Advance”, choose the second step where to be created, select again Displacement/Rotation type and click **Continue ...**. Select the top platen, and in the Edit Boundary Condition check U3 and input the values of 2(m). This will lead to a maximum displacement (only as translation along the 0Z or 3 axis) of the top platen of 40cm over the 0.2s period.

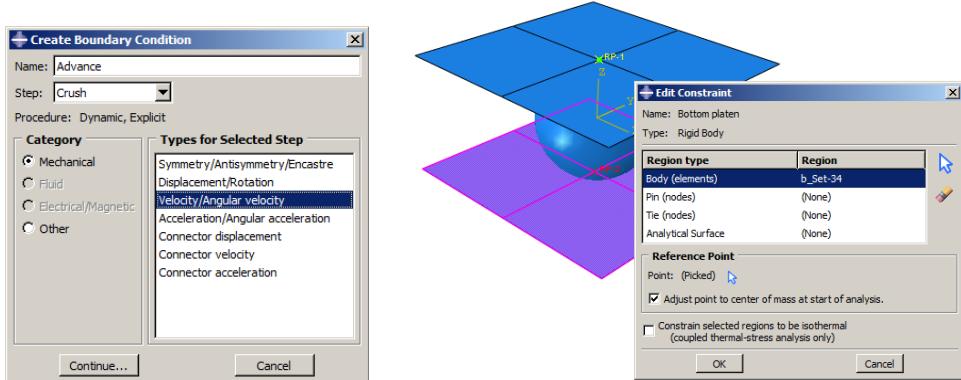


Fig. 10.9: Creating the velocity boundary condition on the platen

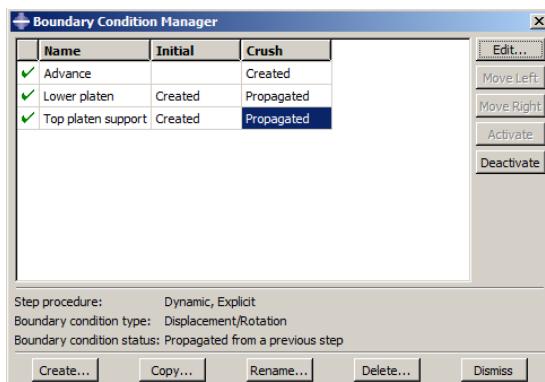


Fig. 10.10: The boundary conditions applied to the model

As explained in the 10.3 chapter, this exercise will be solved for both cases of taking and not taking into consideration the SPH technique. From this point on, all the command sequences and explanations given are related only to the case of using SPH. The classical approach should be possible to solve without any more explanations, based on the experience gained from the previously presented problems.

Advancing to **Module: Mesh**, impose a seed on the sphere of 0.15(m), by using the **SR** **Seed Part Instance** button, and from the Mesh Controls window, accessed by using the **SR** **Assign Mesh Controls**, check **Tet**. Finally, discretize the sample.

The two platens should be seeded at a maximum value of 0.20(m), as quad structured elements.

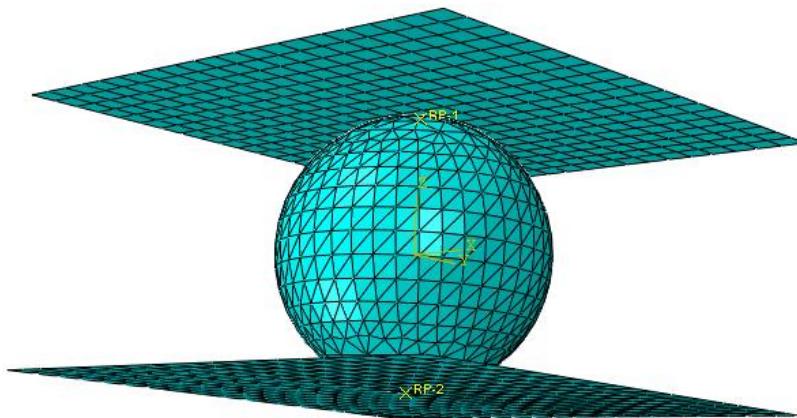


Fig. 10.11: The meshed model

Now, go to SR  Assign Element Type, and choose, for the cylinder, under the Geometric Order Linear, and for the Element Controls tab, at the Conversion to particles option - Yes. The chosen conversion criterion will be Strain, which you may pick from the drop down menu. Set the Threshold at 0.0002 (2%), which is acknowledged limit at which the concrete starts cracking. For the PPD (Particles Per Parent) set the value 4. This will generate 4 particles for each cell that met the threshold criteria, formulated in terms of strain.

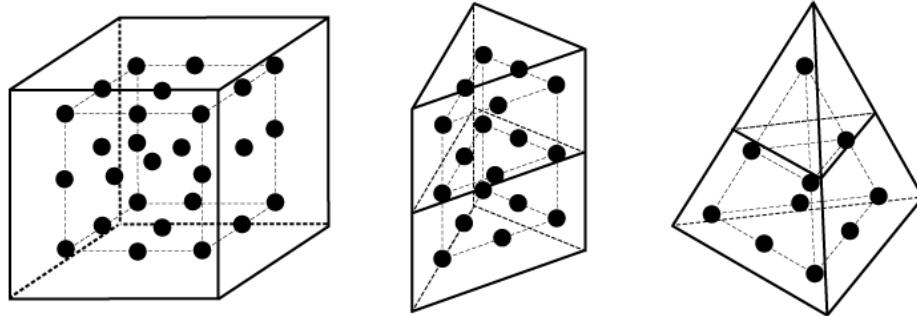


Fig. 10.12: Internally generated particles per parent element illustrated for three particles per isoparametric direction as presented in ABAQUS User's manual

In the end, click **OK** to end the command sequence.

Go to **Module: Job**, create a job named “Concrete SPH” and submit it to being calculated. Following the calculation process, the following results are to be obtained. All of the displayed figures are to be considered as a comparison.

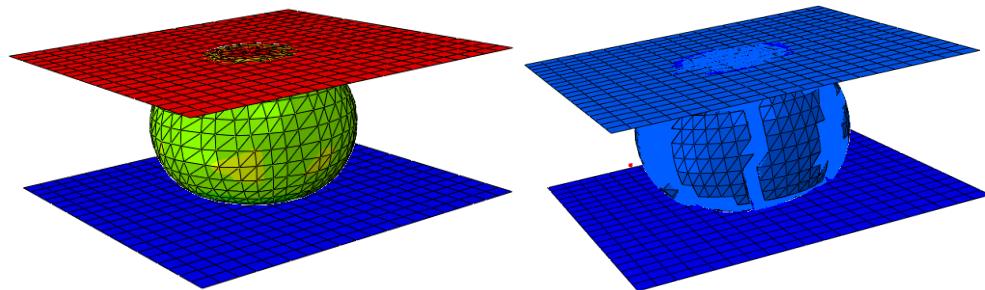


Fig. 10.13: Displacements on the analysed model: left) Classical FEM approach; right) FEM SPH approach

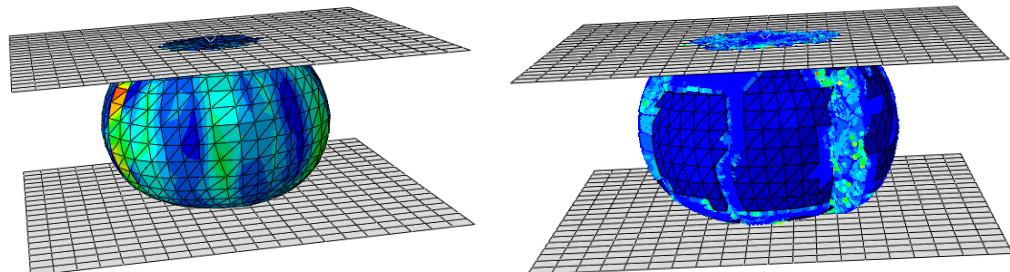


Fig. 10.14: Plastic strain on the analysed model: left) Classical FEM approach; right) FEM SPH approach

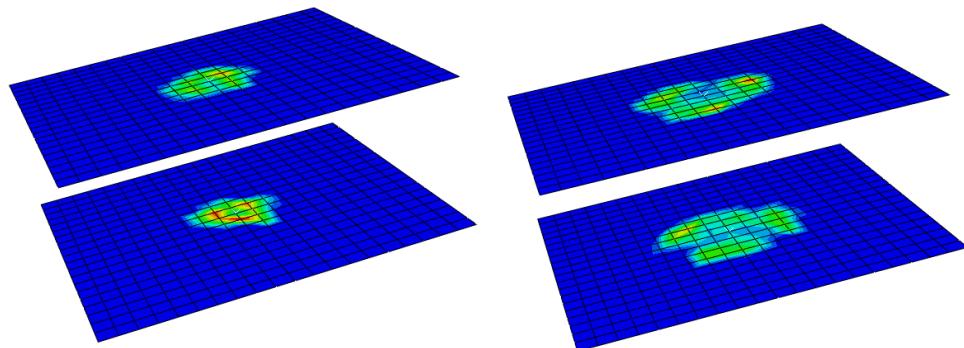


Fig. 10.15: Contact pressures on the two platens: left) Classical FEM approach; right) FEM SPH approach

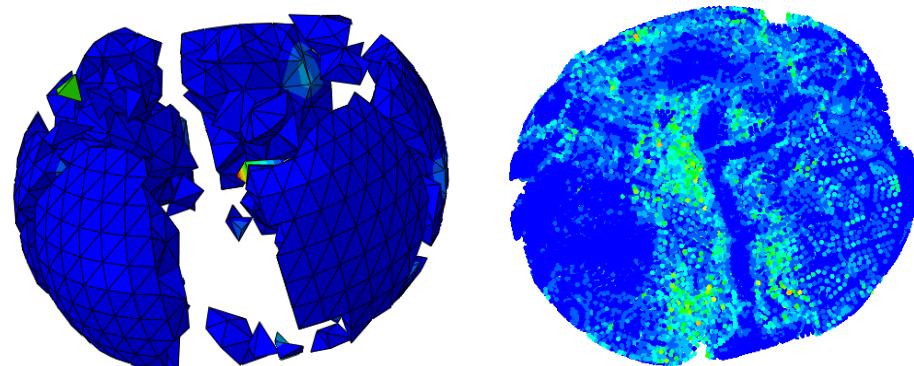
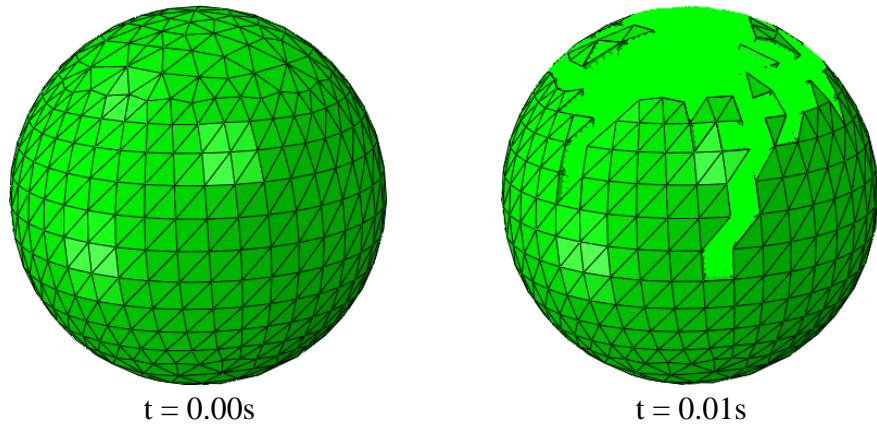


Fig. 10.16: The two element types of the concrete sphere: left) Continuous 3D elements (C3D4); right) SPH particles (PC3D)

10.5 Results interpretation

As the Fig. 10.13, Fig. 10.14 and Fig. 10.15 depict, the radial (horizontal) displacements recorded by using the SPH method are larger than the ones obtained using classic FEM approach. This brings us closer to crushing material's behaviour, such as rock or concrete, where certain parts, under substantial loads scatter parts of the initial matrix. Such behaviour can only be achieved by using meshless techniques, as SPH, where the particles resulted from attaining the strain. Another important feature of the technique is that the particles not only leave the initial geometrical position, they behave as individual material points.

If we take into consideration the results presented in Fig. 10.16 and Fig. 10.17, we can observe that, starting from the outside towards the inside, as the development of the lateral strains due to the tension stresses to which the concrete is subjected reach the imposed threshold. Moreover, the larger number of elements transform from continuous to meshless elements, the larger the allowed displacements of the



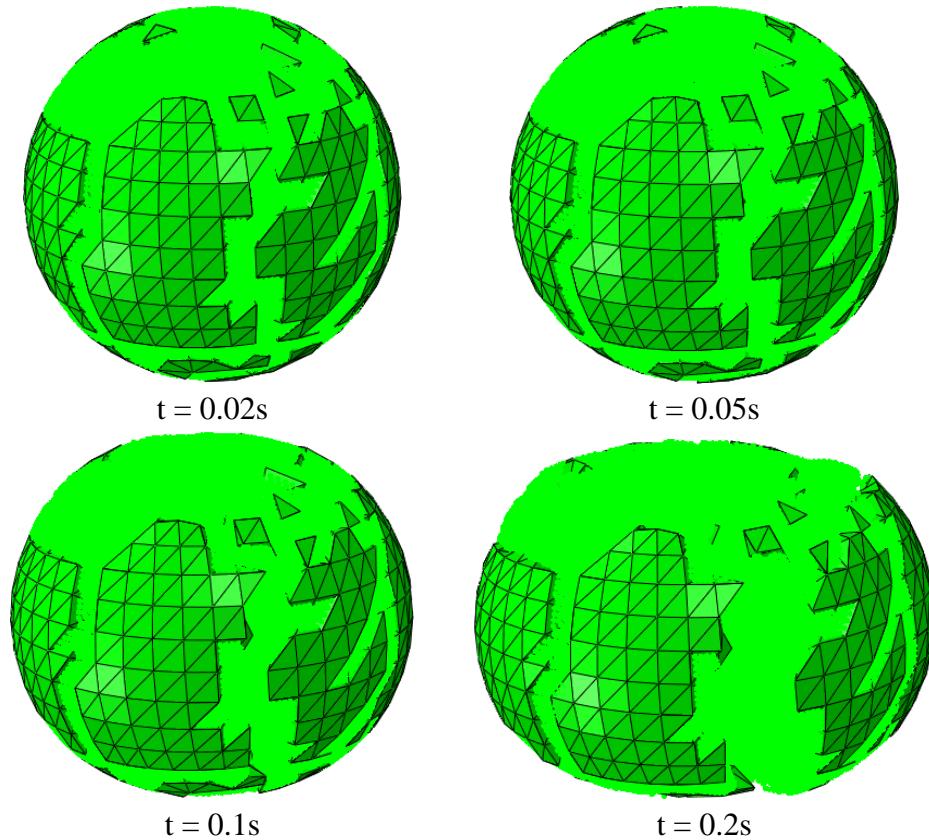


Fig. 10.17: The evolution of the conversion of elements from C3D4 type to PC3D

Taking into consideration the total energy variation of the model, depicted in Fig. 10.18, one can observe the fact that following an initial cracking stage, which leads to a drop of the material's resistance, a partial hardening of the entire sample may be observed. The ultimate stage, from around 0.10 to 0.20s, is based on a residual resistance of the material – SPH generated particles did not have enough space and time to leave entirely their position and the cluster formed inside offered still some resistance.

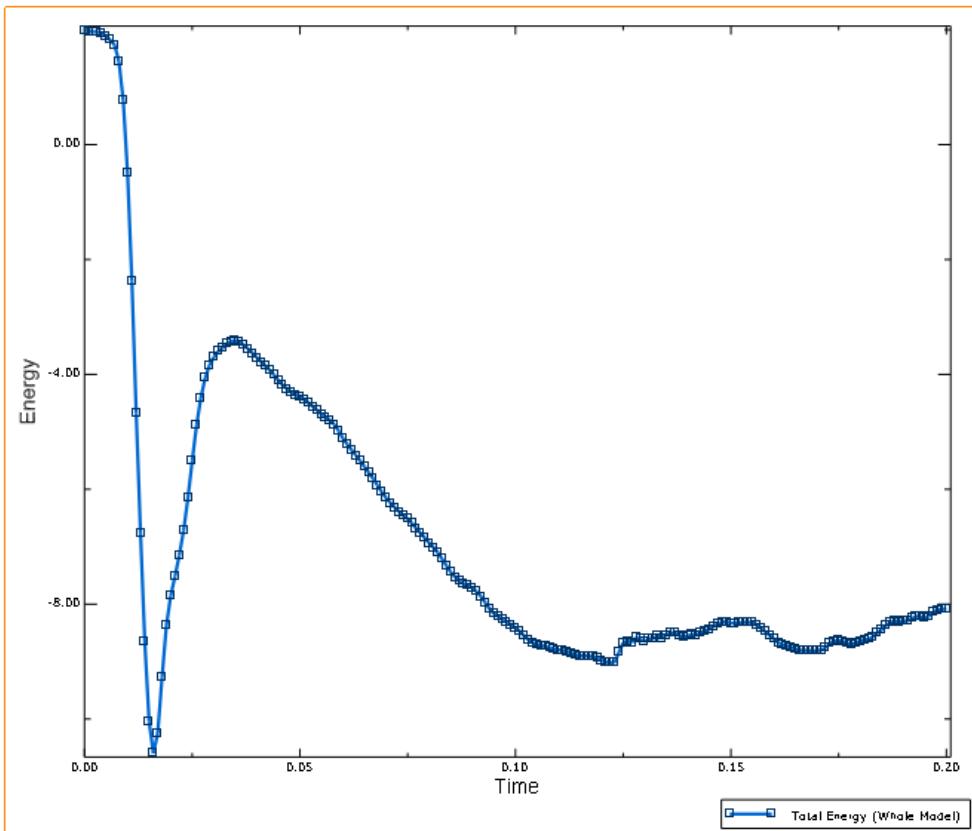


Fig. 10.18: The total consumed energy of the whole model

Although it is an advanced method, the SPH technique, combined with the complexity of a classical FEM model, such as the one presented hereby, may lead to the impossibility of calculating the output of the considered assembly, unless further advancements regarding the computational power available.

10.6 Things to remember

This problem exemplified the usage of a crushable material law along with the integration of meshless techniques in FEM. Adding to that, infinitely rigid parts have been created and a unique, general interaction has been assigned to the entire model, under a dynamic explicit calculation stage.

11. TIME - DEPENDENT SEISMIC LOADING OF A SYSTEM WITH CONTINUOUS ELEMENTS

11.1 Keywords

Dynamic analysis, seismic loading, point wise history output, mechanical coupling

11.2 Aims

- Construct and use a time dependent varying load/boundary condition
- Create a dynamic analysis on an inverted pendulum
- Obtain the variation history on interest points across the height of the pendulum

11.3 Problem description

The problem considers the seismic input that will subject an inverted pendulum, whose geometry is displayed in Fig. 11.1. There are two aspects that this problem is treating: the first, is to determine the natural frequency and eigen period of the system, and the second to study the assembly's response to a given ground acceleration variation.

There are two stages that the problem takes into consideration: the first, of 1.5(s) duration is the one with the applied excitation, and during the second, of 45(s), the damping of the system down to obtaining an overall equilibrium.

The seismic action is to be given as a sinusoidal function of ground acceleration, acting on the 0Y axis, having a maximum amplitude of 0.2g and a wave length of 3.14162s (approximately π).

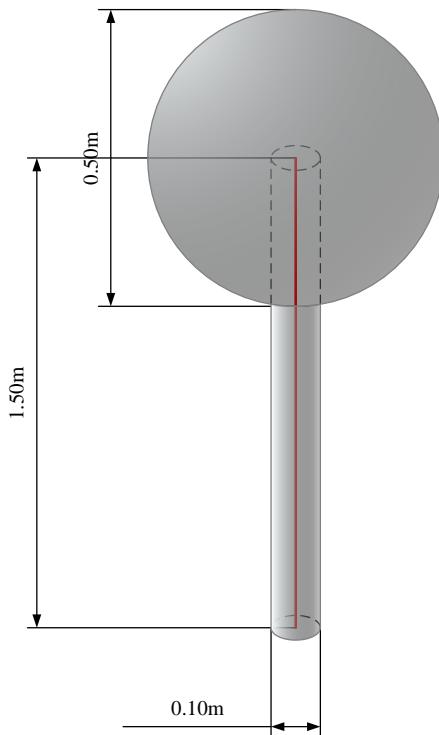


Fig. 11.1: The considered model's geometry

The considered time dependent function is: $a(t) = 0.3g * \sin(\frac{\pi}{2} * t)$, where g is the gravitational acceleration, usually considered as $9.81(m/s^2)$, $0.3 \cdot g$ is the maximum amplitude of the function and $\pi/2(rad/s)$ is the angular velocity (see Fig. 11.2).

At the base, the system is encastred, in order to assume the hypothesis of a theoretical one dynamic degree of freedom inverted pendulum. The problem will use a mechanical coupling between the pole, which is represented by a beam (one dimensional element), and a solid three dimensional continuous element – the mass.

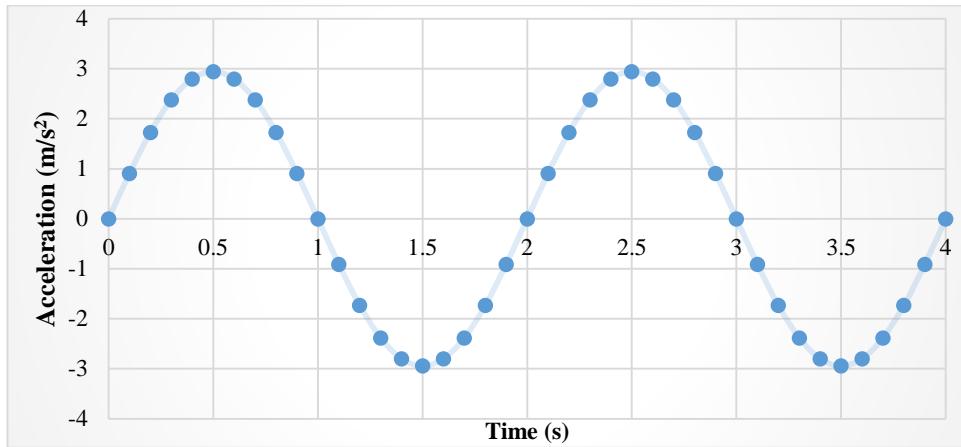


Fig. 11.2: The time dependent acceleration function

11.4 Solving steps

First of all, the user has to create the geometry. Therefore, either using a CAD software, either the **Module: Sketch**, create the two components of the model, as previously described (see Fig. 11.1): a sphere having a radius of 0.25(m) and a bar (line) of 1.50(m) length. If the case of using a third party software, import the two parts to the model by using the **File ▶ Import ▶ Part....**

The pole, as well as the top mass will be imported as **③3D** and Type **④Deformable**, the first being saved as IGES file, while the second as ACIS.

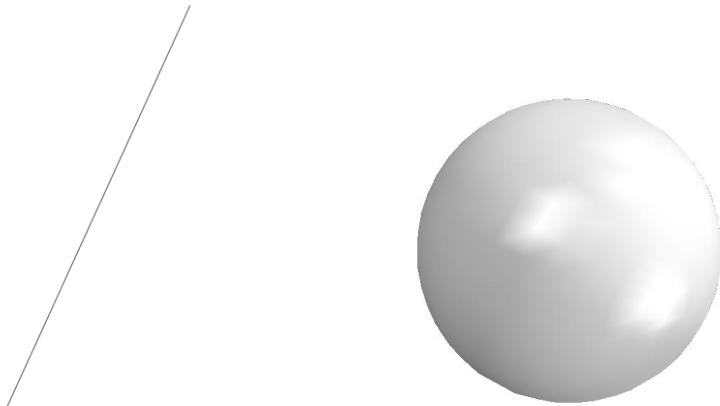


Fig. 11.3: The two components of the model: the pole and the mass

Moving to the **Module: Property**, we have to create the material – a damping steel. Go to SR **Create material** button, and input the following properties:

- **General:** **Density** – 7.85(to/m^3);
- **Mechanical:** **Elasticity** ► **Elastic** – Young Modulus of 210E6(kPa) and Poisson's Ratio of 0.3(-);
- **Mechanical:** **Damping** – alpha of 0.20; beta of 0; composite – 0; structural – 0.

This implies that only a mass dissipation will be provided, as alpha is the only term different than 0.

We move to the creation of the section to applied to the pole. In order to create the circular section, click **LMB** on the SR **Create Section** button and select **Beam**, type **Beam**. Click **LMB** on the **Create Beam Profile** button and select under the **Create Profile** window the **Circular** type. Click the **Continue ...** button and in the **r:** window, insert the value of 0.05m (50mm).

Automatically, the name of the created profile will appear under the **Profile name** selection pop-down menu. Also, fill in the **Section Poisson's ratio** with

a value of 0.3 (equal to the one used for the material creation). Finish the instance, click LMB the **OK** button.

Assign the section the pole, by using the **SR**  **Assign Section** button. In order to end the assignment, go to **RS**  **Assign Beam Orientation** button and select the part, in our case the pole, and when asked to provide the vector parallel to the longitudinal axis, allow the default value of (0,0,0) for the beginning and regarding the tip of the vector, input the (0,0,1) direction. In the end, click the the **OK** button.

In order to confirm the validity of the assignment, click **View** **Part Display Options...**, and in the **Part Display Options** window, check Render beam profiles. After selecting the **OK** button, the previously selected beam is drawn at its full thickness, provided by the considered profile (in this case, a circle).



Fig. 11.4: The rendered pole section

Regarding the upper mass, a **◎Solid**, type Homogeneous section is to be created, and later assigned to the sphere part.

In **Module:** **Assembly**, import all the parts to the instance by using **RS**  **Instance Part** button, check **◎Independent** (mesh on instance) and while

holding the **Shift** button, select the two parts.. Finish the procedure by clicking the **OK** button.

In order to select the middle of the sphere, and later to create a quad mesh based on its geometry, cut the part in eight smaller parts, using three orthogonal planes, using the **SR**  **Partition Cell: Define Cutting Plane**, and the Point&Normal and 3 Points techniques. In the end, the assembly should look as displayed in Fig. 11.5.



Fig. 11.5: The assembly's geometry

Go to **Tools** ► **Set** ► **Create...**, name it “Interest points”, click **Continue ...**, and when prompted for the geometry of the set, choose the top and bottom points of the mass, and the bottom point of the pole.

In the **Module: Step**, click the **SR**  **Create Step** button, and name the calculation step “Vibration”. The default Procedure type is General and the type of calculation step is Static, General. Change to Dynamic, Explicit and

click the **Continue ...** button, and a new window will appear. Input the value of 1.5(s) at the Time period, and allow an Automatic time stepping. Click **OK** to end the sequence. The second step, named “Free vibration” is 45(s) long and, under the Incrementation tab, check the Automatic Type, Element-by-element Stable increment estimator, and a Maximum time increment of Value 0.2 (Fig. 11.6).

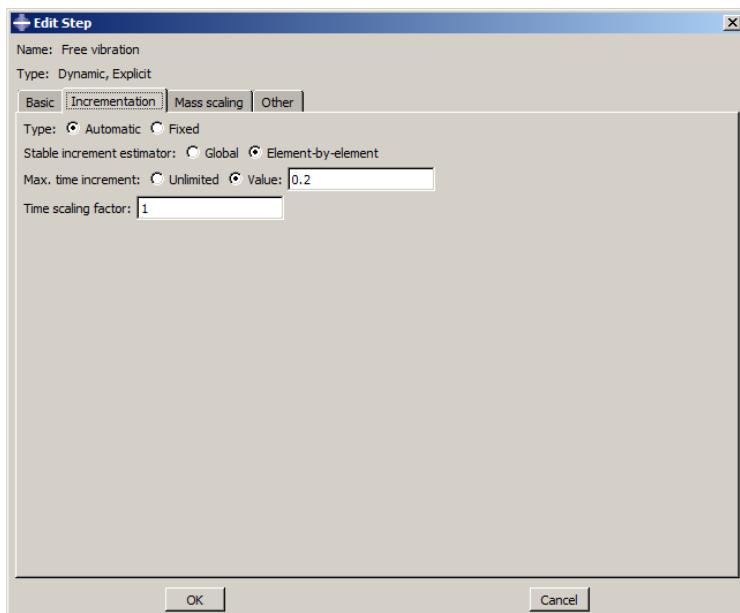


Fig. 11.6: The Edit Step dialogue window of the "Free vibration" step

Now, go to **Output ▶ History Output Requests ▶ Manager...** and click the **Edit ...** button. For the domain, select Set, and choose the set previously created “Interest points”. The frequency will remain default at every 1 increment. In this case it is of interest to monitor the displacement, velocity and acceleration variation for each point.. Therefore, check Displacement/Velocity/Acceleration of the possible output variables. Click the **OK** button in order to save the modifications.

Under the **Module: Interactions**, in order to connect the two entities, create a constraint that will link the mass to the top part of the pole using the **SR**  **Create Constraint** button, and choose Coupling. For the Control points, select the top of the pole, while for the Surface (Node Region) select the sphere. In the end click the **OK** button.



Fig. 11.7: The connected sphere's regions to the pole's top point

Another constraint must be taken into account: a tie between the sphere and the pole. Again, for the master surface, disable from the view the upper mass, and select the point at the top of the pole, by using a master type Node Region. Again, for the slave part, select Node Region and select the sphere. For the Position Tolerance let the check on Use computed default, and Adjust slave surface initial position, and Tie rotational DOFs if applicable, respectively.

At this moment, the two assembly components are tied together, transferring the deformations of the top of the pole to the centre of the mass.

In order to create the load that will impose the vibration on the inverted pendulum, advance to **Module: Load**, where using the **SR**  **Create Boundary Condition**, impose the encastre in the base of the pole. In order to do this, select the Symmetry/Antisymmetry/Encastre option, followed by the bottom point of the line. In the end, complete the sequence by clicking the **OK** button.

In order to impose the load, go to **SR**  **Create Load** button, click it, and the Create Load window will appear. Name the load “Vibration”, make sure the set step is “Imposed vibration” and choose Mechanical for the load category and Gravity for the Types for Selected Step. Click **Continue ...** and a second window (Edit Load) appears in which it is required to provide the domain on which the gravity load should be applied and on the three directions its components. Therefore, as it can be observed, the region is the whole model, by default, and in the field near the Component 2 (0Y axis), fill with the 1(-) value. This will imply that 100% of the declared amplitude time dependent function will be applied.

To create the varying amplitude function, in the Edit Load dialog window, go to  **Create Amplitude** button, name it “Variation”, check Tabular and click **Continue ...**. In the Edit Amplitude window (see Fig. 11.8), insert by directly pasting the values presented in Tab. 11.1.

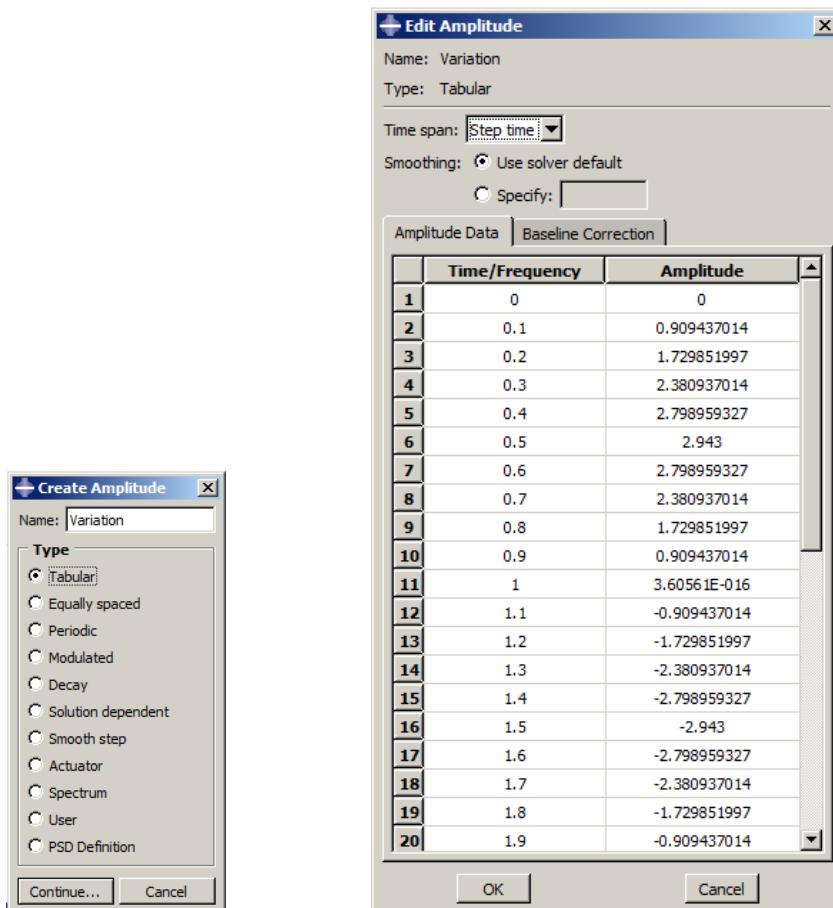


Fig. 11.8: Creating an Amplitude function: left) Choosing the amplitude type to create; right) Edit Amplitude dialog window

In the end, make sure that for the Amplitude option, the “Variation” property is selected and, in the Load Manager window, click on the load during the “Free vibration” step and deactivate it (Fig. 11.9).

Tab. 11.1: Amplitude as function of time variation

Time/Frequency	Amplitude	Time/Frequency	Amplitude
0	0	2.1	0.909437014
0.1	0.909437014	2.2	1.729851997
0.2	1.729851997	2.3	2.380937014
0.3	2.380937014	2.4	2.798959327
0.4	2.798959327	2.5	2.943
0.5	2.943	2.6	2.798959327
0.6	2.798959327	2.7	2.380937014
0.7	2.380937014	2.8	1.729851997
0.8	1.729851997	2.9	0.909437014
0.9	0.909437014	3	1.08E-15
1	3.61E-16	3.1	-0.909437014
1.1	-0.909437014	3.2	-1.729851997
1.2	-1.729851997	3.3	-2.380937014
1.3	-2.380937014	3.4	-2.798959327
1.4	-2.798959327	3.5	-2.943
1.5	-2.943	3.6	-2.798959327
1.6	-2.798959327	3.7	-2.380937014
1.7	-2.380937014	3.8	-1.729851997
1.8	-1.729851997	3.9	-0.909437014
1.9	-0.909437014	4	-1.44E-15
2	-7.21E-16		

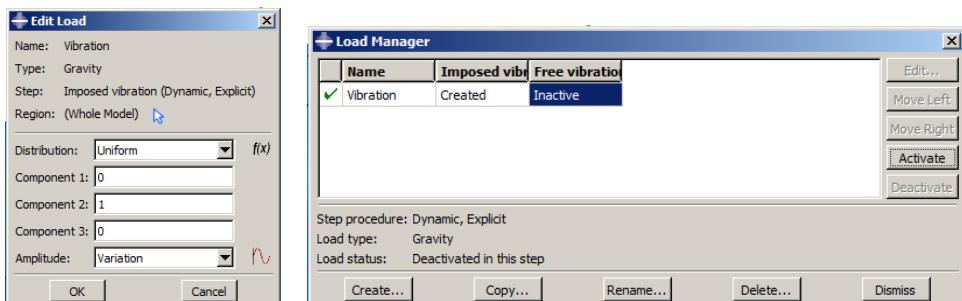


Fig. 11.9: The vibration load options: left) applying the Amplitude function; right) disabling the load during the second step



Fig. 11.10: The assembly submitted to the external loads and the boundary conditions

Before submitting the model to calculation, mesh is using a maximum element size of 0.1(m) or finer, depending on your computational power. In order to create the discretization, use firstly **SR** **Seed Part Instance**, followed by **SR** **Mesh Part Instance**. The meshed assembly should be as Fig. 11.11 displays.

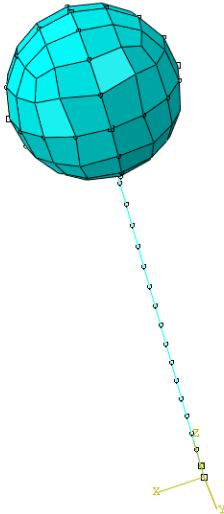


Fig. 11.11: The meshed assembly

In order to conclude the model constructing step, go to **Module: Job**. Go to **SR** **Create Job**. Change the name to “Dynamic”. The source is the Model-1, which has been created. Click **Continue ...**. Under the Edit Job window, go to the Parallelization tab and, if the case, select Use multiple processors and change to the number of cores your CPU have, in order to improve (reduce) the calculation period.

11.5 Results interpretation

In order to check that the equilibrium state of the excited top mass has been obtained, go to **SR** Common Options and under the Basic Tab, at the Deformation Scale Factor, check Uniform and input the 1000 value. This will make a 1mm displacement be displayed equal to a initial 1m displayed, if we consider the default Auto-compute one unit factor. Moreover, switch on the **SR** **Allow Multiple Plot States** button and select both the **Plot Undeformed Shape** and **Plot Deformed Shape** buttons. The overall view should resemble the one presented in Fig. 11.12.

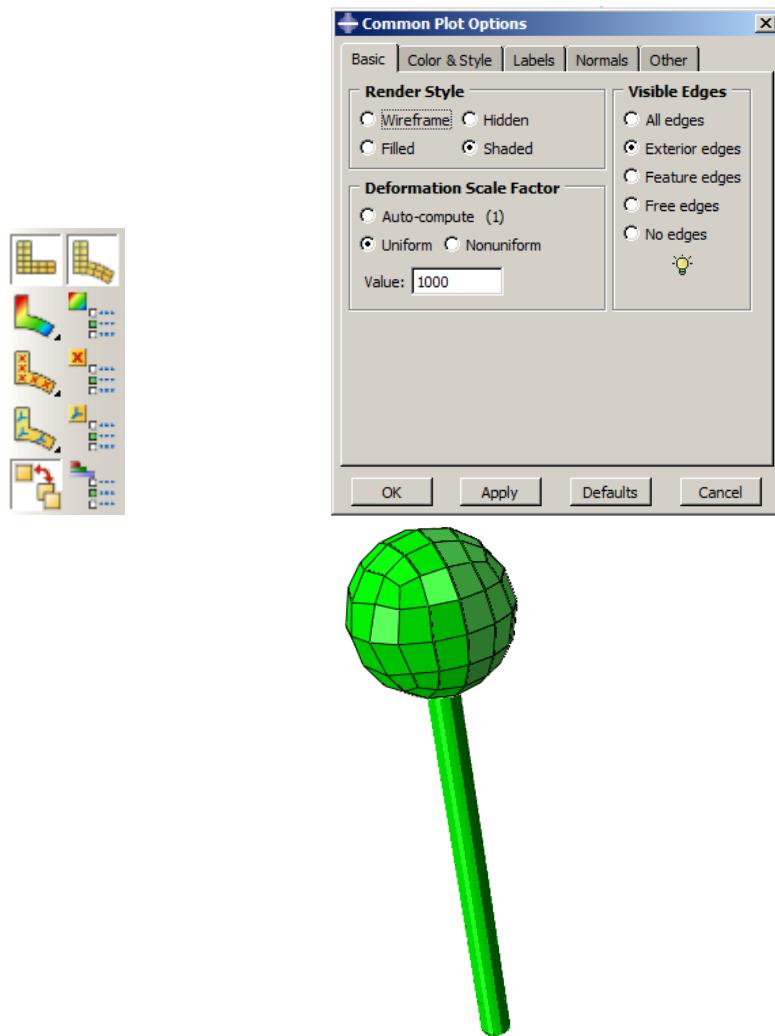


Fig. 11.12: Graphically displaying the difference between the initial equilibrium position and the one obtained: top) options to be checked; bottom) the final figure

A second option, based upon displacement values, may be obtained by going to **Result ▶ History Output...**, and in the dialog window, check The three points' U2 (0Y) displacement history.

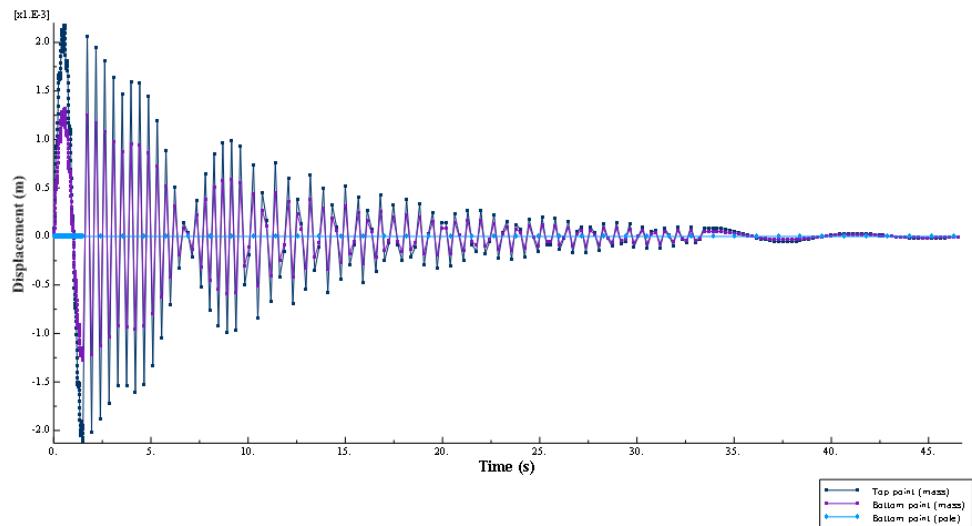


Fig. 11.13: The displacement variation for the three interest points

As Fig. 11.13 displays, an overall equilibrium has been obtained, this being easily verified against the bottom pole point, which is encastred and whose displacement is always null. The data can be extracted to be used with different processing software, by going along the **Report ► XY....**

11.6 Results interpretation

Following the model construction and its calculation, the following may be observed and emphasized:

- during the forced vibration phase, both negative and positive maximum amplitudes have been achieved, disregarding the damping of the material;
- the material's damping has been acting exponentially on the amplitudes of the displacement, velocity and acceleration, down to an almost complete attenuation of the wave;
- the fact that the velocity is the first derivative of the displacement and the primitive of the acceleration – one's peak is the other's null (see Fig. 11.14, Fig. 11.15 and Fig. 11.16).

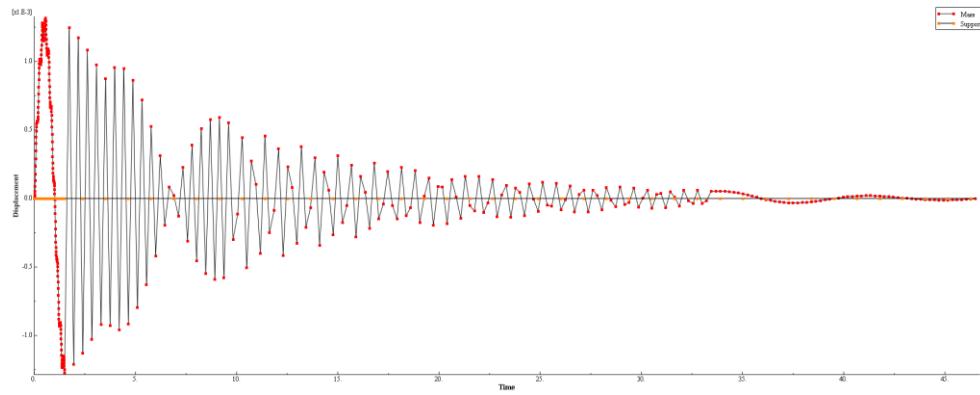


Fig. 11.14: Displacement variation along the excitation direction

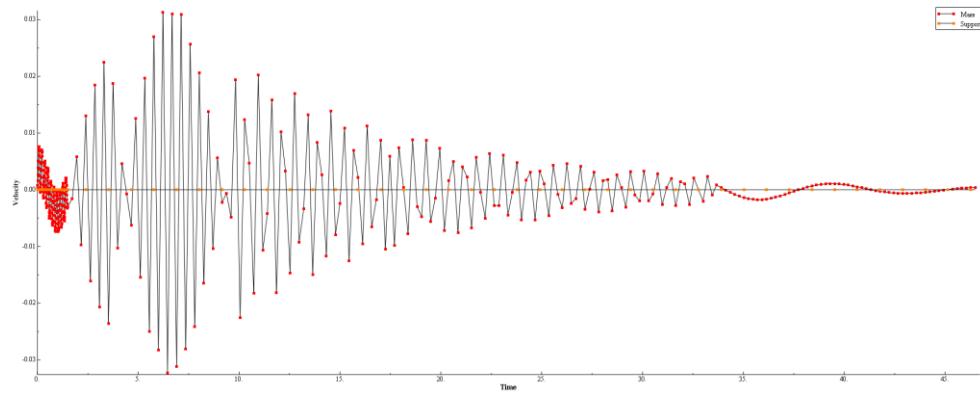


Fig. 11.15: Velocity variation along the excitation direction

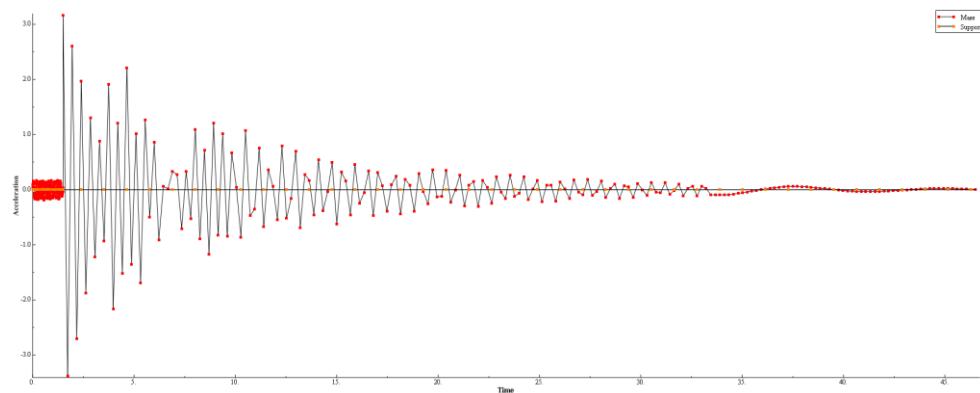


Fig. 11.16: Acceleration along the excitation direction

12. EQUIVALENT LIQUID – SOLID (EULER-LAGRANGE) COUPLING

12.1 Keywords

Euler-Lagrange Coupling, Volume fraction tool, Predefined Fields;

12.2 Aims

- Modelling the interaction between a solid and a fluid part in the same analysis;
- Proving a method of determining the behaviour of high displacements elements;

12.3 Problem description

The problem consists of a water volume moving under gravitational loads, modelled using the Euler formulation, interacting with a solid cube made of concrete, modelled using the Lagrange formulation.

The whole model will be considered to have 1 m thickness. In order to model the water volume, two parts will be used (the reference part having 5x9 m and the medium part having 15x10 m) and for the solid concrete cube a single part (1x1 m), the position being presented in Fig. 12.1. The medium part has to be chosen such that it covers all the possible or intended movements of the water volume, this part representing the geometrical place where the water is allowed to exist. The expected contact between the Eulerian and Lagrangian

parts must be within the boundaries of the Eulerian medium. The Lagrangian part and Eulerian medium can overlap, because the medium represents all the possible positions of the Eulerian volume, not the necessary presence of material. The Eulerian reference (the initial position of the material in the medium) cannot overlap the Lagrangian part.

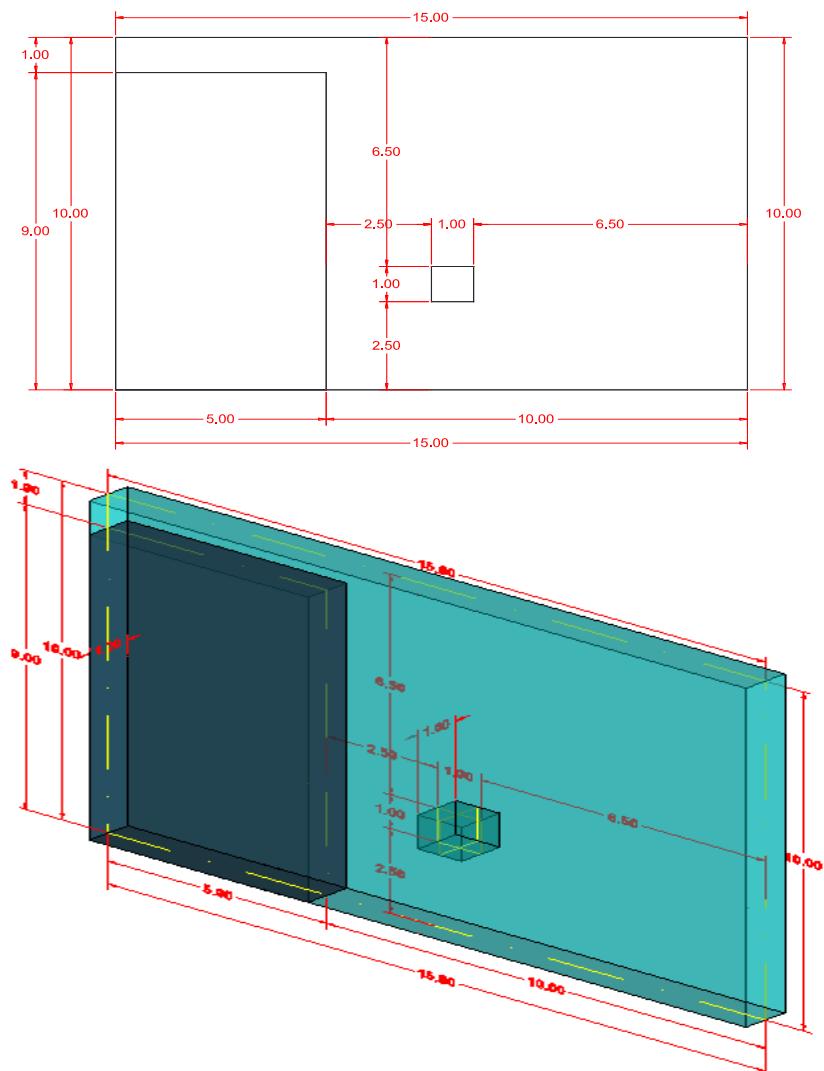


Fig. 12.1: The parts of the model and their positioning

The solid concrete cube will be modelled to have an elastic behaviour, the parameters necessary being the density $\rho = 2.5 \text{ g/cm}^3$, the Young's modulus $E = 2.7 \times 10^7 \text{ kPa}$ and Poisson's coefficient $\nu = 0.25$.

The material used to model the behaviour of water volume uses the following parameters: the density equal to $\rho = 1 \text{ g/cm}^3$, the bulk viscosity $\eta = 1 \times 10^{-5} \text{ kN s/m}^2$ and the parameters of the Hugoniot linear form ($U_s - U_p$) of the Mie-Grüneisen equation of state ($c_0 = 1450 \text{ m/s}$, $s = 0$, $\Gamma_0 = 0$).

A dynamic explicit step will be used, the defined period being 3 s.

For the Lagrangian part, an encastre boundary condition is applied on the bottom of cube (normal to z-axis).

In the case of the Euler part, the boundary conditions are applied to the Euler medium, considering zero normal velocities on all the faces.

A gravity load is applied on the whole model, considering an acceleration of -9.81 m/s^2 on z-axis.

The Eulerian mesh used as a medium through which the Euler material can flow is made out of EC3D8R elements (Element Type – Eulerian), Hex structured mesh control, with cubes having 0.5 m edges. The reference part does not necessarily need to be meshed, but the usage of it improves the results of the procedure performed in order to identify the presence of material in the Eulerian medium.

The solid part is meshed using C3D8R (Element Type – 3D Stress) elements, Hex structured mesh control, using cubes having the 0.25 m edges.

12.4 Solving steps

The geometry presented in Fig. 12.1 is recommended to be drawn in a CAD software (the whole assembly should be positioned in the point of choice in 0, 0, 0 coordinates to facilitate the model manipulation in Abaqus) and each part that will be used in the model shall be exported with “.sat” extension. The parts will be imported one at a time.

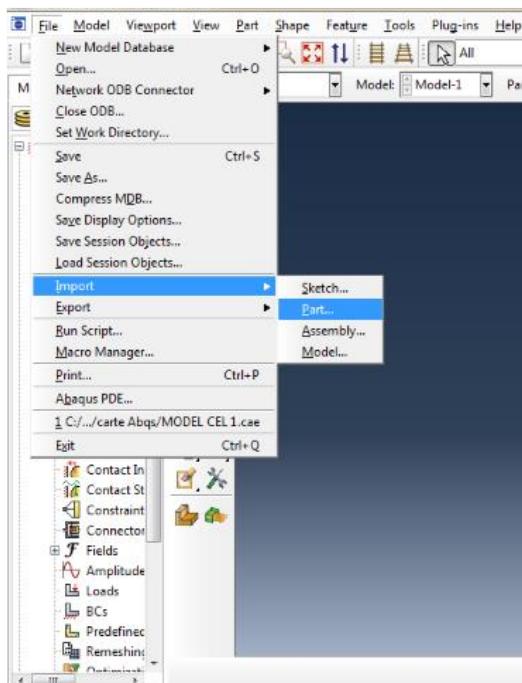


Fig. 12.2: The import procedure

In order to import the solid part, the following command sequence is to be followed **File** ► **Import** ► **Part...** (Fig. 12.2) and select from the work folder the file containing the geometry. A “**Create part**” window will be opened (Fig. 12.3), having 3 sub-menus. In the sub-menu **Name-Repair**, the text box **Name: Part name:** will be filled default with the file name of the exported part selected for import. This name can be changed arbitrary. Also, the radio buttons **Part Filter:** **Import all parts** and **Create individual parts** should be checked by default. In the sub-menu **Part Attributes**, the radio

buttons **Modeling Space:** **3D** and **Type:** **Deformable**. To import the part with the above properties, click **LMB** on the **OK** button.

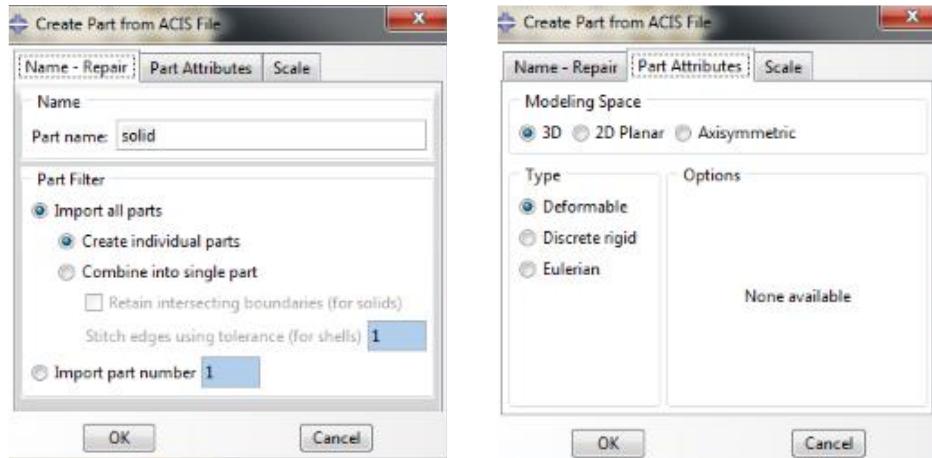


Fig. 12.3: The “create part” window for the import of the solid part

Like in the case of the solid part import, the same sequence should be used (**File** ► **Import** ► **Part...** (Fig. 12.2)), selecting for each of the 2 Eulerian parts the specific file from the work folder. A “**Create Part**” will appear for each of the two parts (Fig. 12.4). The text box **Name: Part name:** will be filled by default with the name of the parts imported. For importing Eulerian parts, the following radio buttons should be checked: **Part Filter:** **Import all parts** and **Create individual parts** in the **Name – Repair** sub-menu and **Modeling Space:** **3D** and **Type:** **Eulerian** in the **Part Attributes** sub-menu. To import each of the 2 parts with the selected properties, click **LMB** on the **OK** button.

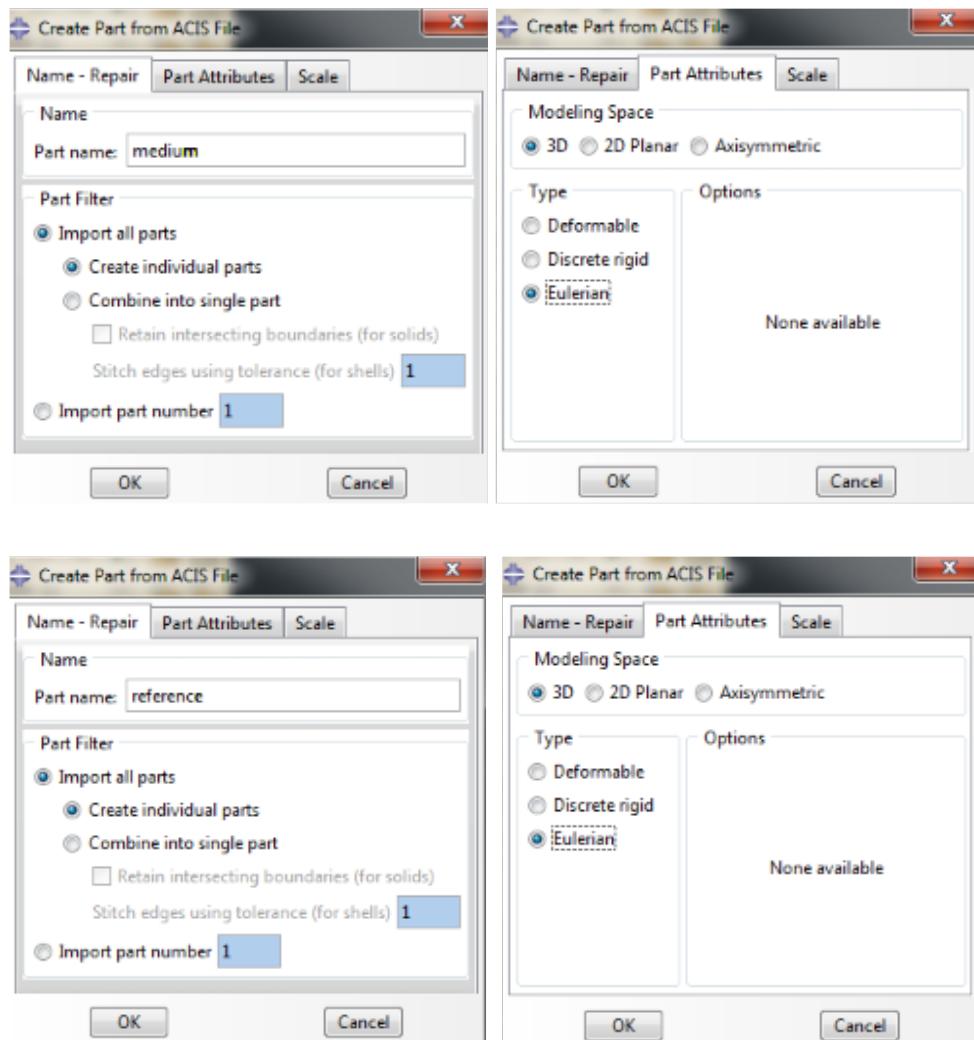


Fig. 12.4: The “create part” windows for the import of the water parts

The two materials needed for the analysis will be created separately, one at a time. To create a material, select **SR**  **Create material** button, using the **LMB**. An “Edit Material” window will appear (Fig. 12.5).

ABAQUS for Geotechnical Engineers

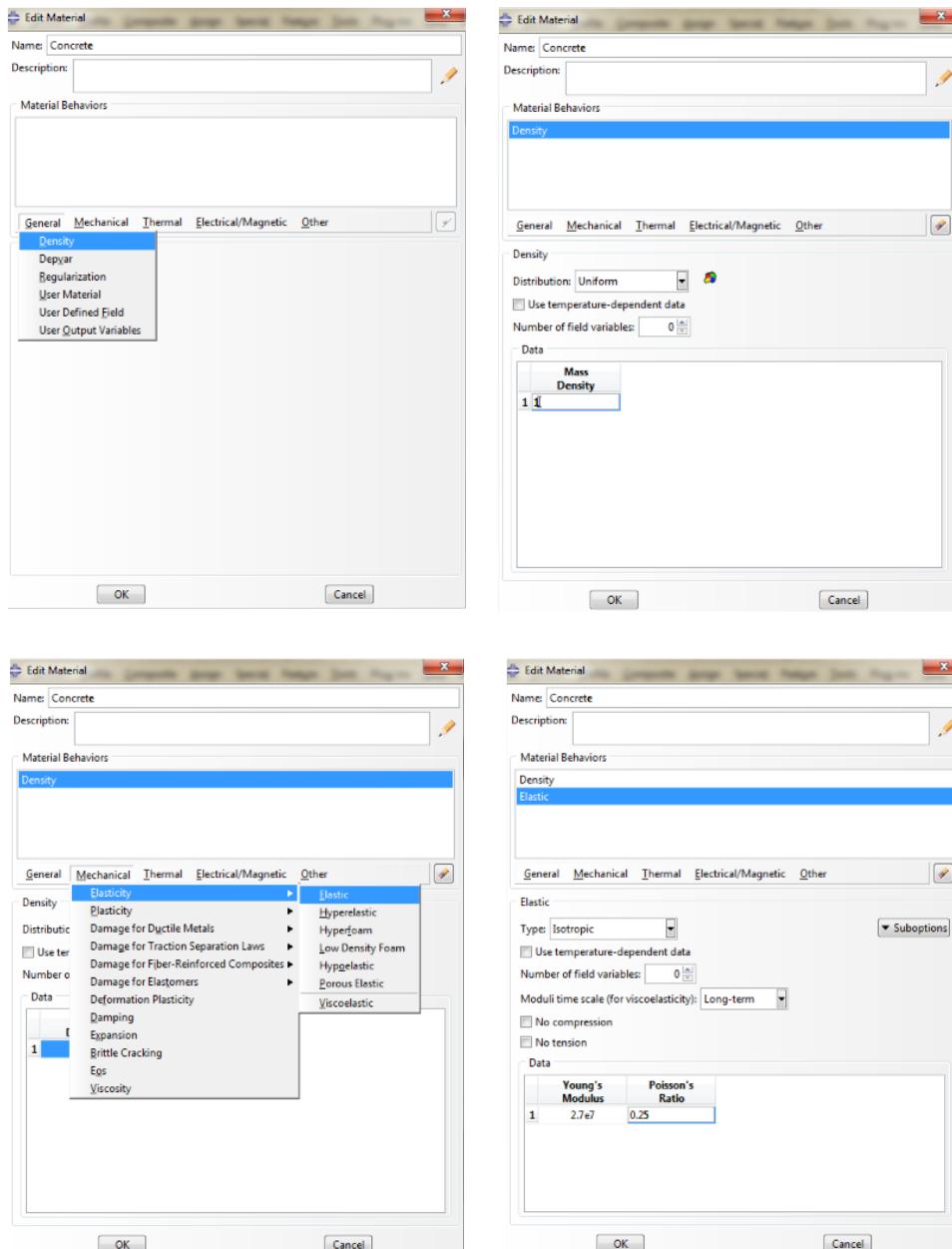


Fig. 12.5: The solid material (concrete) definition

To define the density of the solid part, the path is **General: Density** and in the **Mass Density** textbox insert the density, in this case 2.5 g/cm^3 . For the

elastic behaviour definition, the path is **Mechanical: ▶ Elasticity ► Elastic**, and in the two textboxes (**Young's Modulus** and **Poisson's Ratio**) the value specified for the concrete in the material section shall be written. To create the material with the above properties, click **LMB** on the **OK** button.

To create the material for fluid part, select again **SR**  **Create material** button, using the **LMB**. The density is defined the same as for the solid part case. To declare the equation of state, the path is **Mechanical: ▶ EOS** (Fig. 12.6), for this case (using the Hugoniot U_s - U_p linear form of the Mie – Grüneisen equation of state) **Type: ▶ U_s - U_p** and use the values of the EOS given for the material (c_0 , s , Γ_0).

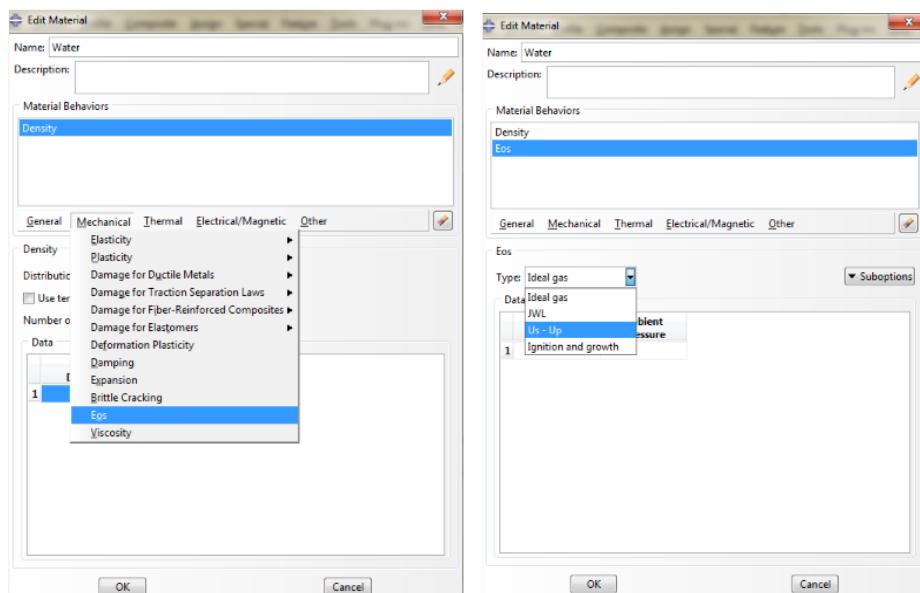


Fig. 12.6: The EOS definition for water material

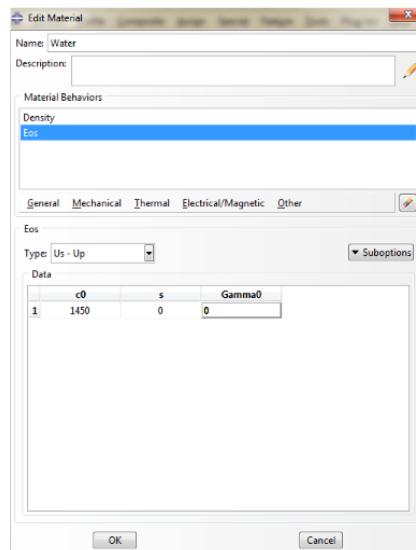


Fig. 12.6: The EOS definition for water material (continued from the previous page)

To define the viscosity for the fluid material, the path is **Mechanical**: **Viscosity** and in the textbox **Dynamic Viscosity** fill the value for the material (Fig. 12.7).

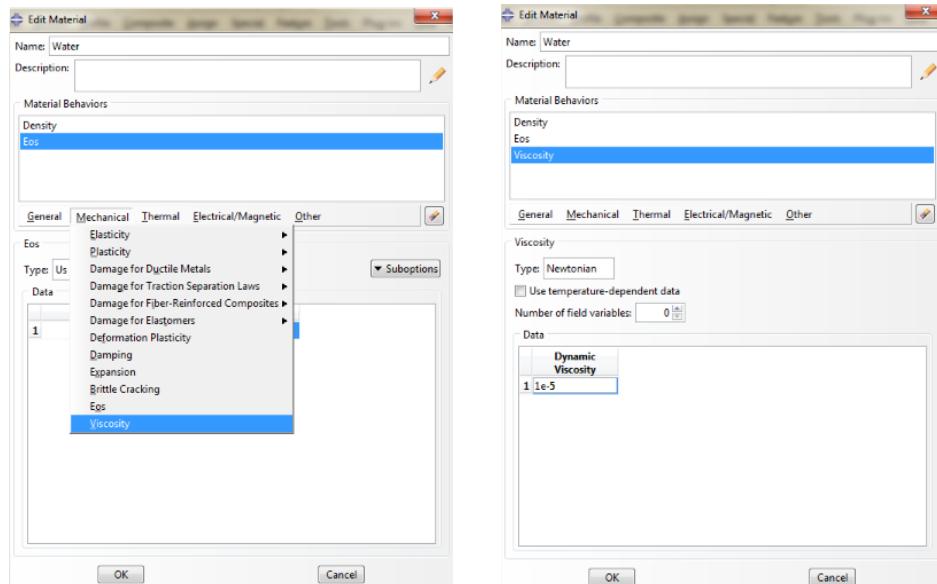


Fig. 12.7: The definition of the viscosity for the water material

The next step will be the creation of the section for the concrete cube and for the water volume, using the SR  **Create Section** button. For the solid section, select **Category**  **Solid**, **Type:** **Homogeneous**. In the **Edit Section** window, select the material previously created with **Material:**  (in this case) Concrete and create the section by clicking **LMB** on **OK** button (Fig. 12.8).

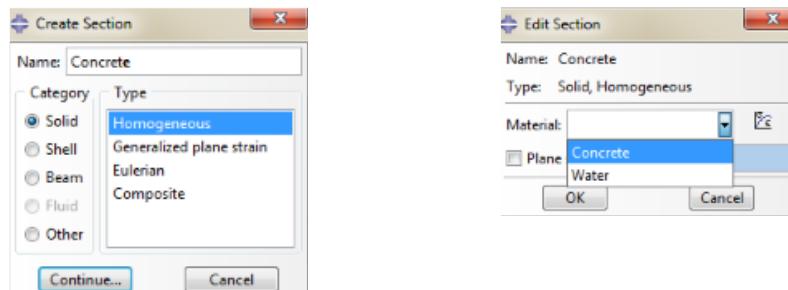


Fig. 12.8: The section creation for the solid

For the liquid section, select **Category**  **Solid**, **Type:** **Homogeneous**. In the **Edit Section** window, select the material previously created with **Base Material:**  (in this case) Water and create the section by clicking **LMB** on **OK** button (Fig. 12.9).

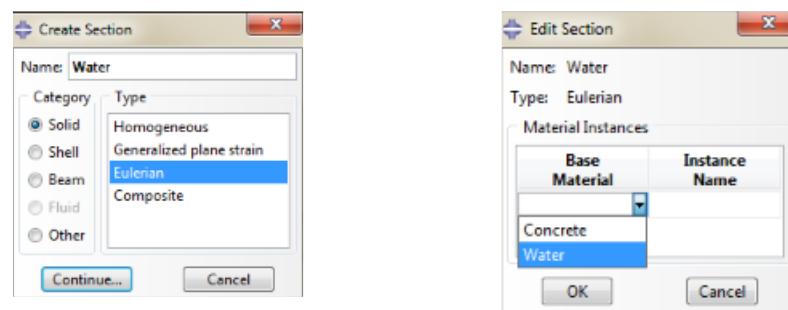


Fig. 12.9: The section creation for the liquid

Assigning the sections to the parts is done using the SR  **Assign Section** button, on which click the **LMB**. In the case of the part for the solid cube, the user is asked to select the part. Click **LMB** on the solid part and click **BR**

Done. The window that appears will only show the sections that can be assigned to solids (that is why, in this case, only the “Concrete” section appears). Click **LMB** on **OK** button to assign the section to the part. In the case of the fluid part, the user will not be asked to select the part (which has been declared as an Eulerian part from the beginning), instead can choose an Eulerian section previously created for that part. Click **LMB** on **OK** button to assign the section to the part. (Note: The section assignment should be done for the medium as well as for the reference part).

Moving to the **Module: Assembly**, in order to insert the three parts into the whole system, **LMB** click the **SR**  **Instance Part** button, check **Instance Type** **Independent (mesh on instance)** and while holding the **Shift** button, click **LMB** on the three parts in order to bring them to the same instance. Finish the procedure by clicking the **OK** button.

In the **Module: Step**, click on the **RS**  **Create Step** button, the name of the step will be default “Step 1” (Fig. 12.10). The default Procedure type is General and the type of calculation step is Dynamic, Explicit. Click the **Continue ...** button, and a new window will appear. In the description field, write “Loading step”. The time period will be set to 3 (s). Click the **OK** button in order to complete the creation of the calculation step.

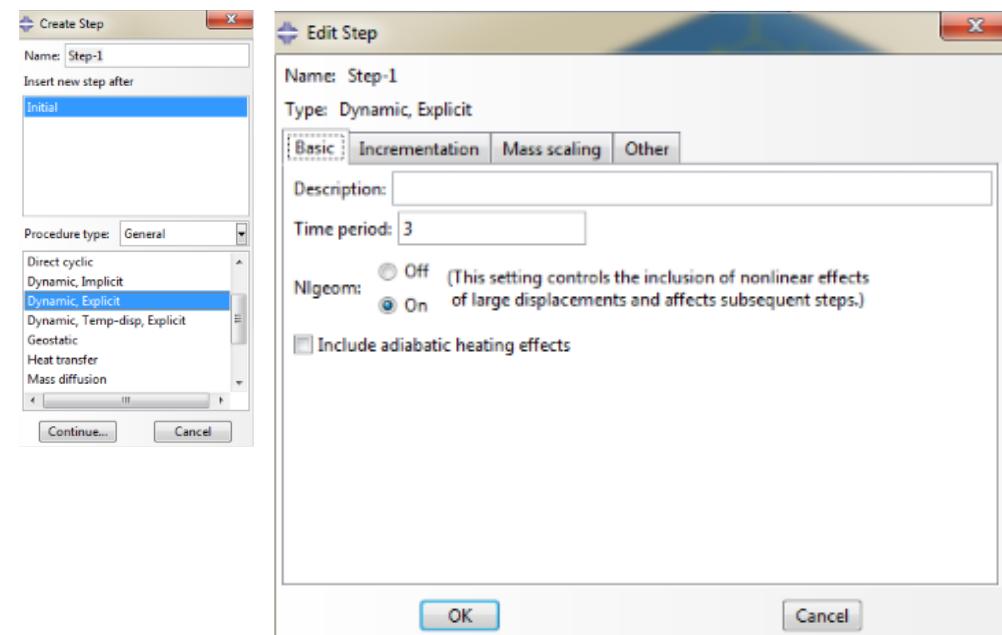


Fig. 12.10: Create step procedure for the analysis

In the **Module: Interaction**, click LMB on SR **Create Interaction Property** in order to set the parameters of the interaction. This will open a **Create interaction property** window (Fig. 12.11) where the user chooses the type of interaction, in this case **Type: Contact**. Click the **Continue ...** button to confirm the type.

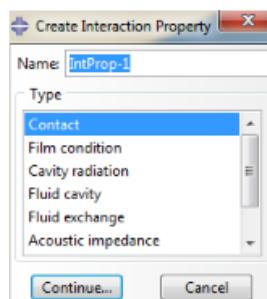


Fig. 12.11: Create interaction property window

The **Edit Contact Property** window will open and the user can select the contact properties. In this case, the tangential behaviour –frictionless (with the path **Mechanical: Tangential behaviour, Friction formulation: Frictionless**) and normal behaviour – hard contact (with the path **Mechanical: Normal behaviour, Pressure – Overclosure: "Hard" Contact** – make sure to check the Allow separation after contact box) will be used (Fig. 12.12).

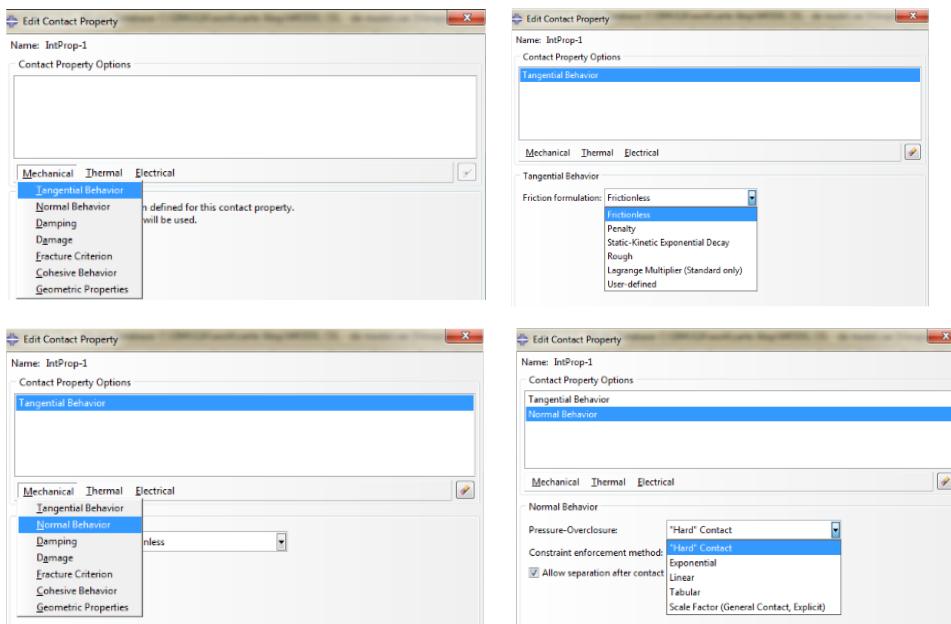


Fig. 12.12: The choosing of the interaction properties

In the **Module: Interaction**, click LMB on SR **Create Interaction** to define the type of interaction wanted (Fig. 12.13). The interactions are selected for a particular step, the options available depending on the type of step chosen previously. In this case, the choice is **Types for Selected Step: General contact (Explicit)**. This will open the **Edit Interaction** window and the options for this model are **Contact Domain: Included surface pairs: ⊖ All* with self** and **Global property assignment: "name of the interaction**

property defined previously. Click the **OK** button in to accept the properties of the interaction.

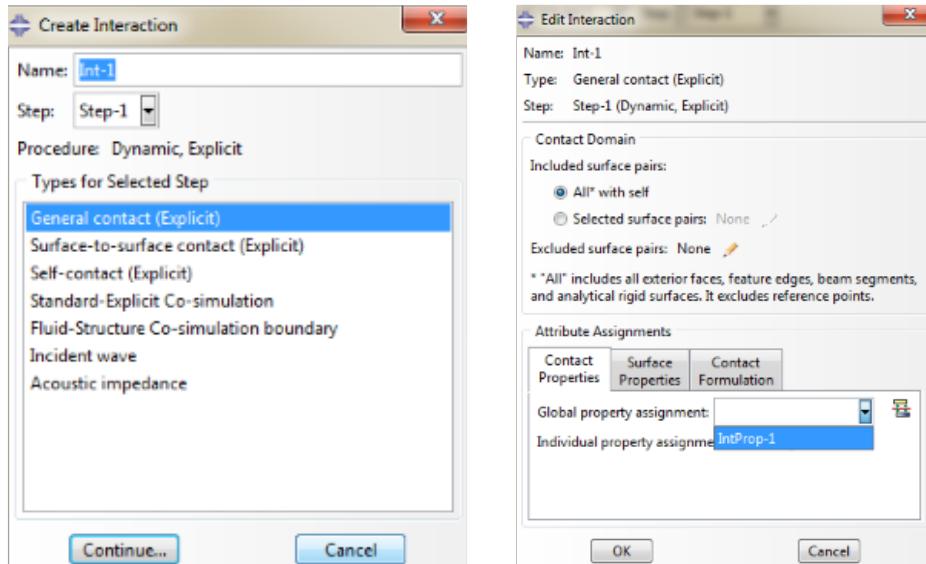


Fig. 12.13: The create interaction procedure

Usually, the working procedure follows the modules one by one from top to bottom but in the case of Coupled Euler – Lagrange analysis, the discretization of the model must be done before the load module, because the usage of the **Predefined Field** requires that at least the medium (if not the reference part as well) be meshed. Also, this tool is available only in the Load module.

In **Module: Mesh**, the whole model appears green, meaning that the mesh elements are hexahedrons, created through Structured technique. The solid part and the Eulerian parts will be meshed separately, having different properties.

For the solid part, the size of the cells will be 0.25 m, while for the Eulerian parts will be 0.5 m, both determined using **SR Seed Part** button, which will open the **Global Seeds** window. At the **Approximate global size:** the

user can write the dimension of the cells' size (Fig. 12.14). Click **LMB** on **OK** button to accept the dimensions of the mesh cells. Click **LMB** on **BR Done** button to end the seeding procedure.

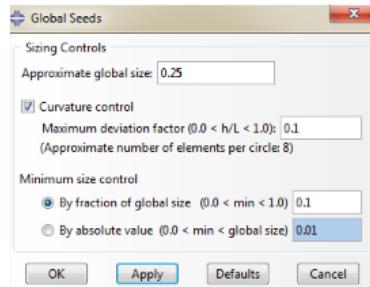


Fig. 12.14: Seed window

To set the type of mesh elements used, click **LMB** on **SR** **Assign Element Type**, select the part for which you want to assign the element type and click **BR Done**. This procedure will open an **Element Type** window. In the case of the solid part, the type of element to be used is C3D8R, which corresponds to **Element Library: Standard, Family Type: 3D Stress** (Fig. 12.15). Click **LMB** on **OK** button to accept the parameters. In the case of the Eulerian parts, the types of elements are EC3D8R, given by the settings **Element Library: Explicit, Family Type: Eulerian**. (Fig. 12.16). Click **LMB** on **OK** button to accept the parameters. Click **LMB** on **BR Done** button to end the elements type assignment.

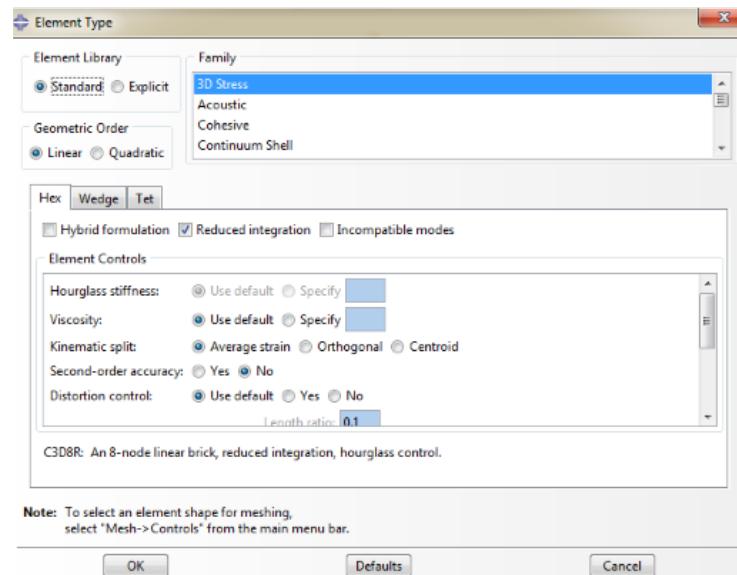


Fig. 12.15: The Element Type assignment for the solid part

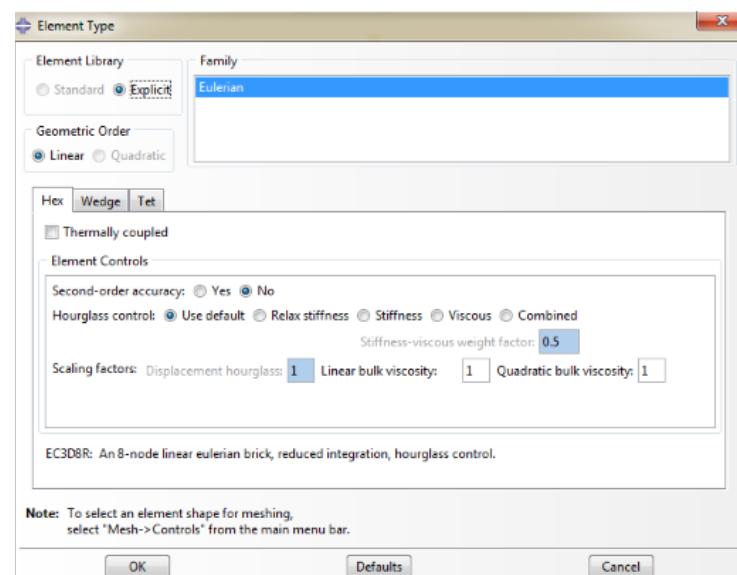


Fig. 12.16: The Element Type assignment for the Eulerian parts

In order to complete the meshing technique, go to **SR**  **Mesh Part Instance**, click **LMB** and select the whole model. Click **Done** in order to end the discretization. The result should be very similar to Fig. 12.17.

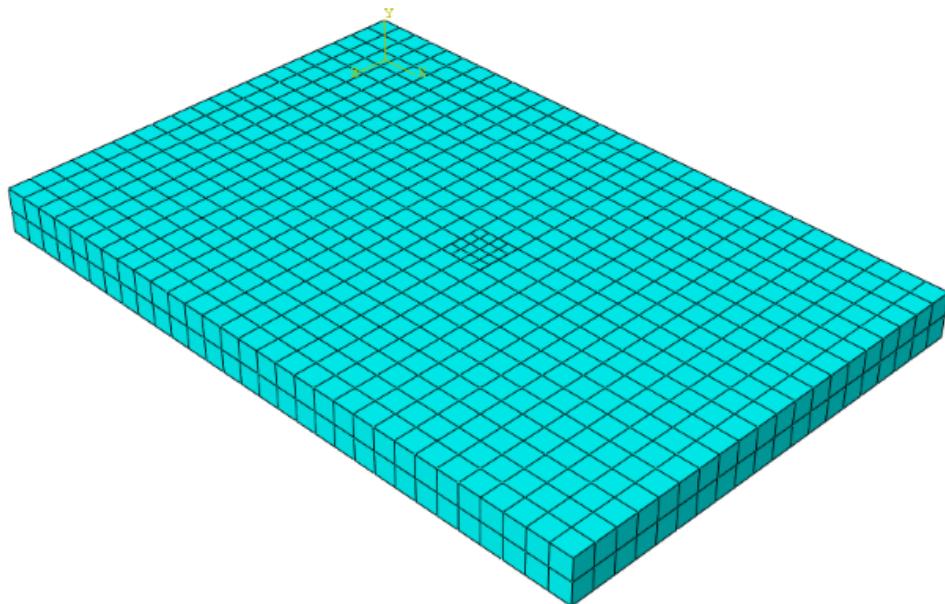


Fig. 12.17: The meshed model

After the model mesh has been defined, the **Module: ▾ Load** will be used, first to define the discrete field used for modelling the water behaviour in the given medium (using the Volume fraction tool) and then to define the boundary conditions and loads acting on the model.

To access the Volume fraction tool, the path is **Tools ► Discrete Field ► Volume Fraction Tool...** (Fig. 12.18). The user will be asked to select the Eulerian part instance (referring to the medium) and the reference part instance. This will open the volume Volume Fraction Tool window (Fig. 12.19). The options chosen for this example are **Accuracy: ⓁMedium** and **Material Location: ⓁInside reference instance** (this option will consider the existence of material inside the volume of the reference part). Click **LMB** on **OK** button to accept the parameters.

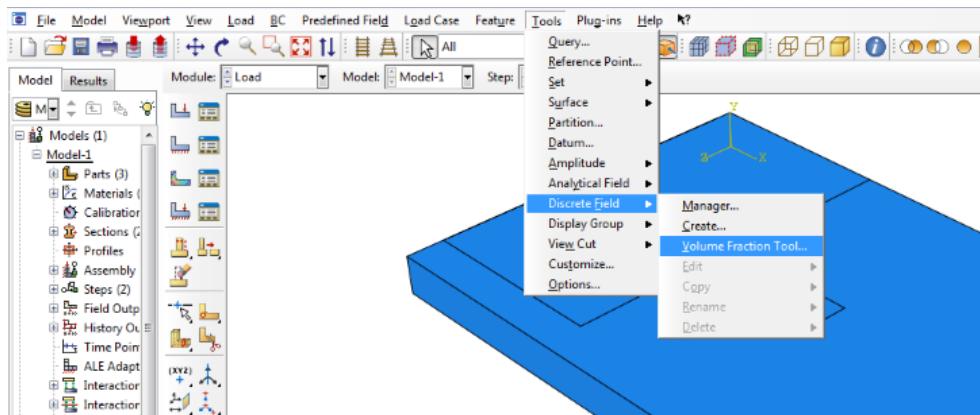


Fig. 12.18: The volume fraction tool path

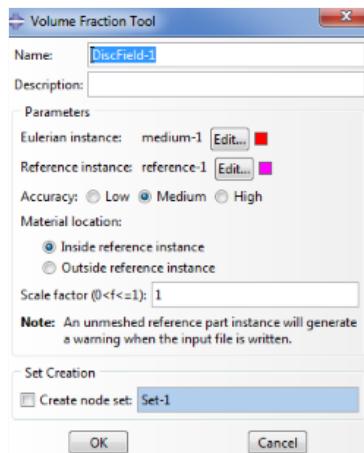


Fig. 12.19: The volume fraction tool window

To create a predefined field, the path is **Predefined Field** ► **Create...** (Fig. 12.20). This command will open the Create Predefined Field window, where the user can select the step in which the predefined field is created, the category of the predefined field and the type. For the Euler-Lagrange coupled model, the options chosen should be **Step:** **Initial**, **Category:** **Other**, **Types for Selected Step:** **Material assignment** (Fig. 12.21). Click LMB on

Continue ... button. It will be necessary to select the Eulerian part instance to which the predefined field should be assigned.

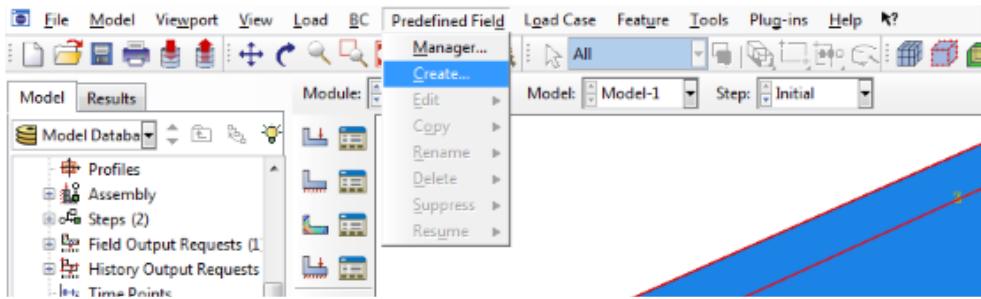


Fig. 12.20: The predefined field path

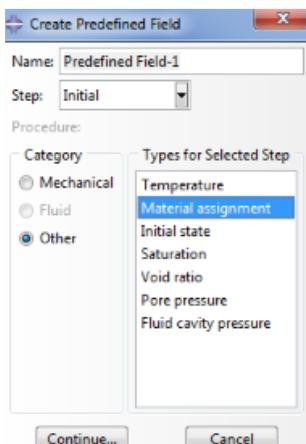


Fig. 12.21: The predefined field window

After selecting the Eulerian part instance, an Edit Predefined Field window will appear (Fig. 12.22). The options chosen for the coupled analysis are **Definition: ◎Discrete fields**, the name of the parts involved in the predefined field are shown and the discrete field created using the Volume fraction tool '*<Discrete field name>*'. Click LMB on **[OK]** button to assign the discrete field to the Eulerian medium.

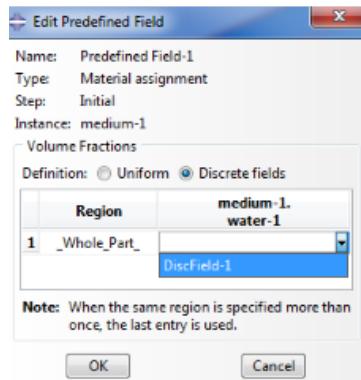


Fig. 12.22: The edit predefined field window

Next the definition of the boundary conditions and the loads will be created.

SR **Create Boundary Condition** button, which brings forth the Create Boundary Condition window: name this restraint “Fixing” and choose the following options: **Step:** **Initial**, **Category:** **OMechanical** and **Types for Selected Step: Symmetry/Antisymmetry/Encastre**. Clicking **Continue ...** will advance to the next step, where the user is prompted to select the regions for the boundary condition. Select the base of the solid cube (while hovering over them, they will appear thickened orange dots) and click **Done**. This will open an Edit Boundary condition, and the option for the solid cube is **ENCASTRE (U1= U2= U3= UR1= UR2 = UR3 = 0)**. Clicking the **OK** button will end the creation of the boundary condition.

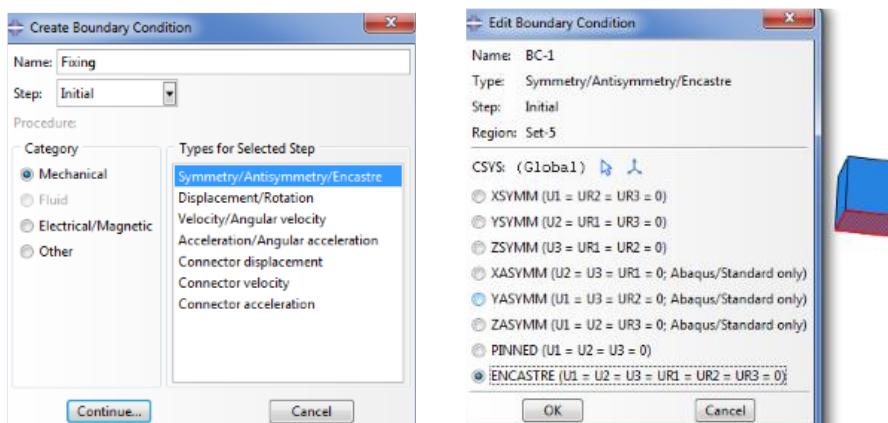


Fig. 12.23: Creating the Boundary Condition for the solid part: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used

To create the boundary conditions for the Eulerian medium, press **SR**  **Create Boundary Condition** button, which brings forth the Create Boundary Condition window: name this restraint “Euler z” and choose the following options: **Step: Initial**, **Category: Mechanical** and **Types for Selected Step: Displacement/ Rotation**. Clicking **Continue ...** will advance to the next step, where the user is prompted to select the regions for the boundary condition. Select the faces of the Eulerian medium perpendicular to z axis (while hovering over them, they will appear thickened orange dots) and click **Done**. This will open an Edit Boundary condition, and the option is U3 for the boundary condition on z axis. Clicking the **OK** button will end the creation of the boundary condition. Repeat the procedure for the faces perpendicular on y and x axes.

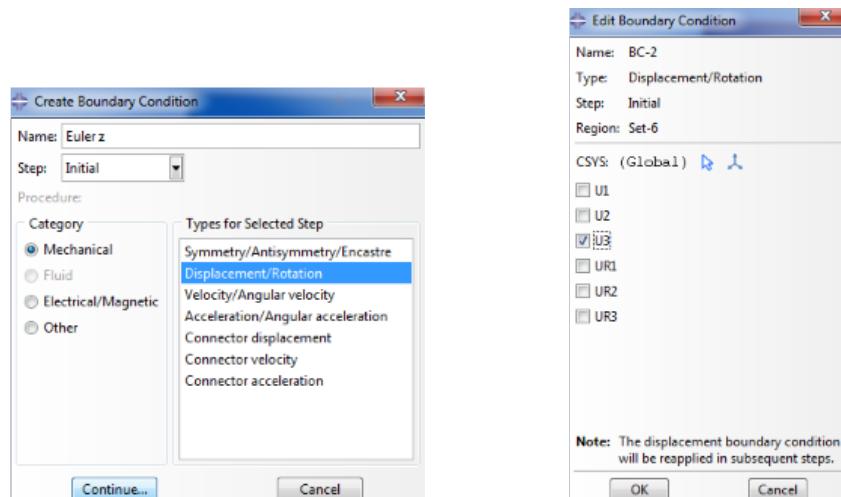


Fig. 12.24: Creating the Boundary Condition for the Eulerian medium part: left) Selecting the general boundary condition type and acting step; right) Selecting the exact boundary condition to be used

As load of interest, we will consider the self-weight of the system, using a gravitational acceleration applied to the whole model. Go to the **RS**  **Create Load** button, click it, and the Create Load window will appear. Name the load “g” for gravitational, make sure the set step is “Step-1” and choose **Category: ⓠMechanical** for the load category and **Types for Selected Step: Gravity**. Click **Continue ...** and a second window (Edit Load) appears in which it is required to provide the domain on which the gravity load should be applied and on the three directions its components. Therefore, as it can be observed, the region is the whole model, by default, and in the field near the Component 3 fill with the $9.81 \text{ (m/s}^2)$ value. Click the **OK** button to finish the sequence.

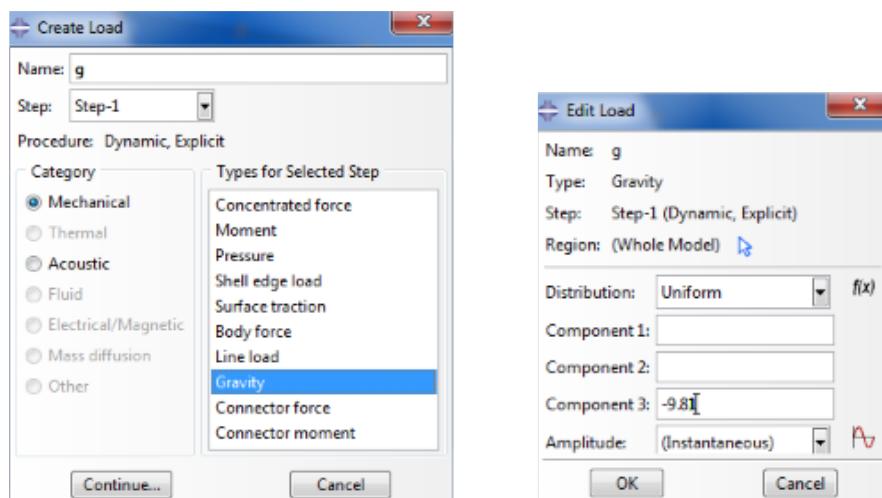


Fig. 12.25: Creating the Load: left) Selecting the general load type and acting step; right) Selecting direction on which the load acts

At this moment, the model is ready to be submitted to the calculation phase. Therefore, we advance to the **Module: Job**. Go to **SR**  **Create Job**. A Create Job window will appear in which the name shall be changed from the default “Job-1” to “CEL”. The source is the Model-1, which has been created. Click **Continue ...**. Under the Edit Job window, go to the Parallelization tab

and, if the case, select **Use multiple processors** and change to the number of cores your CPU have, in order to improve (reduce) the calculation period.

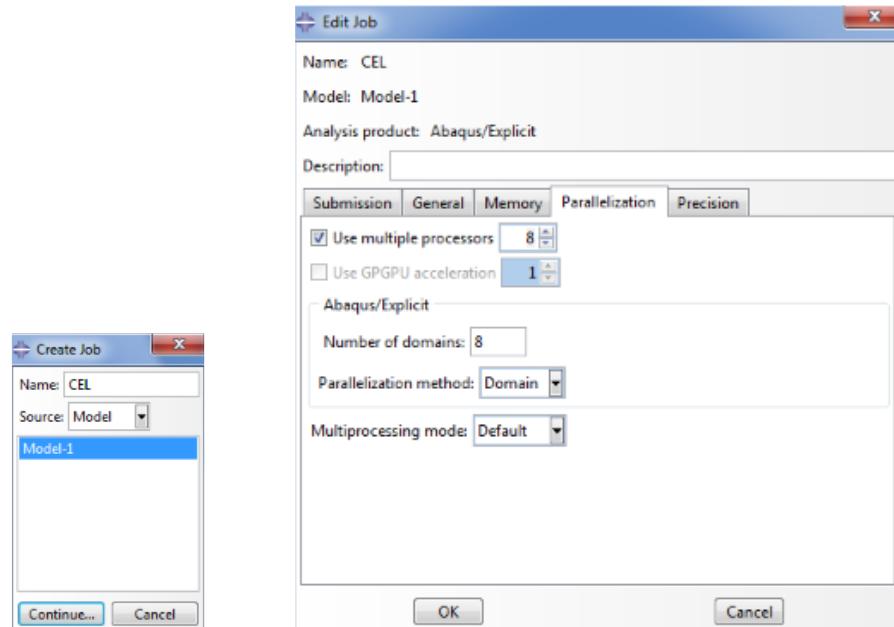


Fig. 12.26: The Create Job window and Edit Job

In order to submit the job to the calculation process, go to **SR**  **Job Manager**, make sure the job is selected and **LMB** click the **Submit** button.

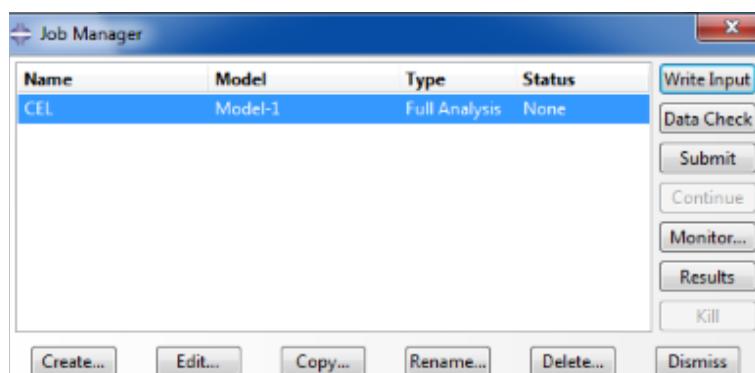


Fig. 12.27: The Job Manager window

On submitting the job to be calculated, the status will change from “None” to “Submitted”, and after finalization of the calculation process, will turn into “Completed”.

In order to visualize the result, in the Job Manager window, click the **Results** button. Now, the interface has moved to the post-processing part of the software, as it can be observed under the module section: **Module: Visualization**. Click the **SR  Plot Contours on Deformed Shape** and the model will provide both the contours of the variable and the deformed shape of the model (default scale is Auto-compute). It can be noticed that the Euler reference part is not coloured, this part being used only for determining the presence of the Eulerian material in the medium. In order to view the results more clearly, the user has to remove this part from the view. Also, to be able to view the behaviour of the fluid material rather than the behaviour of the whole medium, press **SR  View Cut Manager** (Fig. 12.28 ①). This will open the View Cut Manager Window (Fig. 12.29), where the user can select to view only the material from the Eulerian medium (**EVF_VOID**).

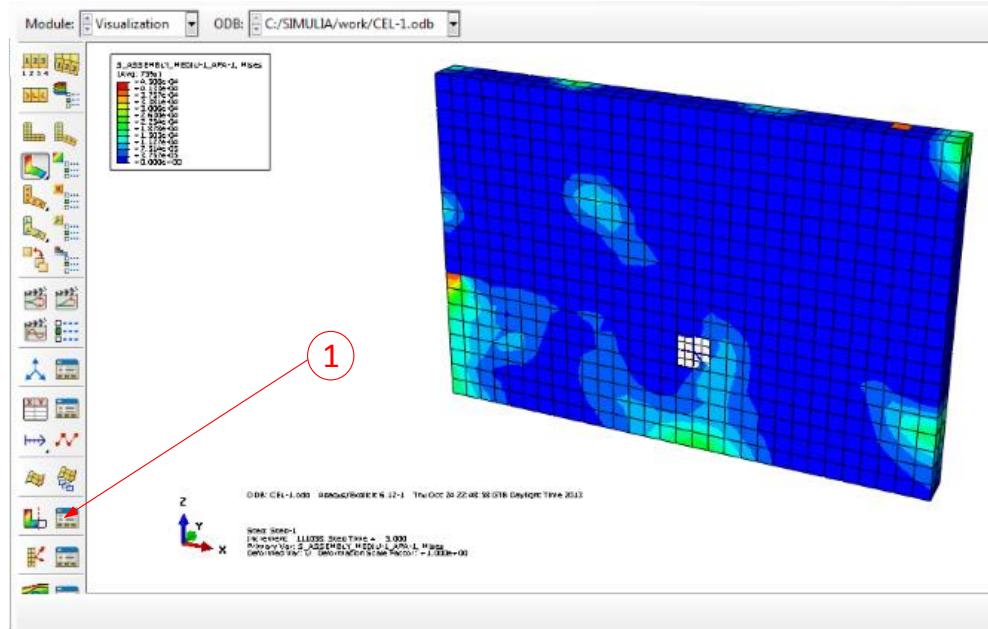


Fig. 12.28: The view cut manager

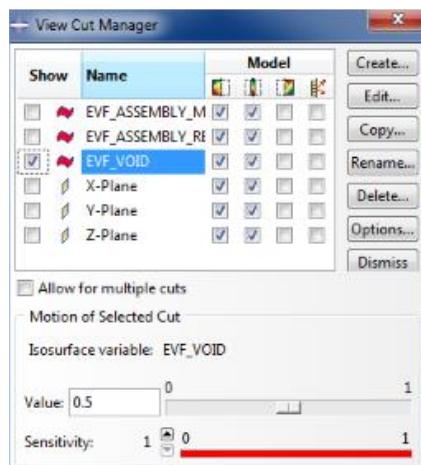


Fig. 12.29: The view cut manager window

12.5 Results interpretation

The results show the interaction between the solid and the fluid parts and the stresses induced in each part due to this interaction modelled through Euler-Lagrange coupling, the goal of this analysis. The stresses and displacements can be viewed in both parts, but not in the same time.

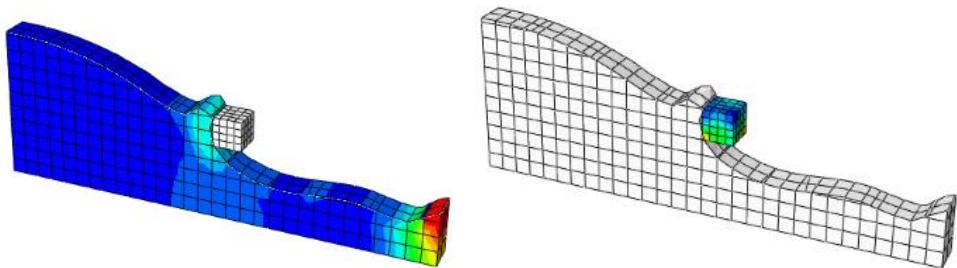


Fig. 12.30: The stresses induced in the fluid part (a) and solid part (b)

12.6 Things to remember

The model presented analyses the interaction between a solid part and a fluid one (having the properties of water). This model can be used to model high displacements models, where the classical Lagrangian formulation cannot show the true behaviour, mainly due to convergence problems.

In real case analyses, the most important part is the parameters determination for the model, a fairly difficult task due to the complexity of the tests needed.